Modeller Reference Manual

Version 15.0 Issue 2

LUSAS
Forge House, 66 High Street, Kingston upon Thames,
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999
Fax +44 (0)20 8549 9399
Email: info@lusas.com
http://www.lusas.com

Distributors Worldwide

Table of Contents

Chapter 1 : Introduction

Welcome to LUSAS

LUSAS is one of the world's leading structural analysis systems. By choosing to use LUSAS you are joining a large worldwide community of engineers who use LUSAS everyday to solve a wide range of engineering analysis problems.

Before starting to use LUSAS it is strongly recommended that you read the Getting Started Guide in its entirety. It provides an essential overview of the LUSAS Modeller user interface and of the processes involved in creating models, running analyses and viewing results. The guide also lists the LUSAS documentation available for you to use and provides information on additional resources including how to obtain technical support.

About LUSAS

The LUSAS system uses finite element analysis techniques to provide accurate solutions for all types of linear and nonlinear stress, dynamic, and thermal/field problems. The two main components of the system are:

- **LUSAS Modeller** - a fully interactive graphical user interface for model building and viewing of results from an analysis.
- **LUSAS Solver** - a powerful finite element analysis engine that carries out the analysis of the problem defined in LUSAS Modeller.

The Application Products

LUSAS is made available as different application products with the user interface tailored specially to the needs of a particular application. For example bridge engineers will see facilities provided specifically for the analysis of bridge structures. Such facilities are normally conveniently grouped together inside a product application menu, such as Bridge, Civil, Composite etc, that is loaded between the Utilities and Window menu items in the main menu.

The application products include:

- **LUSAS Bridge** – for bridge engineering analysis, design and assessment.
- **LUSAS Civil & Structural** – for civil, structural, nuclear, seismic, geotechnical and offshore engineering.
- **LUSAS Analyst** – for automotive, aerospace, defence, manufacturing, and general engineering analysis.
- **LUSAS Composite** – for engineers designing composite products or components.
- **LUSAS Academic** – available to academic establishments for teaching and research use and allows use of all of the above products together with most of the options available for those products.

LUSAS Product Options

In addition to LUSAS software being provided as particular individual application products, product options are also available which extend the analysis capabilities of these products. The following options are available for all products:

- **Nonlinear** - Nonlinear stress analysis is becoming increasingly important with designers employing a wider variety of materials in a multitude of different applications. The Nonlinear option, rightly regarded as the leader in nonlinear analysis, provides the very latest powerful techniques for solving problems having either material, geometric or boundary nonlinearity.
- **Dynamics** - Straightforward modal dynamics problems can be solved using Interactive Modal dynamics (IMD) techniques provided in all LUSAS products. The Dynamics Option contains the facilities required to solve a wider range of dynamic problems in both the time and frequency domains. By combining the Dynamics and Nonlinear options both high and low velocity nonlinear impact problems can be solved using either implicit or explicit solution techniques.
- **Thermal/Field** - The Thermal / Field option contains extensive facilities for both simple and advanced steady state, and transient thermal / field analyses. By combining the Thermal / Field option with other appropriate options, heat transfer due to conduction, convection and radiation can be analysed. In addition, the effects due to phase change of material may also be included.
- **Heat of Hydration** - Allows modelling the heat of hydration of concrete. The heat of concrete hydration can be computed during a thermo-mechanical coupled analysis and the temperatures and degree of hydration can be read into a structural analysis.
- **Fast Solvers** - This option includes additional solvers. The Fast Multifrontal Direct Solver can provide solutions several times faster than the standard Frontal Direct Solver for certain analysis problems. The Fast Multifrontal Block Lanczos Eigensolver can, similarly, return results several times faster than the standard Frontal Eigensolvers for certain problems. The complex eigensolver provides efficient solutions for large-scale damped natural frequency problems.
- **IMDPlus** - The IMDPlus option extends the Interactive Modal Dynamics (IMD) techniques provided in all LUSAS products. Whilst IMD models a single loading

event in a single direction, IMDplus allows multiple loading events with more advanced loading conditions to be solved. IMDPlus is used for two primary uses: seismic response analysis of 2D and 3D structures subjected to acceleration time histories of support motion, and for the analysis of 3D structures, such as bridges, subjected to constant moving vehicle or train loads.

The following options are available for LUSAS Bridge only:

- **Vehicle Load Optimisation** - Used to identify critical vehicle loading patterns on bridges and apply these loading patterns to analysis models. It reduces the amount of time spent generating models and leads to more efficient and economic design, assessment or load rating of bridge structures.
- **Rail Track Analysis** - Carries out rail track-structure interaction analysis to the International Union of Railways Code UIC 774-3. It allows models to be built automatically from data defined in MS Excel spreadsheets, an analysis run, and results produced in spreadsheet or results file formats.

What's new

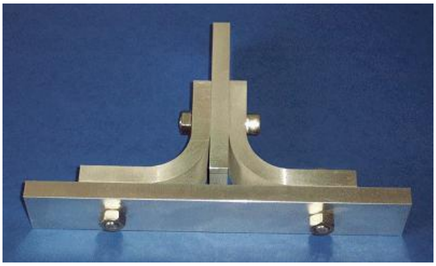
When you run LUSAS Modeller you will be presented with information that links to a 'What's new' page. This will tell you what has changed since the last major version and also covers any interim releases. It will be of particular interest to existing users upgrading from an earlier version of LUSAS.

What is Finite Element Analysis?

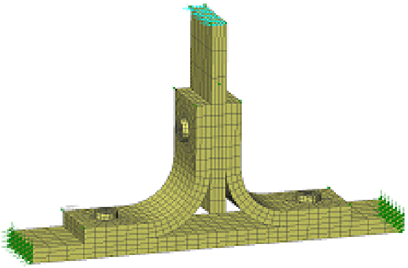
Until the advent of computers, the only way to find the answer to the engineering question "What would happen if I did this to my new design?" was to build a prototype and carry out the necessary tests. Today computers allow designs to be assessed much more quickly and easily. Evaluating a complex engineering design by exact mathematical models, however, is not a simple process.

Since we cannot calculate the response of a complex shape to any external loading, we must divide the complex shape up into lots of smaller simpler shapes. These are the finite elements that give the method its name. The shape of each finite element is defined by the coordinates of its nodes. Adjoining elements with common nodes will interact.

Engineering Problem



Finite Element Model



The real engineering problem responds in an infinite number of ways to external forces. The manner in which the Finite Element Model will react is given by the degrees of freedom, which are expressed at the nodes. For example, a three-dimensional solid element has three degrees of freedom at each node representing the three global directions in which it may move.

Since we can express the response of a single Finite Element to a known stimulus we can build up a model for the whole structure by assembling all of the simple expressions into a set of simultaneous equations with the degrees of freedom at each node as the unknowns. These are then solved using a matrix solution technique.

For a mechanical analysis, once the displacements are known the strains and stresses can be calculated. For a thermal analysis, the gradients and fluxes can be calculated from the potentials.

What Help and Documentation is Provided?

Comprehensive documentation is provided with LUSAS. Some is available in the form of printed manuals, whilst some is only available in electronic format.

The documentation includes:

- ❑ **Installation Guide**
- ❑ **Getting Started Guide**
- ❑ **Modeller Reference Manual**
- ❑ **Examples Manual**
- ❑ **Application Examples Manual (Bridge, Civil & Structural)**
- ❑ **Application Manual (Bridge, Civil & Structural)**
- ❑ **Autoloader Reference Manual**
- ❑ **IMDPlus User Manual** (includes worked examples)
- ❑ **Rail Track Analysis User Manual** (includes worked example)
- ❑ **Element Reference Manual**
- ❑ **Solver Reference Manual**
- ❑ **Theory Manual (Volume 1 and 2)**
- ❑ **Verification Manual**
- ❑ **CAD Toolkit User Manual**
- ❑ **LUSAS Programmable Interface (LPI)**
- ❑ **Glossary**

Installation Guide

- Details of installing LUSAS software for various licensing options.
- Available in PDF and printed form.

Getting Started Guide

- Contains a brief overview of LUSAS.
- Available in PDF and printed form.

Modeller Reference Manual

- Provides detailed reference material for modelling and results viewing with LUSAS Modeller.
- Provided in on-line help format and also available in PDF and printed form.

Examples Manual

- Contains general worked examples to help you get up to speed with modelling, analysis and viewing of results for a range of different analysis types.
- Available in PDF and printed form.

Application Examples Manual (Bridge, Civil & Structural)

- Contains application specific worked examples to help you get up to speed with modelling, analysis and viewing of results for a range of different analysis types.
- Available in PDF and printed form.

Application Manual (Bridge, Civil & Structural)

- Describes the bridge, civil and structural application specific features of LUSAS and their uses.
- Available in PDF and printed form.

Autoloader Reference Manual

- Provides detailed reference material for Autoloader, a bridge loading optimisation module for use with LUSAS.
- Available in PDF form.

IMDPlus User Manual

- Contains details of how to carry out multiple loading events with advanced loading conditions for two main uses: seismic response analysis of structures subjected to acceleration time histories of support motion, and for the analysis of 3D structures, such as bridges, subjected to constant moving vehicle or train loads.
- Provided in on-line help format and also available in PDF and printed form.

Rail Track Analysis User Manual

- Provides detailed reference material for the Rail Track Analysis option which permits track/bridge interaction analysis to the International Union of Railways Code UIC 774-3.
- Available in PDF form.

Element Reference Manual

- Contains full element specifications. This is the place to go to find out which functionality your elements support and what output you will obtain from your element selection.
- Provided in on-line help format and also available in PDF and printed form.

Solver Reference Manual

- The data files required by the LUSAS Solver can be edited directly with a text editor. This manual contains full details of the data syntax supported by LUSAS Solver.
- Available in PDF and printed form.

Theory Manuals (Volume 1 and 2)

- These contain more detailed theoretical information for the more experienced user. They cover topics specific to LUSAS and where appropriate list references to other publications. The topics covered include:
    - Analysis procedures including: linear, nonlinear, dynamics, eigenvalue extraction, modal analysis, all forms of field analysis, fourier analysis and superelement analysis.
    - Geometric nonlinearity.
    - Constitutive material model formulations.

    - Loads and boundary conditions with particular reference to general load types, constraint equations, slidelines and thermal surfaces.
    - More complex post processing calculations, including nodal extrapolation and calculation of Wood Armer reinforcement moments.
    - Element formulation theory.
- Available in PDF and printed form.

Verification Manual

- A manual of LUSAS testcase examples benchmarked against known solutions.
- Available in PDF form only.

CAD Toolkit User Manual

- Describes interfaces to LUSAS involving the use of external pre- and post-processing packages.
- Provided in PDF form only.

LUSAS Programmable Interface (LPI)

- Provides information for application programmers wishing to customise the LUSAS environment or interface LUSAS with other applications.
- Provided in on-line help format only. This can be accessed from the Start > All Programs > LUSAS > LUSAS Help menu.

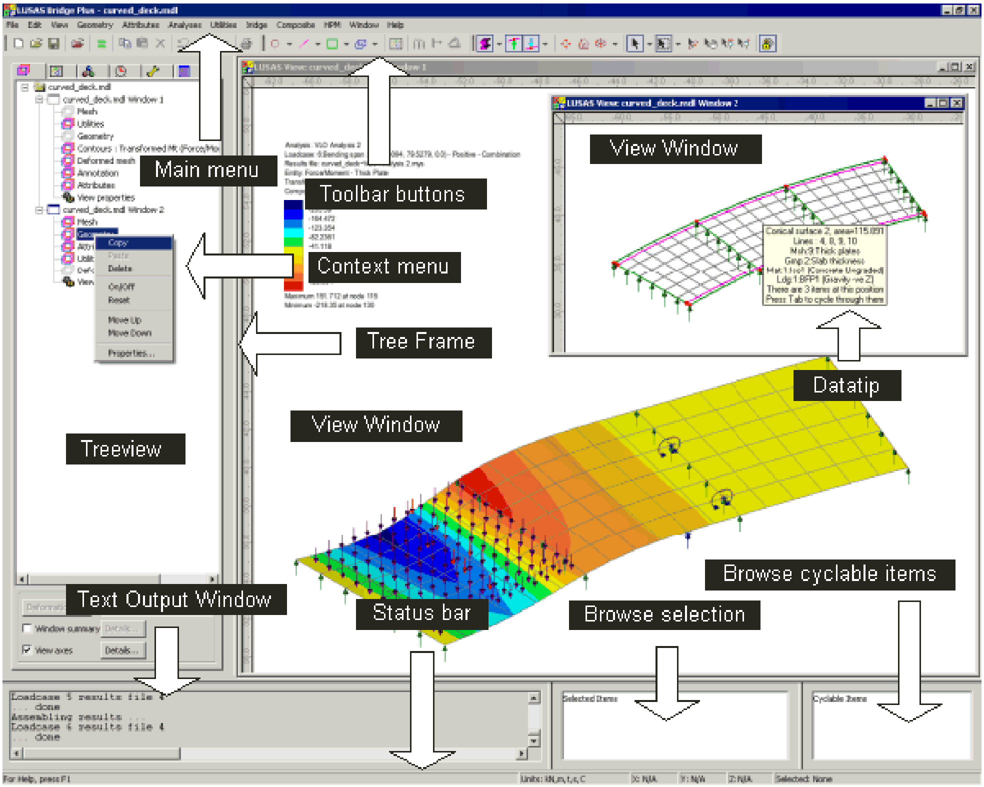
Glossary

- Contains definitions of general terms used in all manuals.
- Provided in on-line help format. Included in the *Modeller Reference Manual.*

Chapter 2 : Using Modeller

LUSAS Modeller User Interface

LUSAS Modeller is an easy to use Windows-based finite element modelling system. The principal regions and components of its user interface are shown on the following image and are described below.



LUSAS Modeller Interface

Main Menu and Toolbars

The main menu and associated toolbars which contain toolbar buttons provide the means to define model related geometry and other data. On initial start-up of LUSAS Modeller the Main, Define and View Toolbars are displayed. All toolbars can be shown, hidden, or customised, using the **View> Toolbar** menu item. When a modelling session is completed the current toolbar settings are saved and reloaded the next time Modeller is used. User defined toolbars and buttons can also be added to the user interface. Actions are assigned to user defined buttons using the scripting language.

View windows

A **View Window** is where each model is developed and manipulated. Any number of view windows can be created with each displaying a unique view of a model. The image in a view window is built up from several drawing layers as listed in the layers panel in the Tree Frame. Each layer draws over the top of the one before it.

Tree Frame

The **Tree Frame** has six tabbed panels each with its own **Treeview**. The Treeviews provide access to model and results data and show a list of objects of a particular type. The Treeview panels comprise: Layers, Groups, Attributes, Analyses, Utilities and Reports.

By default a single Tree Frame is displayed. Multiple Tree Frames can be utilised from the **View> Tree Frame** menu item. Treeviews can then be dragged and dropped between Tree Frames as required. This is often used to aid copying and pasting of attributes between Treeviews.

Treeviews

The Treeviews provide access to all defined model data and results data, and show a list of objects of a particular type:

- The Layers Treeview controls the display of selective model data and results data to a view window.
- The Groups Treeview is used to store selected user-defined collections of objects (geometry, nodes or elements) under a collective name.
- The Attributes Treeview contains information relating to the model; the element type and discretisation on the geometry; section properties and thicknesses; the materials used; how the model is supported or restrained; and how the model is loaded.
- The Analyses Treeview shows all analyses defined; loadcases defined including **analysis control loadcases** defined during the modelling stage; **results loadcases** containing solutions for results processing; loadcase combinations and envelopes; and IMD and Fatigue calculations.

- The Utilities Treeview contains utility items used in the definition of model geometry or attributes, or to control an analysis, or to provide a particular functionality, such as to define a load combination or produce a report for example.
- The Reports Treeview contains a user-defined folder structure of reports, chapter and image entries to allow a report to be generated in a variety of formats.

The Treeviews use drag and drop functionality. For example, an attribute in the Attributes Treeview can be assigned to geometry in a View Window by dragging the attribute onto a previously selected **object**. Each object in a Treeview has its own functionality. This can be invoked using its own context menu, the main menu or the toolbar buttons. Treeview data can also be copied and pasted between Treeviews.

Text Output Window

The Text Output Window displays messages and warnings during a modelling session.

By default the Text Window appears at the bottom of the View window just above the Status Bar. It may be resized using the cursor and may be undocked or hidden from view. The **View> Text Output** menu item may be used to hide or show the Text Window. The Text Window context menu allows the Font to be defined and the output to be directed to a log file. By default the selection mode is set to select lines of text. This may be changed to select characters by changing to Character select mode but subsequent output to the text output window will be slower.

Note: When an error message or warning relating to a particular object is written to the text output window, extra information is usually available by double-clicking on that line of text. In doing so, a popup window provides options to help identify the offending area of the model. For example, if a Modeller error refers to an assignment on line 25 it may be selected, moved to the centre of the screen, scaled to fill the screen, have its properties displayed, or be identified by an annotation arrow or a temporary indicator. Similarly, if a Solver error or warning refers to a particular numbered element, the popup window will help you find that element quickly and easily.

Browse Selection

This window is not displayed by default but can be viewed using the **View> Browse Selection** menu item. Once visible it will contain a list of all currently selected items which may then be individually deselected. Any objects listed in this window can be located on the model by using the **Find** menu item on the context menu for the object. The location of the assigned attribute will be identified by a temporary indicator of animating concentric shrinking squares.

Browse Cyclable Items

This window is not displayed by default but can be viewed using the **View> Browse Cyclable Items** menu item. Once visible it will contain a list of all cyclable items. These may then be individually selected or deselected. Note that once an object is selected other selectable items at the same position within the View window can be cycled using the **Tab** key or by reselecting with the left hand mouse button at the same position within the View window.

Browse Selection Memory

This window is not displayed by default but can be viewed using the **View> Selection Memory> Browse Memory Selection** menu item. Once visible it will contain a list of all items currently in Selection Memory which may then be individually deselected.

Note: Any objects listed in this window can be located on the model by using the Find menu item on the context menu for the object. Its location will be identified by a temporary indicator.

Status Bar

The Status Bar displays progress messages and help text during a modelling session, the model units, the current cursor position in model units (if the model is displayed in an orthogonal plane) and the item or number of items in the current selection. The **View> Status Bar** menu item may be used to hide or show the Status Bar.

Data tips and Tool tips

Data tips and tool tips provide basic information about whatever is under the cursor. Datatips generally report information relating to the model geometry, attributes and assignments etc. Tool tips report on uses of toolbar buttons or expected input for grid cells etc. Data tips are invoked when the cursor is hovering over a model feature within the View window (where it reports upon all manner of model data). Tool tips are invoked either when the cursor is hovering over a toolbar button or a grid cell on a dialog that is expecting dimensional input (where the tool tip reports the units that are expected). For the former, if more than one item is present at the location on the screen, the **Tab** key can be used to cycle between all possible selectable objects, with the data tip updated each time. As additional feedback, if a object is selected, dynamic selection is invoked to highlight each object as it becomes the focus. Thus with two intersecting lines or overlapping surfaces, it is clear which one is being displayed in the data tip.

 Pressing the **Enter** key whilst a data tip is active adds the current object to the selection. This provides an easy way to select a specified "one of many" objects at a given location.

Context Menus

Although commands can be accessed from the main menu, pressing the right-hand mouse button with an object selected in the View Window usually displays a context menu which provides access to relevant operations. In addition, most items in the various Treeview panels also have a context menu which provides access to additional functionality such as editing of data, control of visibility, visualisation of assignments, and selective control of results plotting on selected attributes.

Properties

General information relating to a model is presented in property dialogs. Properties may relate to the whole model or the current window, or a single geometric feature - in fact most objects have properties. To view an object's properties, select it, press the right mouse button, then choose **Properties** from the context menu.

Getting Help

LUSAS contains a comprehensive Help system. The Help consists of the following:

- ❑ The **Help** button  on the Main toolbar is used to get context-sensitive help on the LUSAS interface. Click on the **Help** button, then click on any toolbar button or menu item (even when greyed out).
- ❑ **Help Topics** accessed from the **Help** menu provides access to the Help files. They include the *Modeller Reference Manual*, reference help files such as the *Element Reference Manual* and access to other manuals that are available in PDF format such as the Examples Manuals, Theory Manuals and Solver Reference Manuals.
- ❑ Dialogs normally include a **Help** button which provides specific information for that dialog and links to related topics.

Using LUSAS

Carrying out a structural analysis or assessment with LUSAS involves three main stages:

- ❑ **Modelling**
- ❑ **Running the analysis**
- ❑ **Viewing the results**

Modelling is sometimes referred to as 'pre-processing' and viewing and manipulating the the results is sometimes referred to as 'post-processing'. A general overview of the modelling , analysis and results processing stages are provided below.

Model Types

Two types of model can be created in LUSAS:

- ❑ **Feature-based geometry models** - these are based on features (such as points, lines, surfaces and volumes) that are defined inside LUSAS, or are created from imported geometry and require the definition of mesh objects such as elements with their associated nodes.
- ❑ **Mesh-only models** - these comprise only mesh objects (elements and their associated nodes, edges and faces) and can only be created by importing only those types of LUSAS or third party data that are supported.

Modelling

Models are created in view windows. Modelling involves creating a geometric representation of a structure and defining its characteristic behaviour in terms of its physical properties such as material, loading, and support. A model will consist of **Geometry** (Points, Lines, Surfaces and Volumes) to which **Attributes** such as (Mesh type, Materials, Loading, Supports, Mesh, etc.) have been assigned . Treeview panels provide access to defined model data.

- **Geometry**

  Within LUSAS Modeller the geometry of the model is defined using points, lines, surfaces and volumes. A volume needs surfaces to enclose it and to define its boundary. Similarly a surface needs lines to form its perimeter and lines need points to define their ends. The shape of a surface between its boundary lines, and the shape of a line between its end points can be simple (straight lines, flat surfaces) or complex, depending on the manner in which it was created.

  Many different tools exist within LUSAS Modeller to aid the creation of geometry. In particular the geometry can be entered numerically, in terms of coordinates, by cursor (using the mouse) or by copying, splitting, intersecting and transforming previously existing parts of the model. Geometry can also be imported from third-party interface files.

- **Attributes**

  Aspects of the behaviour of parts of the model, or the external factors which are imposed on it, are referred to as attributes. Within LUSAS Modeller, there are several types of attribute – each representing a particular type of behaviour. For example materials, loading and support are all attributes, but are quite different from each other. Within each, there are further sub-divisions, for example there are isotropic materials, anisotropic materials, and orthotropic materials, among others.

  In each case an attribute is first created, and then subsequently attached to all or part of the model. This attachment process is known as assigning. Thus a material can be assigned to a line. Once assigned, the line takes on the properties of the material until further notice. Assignment often involves specification of additional information that does not define the attribute or the geometry, but rather defines how one is applied to the other. This includes directions, scale factors and similar.

  A special attribute, called a geometric attribute, is used when some or all of a model is comprised of lines or surfaces. In this situation, LUSAS Modeller needs to know the cross-sectional appearance and behaviour of lines, and also how thick the surfaces are. Geometric attribute assignments provide this information, and therefore form part of the model's appearance. Models comprised entirely of volumes do not need geometric assignments.

  Many different tools exist within LUSAS Modeller to give interactive feedback on the nature of assigned attributes. This process is generally termed visualisation. Visualisation usually takes the form of coloured infills, arrows, textual labels or similar.

- **Meshing**

  Points, lines, surfaces and volumes allow the exact smooth geometry of the problem to be defined or imported. However, to solve the problem, the model must be broken down into nodes and elements. This process is known as meshing and the collective term for all the elements and nodes, once created, is the mesh. Special attributes, called mesh attributes, can be created and assigned to geometry in the same way as other attributes. They define the type and number of nodes and elements that will be used to represent each part of the geometry.

A consequential advantage of this approach is that the density of the mesh can easily be changed without rebuilding the geometry or reassigning any attributes. Simply modifying a mesh attribute automatically changes the mesh density in any part of the model where it is assigned.

Similarly, the element type can be changed in exactly the same way. Different element types are suitable for different kinds of analysis, and the system provides an easy way to change from one to another.

Generally LUSAS Modeller automatically and incrementally updates the mesh whenever a change is made to any part of the geometry or any mesh attributes. However these automatic updates are not allowed when results are being displayed, and they can also be switched off if at other times if required, for example to save time. This is known as locking the mesh. When the mesh is locked it will only be updated when specifically requested.

- **Loadcases**

  In a linear analysis, it is often convenient to gather certain types of applied loading together under a common loadcase name such as dead load or live load. Each loadcase is solved separately and the results are available separately. Results can be combined together during the results viewing stage if required.

  In a nonlinear analysis, LUSAS loadcases are equivalent to analysis stages. Control parameters can be set for each loadcase and within each loadcase, the loading, supports, or type of analysis can be changed.

- **Utilities**

  Some kinds of analysis require or create extra data that is similar to attributes, but cannot be assigned to geometry. LUSAS Modeller refers to such data as utilities. Tendon profiles, reference paths and the values used to plot a graph are examples of utilities.

- **Controls**

  Finally, for analyses other than simple linear static, it is sometimes necessary to control the analysis process itself. LUSAS Modeller uses controls for this purpose. Controls are either nonlinear, transient, eigenvalue or fourier in nature.

- **Further Modelling Information**

  Browse the LUSAS Modeller interactive online help system, or see the *Modeller Reference Manual* for detailed information about any of the above.

  A large selection of tutorial examples are available to help you to get to know LUSAS and it is strongly recommended that you work your way through a selection of the examples to get a feel for the system before starting on your particular problem. See the *Examples Manual* and/or the *Application Examples Manual (LUSAS Bridge and LUSAS Civil & Structural).*

Running the Analysis

Once the modelling process is complete, the model is passed to LUSAS Solver for analysis. LUSAS Solver will then create results that are automatically loaded back into LUSAS Modeller for viewing and optional additional post-processing.

Viewing the Results

Results are available for individual loadcases and can be viewed in many different ways. Results can, if desired, be combined with attribute visualisation to make a plot, for example, that shows the applied loading and the resulting deformations.

This is just a selection of some of the results facilities that are available:

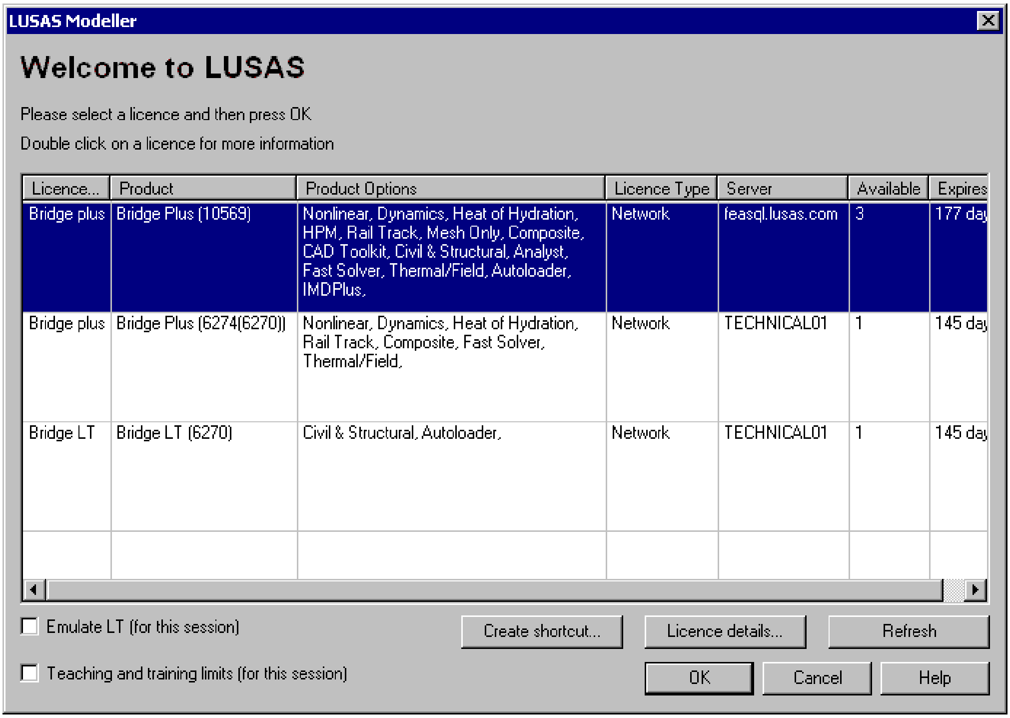
- **Averaged (also known as smoothed) contours**
- **Unaveraged (also known as unsmoothed) contours**
- **Deformed mesh**
- **Wood-Armer reinforcement calculations**
- **Slicing a plane through a solid**
- **Slicing a line across a surface**
- **Yield markers**
- **Graphs**
- **Vectors of stress, displacement, reactions, etc.**
- **Animations of the above, either through time or in response to excitation.**
- **Combinations and envelopes of results.**

Modeller Licence Selection

When running LUSAS, the Modeller licence selection dialog lists all software products that are available for selection, along with details of the licenced product options, licence type, server name, number of available licences and the number of days left until expiry. Invalid licences (those that have already been completely taken by others prior to the display of this dialog, or are unavailable for some other reason) are greyed out. Licences expiring within 14 days are displayed in red.

The licence selection dialog is always displayed on start-up of LUSAS unless it has been disabled by unchecking an option on the Licencing page of the LUSAS Configuration Utility, or unless there is only one licence available to LUSAS.

Use the Modeller licence selection dialog to select a licence for use. Use the LUSAS Configuration Utility to add licences to the list of those available.



License order and usage

Standalone or network licences are 'tumbled' based upon Configuration Utility settings. The order of the licence types listed on the Modeller licence selection dialog follows the order of the keys set on the Licensing page of the LUSAS Configuration Utility. If the display of the main licencing dialog has been disabled the first licence in the list of those available will be used in preference to all others. As licences are used the number of licences available for others to use is updated accordingly. When LUSAS Modeller is run the system will work its way down the list of licences until a valid and available licence is found.

Creating shortcuts

- The **Create shortcut...** button provides the means to tie a licence type to a shortcut used to run **Modeller** or **Solver** and with the choice to **Emulate LT** or use **Teaching and Training limits**. It enables LUSAS Modeller or LUSAS Solver to run without having to select a licence type each time (unless that licence has already been used).

  Example of shortcut created:

```
 <LUSAS Installation folder>\Programs\lusas_m.exe
LICNAM=[SENTINELLM,TECHNICAL01,6274(6270)]
```

Licence details

- By selecting a licence and then selecting **Licence details** additional information such as the product options, the licence version, the Key ID and the licence expiry date can be viewed.
- Selecting a licence that is invalid and clicking the licence details button will yield extra details as to why the license in unavailable.

Refresh

- **Refresh** simply updates the numbers of licences available.

Emulating LT Behaviour

- When **Emulate LT** is selected it restricts the user interface to that of an LT licence, even though the selected licence may be a standard or plus licence. This may be of use when training staff who are new to LUSAS.

Teaching and training limits

- When **Teaching and training limits** is selected no licence is taken. LUSAS can be used as normal for this session only but with the restricted model size, node, element and loadcase limits of the Teaching and Training version.

Creating a New Model

Creating a new model requires a model filename to be entered and a **Working folder** to be defined where the model will be saved. Files relating to a model (such as results files and intermediate files) can be saved beneath a model folder. See **Model File Locations** for details. A short model file name is recommended since this is also used to create folders and filenames that contain associated model data. If the path to a file is too long, the Windows path limit of 260 characters may be exceeded. **See File and Folder Naming in LUSAS** for more information.

Model details such as a model title and a job number can be optionally defined. A set of consistent modelling units must be specified for the model. These can be later updated in the Model Properties dialog, however it is recommended these are not changed after the model is created. Timescale units can also be defined.

Choosing an Analysis Type

When creating a new model it can be defined as being for a Structural, Thermal, or Coupled (dependent upon the licence key) analysis. This tailors the interface for the type of analysis to be carried out and also defines the class of model. The addition of a general thermal analysis to an existing structural base analysis, or the addition of a general structural analysis to an existing thermal base analysis will change the user interface accordingly.

Choosing a Startup Template

By selecting a startup template, useful predefined attributes can be setup in the model. Previously defined startup templates (scripts) can be made available for selection using the button alongside the Startup template drop-down list. These can then be selected from the drop-down list.

Specifying a Vertical Axis

The vertical axis is used to orientate the model, define a default gravity loading direction, and the vertical axis and orientation of particular element types and library items such as steel sections). It may be changed subsequently using the **Utilities> Vertical Axis** menu item.

Modelling Units

Model units are specified as part of the creation of a new model and are reported at all times in the status bar at the bottom of the Modeller window. Once set, for all dialogs with grid cells permitting dimensional input, the units expected are displayed as a tooltip when the cursor is hovered over the input cell.

Timescale units are related to model units and dictate how time-based values are displayed on dialogs during modelling, and how they are output when processing results.

Input

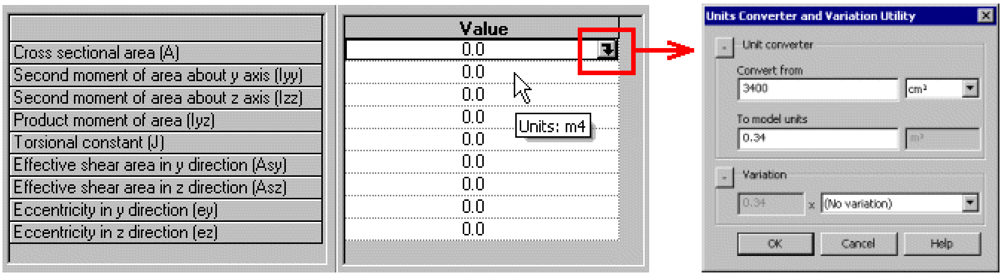
Metric units may be input in UK/US decimal point, or European comma formats. When Imperial modelling units have been set coordinates and lengths can be input in feet and inches, instead of just feet or just inches. For example, a distance of 5 feet, 11 and a half inches can be input and output now as 5:11.5 (or as 5'11.5"), rather than either 71.5" or 5.92'.

Expressions

Expressions may be entered anywhere in LUSAS Modeller where numbers may be input. For example, in point definition, it is possible to enter **3+10**, **4*6**, and **5-1** as valid coordinates. All arithmetic operators, including braces, are available, as well as the standard elementary functions sin, cos, log etc. This facility is available in all text entry fields throughout the Modeller user interface. See **Input and Output of Real Numbers in LUSAS** for more details

Units Conversion

Model units can be converted at the time of input on all dialogs where dimensional input is expected by clicking inside the right-hand edge of the edit box or grid cell and accessing the **Units Converter and Variation Utility**. A range of conversion units are supplied according to the type of input expected. Model units stated are those previously set.



Reporting of expected model units and triggering of the Units Convertor dialog

For edit boxes, or grid cells, that support coordinate entry the conversion utility allows conversion of multiple comma separated values in the one text box. So, for instance, a coordinate entered as 100,200,300 would be converted to (0.1, 0.2, 0.3) if a millimetres to metres conversion was selected.

Displaying and reporting

Displaying and reporting of model and results data (both to a View window and to print file output) is done in accordance with the consistent set of units stated on the Model Properties dialog. A setting on this dialog provides an option for Imperial modelling units to output coordinate and lengths in feet and inches (e.g. 5'11.5") format.

Changing the modelling units in use

Modelling units can be changed on the Model Properties dialog after a model has been started, but it should be noted that this will change only the label of the units for all existing attribute, geometry. loading definitions. For example, a length of one metre in the model will become one foot if the model units were changed to imperial. Note this behaviour is different to that of changing **timescale units**. In general, changing modelling units during the creation of a model should be avoided.

Timescale Units

Timescale units are specified as part of the creation of a new model and can be changed on the Model Properties dialog. Choosing a timescale unit dictates how time-based values are displayed on dialogs during modelling, and how they are output when processing results. Setting a timescale unit does not change the consistent set of **model units** defined for a model, it simply permits the input or display of chosen timescale units in a different unit of time to that defined by the model units, wherever they are expected to be defined or displayed/output.

Timescale units may be input in seconds, minutes, hours or days. By default, seconds are initially selected to comply with the consistent modelling units options provided.

Timescale units are displayed as a tooltip when the cursor is hovered over the input cell on any dialogs that require time-based values.

Timescale units can be changed on the Model Properties dialog after a model has been started, in which case the tooltip seen on dialogs that support time-based input will show the updated units. Any previously entered time-based values will be dispayed accordingly, for example, a value of 60 representing a timescale in seconds will be displayed and reported as 1 if a timescale unit of minutes was subsequently chosen. Note this behaviour is different to that of changing **model units**.

Model Types

Two types of model can be created in LUSAS:

- **Feature-based geometry models** - these are based on defined features (such as points, lines, surfaces and volumes) or are created from imported geometry and require the definition of mesh objects such as elements with their associated nodes. Prior to version 14.7 of LUSAS these were the only model type available in LUSAS.
- **Mesh-only models** - these comprise only mesh objects (elements and their associated nodes, edges and faces) and are created by importing only those types of LUSAS or third party data that are supported. Initial support for these models was introduced in version 14.7 of LUSAS.

The Modeller user interface presents different menu and context menu items according to the type of model in use. Cursor **selection filters** also differ according to the model in use.

Attribute assignments to different model types

Both feature-based models and mesh-only models require the assignment of materials, properties and thicknesses, loading and supports etc to geometric features or the comparable and equivalent mesh objects prior to an analysis taking place.

Note: In general, whenever an assignment of an attribute to a geometric feature is described in this manual, the same attribute assignment can be made to an equivalent mesh object if a mesh-only model is being used. See the table below for details.

Feature-based model	Mesh-only solid model	Mesh-only surface model	Mesh-only line model
POINT	NODE	-	-
LINE	ELEMENT EDGE	-	ELEMENT
SURFACE	ELEMENT FACE	ELEMENT	-
VOLUME	ELEMENT	-	-

Geometric features and their comparable equivalent mesh objects for attribute assignment purposes

Feature-based geometry models

Feature-based geometry models are created using the four geometric feature types in LUSAS (points, lines, surfaces and volumes). In LUSAS, **geometry** is defined using a whole range of tools under the Geometry menu, or the buttons on the Toolbars. Feature-based geometry models can also be created by importing supported data from third-party software packages using the **File > Import** menu item. Once created, the geometry features are then assigned attributes (materials, properties and thicknesses, loading, supports and mesh/element type etc.) to fully describe the model prior to an analysis taking place. One of the many benefits obtained from using this feature-based modelling method is that built-in associativity ensures that if the model geometry is amended, all assigned loadings, supports, geometric attributes and particularly any mesh assignments and arrangements are automatically updated to suit.

When a feature-based geometry model is being edited all user interface menu items appropriate to the application software in use are available.

Mesh-only models

Mesh-only models are comprised of nodes and elements and do not contain any geometric layer data or feature types, or indeed any geometric data at all. As a result, the option to add a Geometry layer is not accessible. Mesh only models are created by importing finite element data files created either by the prior running of an analysis in LUSAS, or by importing data files from other supported software applications. The File > Import menu item is used to do this. During the mesh import process, LUSAS Modeller creates separate Groups for each element type encountered and for models created from LUSAS data file these will be familiar LUSAS element names. For models created from other third-party software they will be the names used within that system, whatever they may be. An option to create additional groups during the import process based upon the element material type is also available.

On import of a LUSAS Solver data file, element types that are present in the datafile are used directly for the elements that are imported. For datafiles other than those created by LUSAS Solver, the proprietary element types that are present in the datafile are mapped to an equivalent LUSAS element based upon each element's shape and its topology. If the Coupled analysis type has been selected on the New Model dialog prior to a mesh import being carried out then coupled elements will be created during the import process. If a different analysis type is specified after the import of mesh data the element types previously used will be changed accordingly.

Local coordinate systems are not imported into newly created mesh-only models so these need to be defined as necessary to obtain specific types of results as, for example, to get moments around a cylindrical axis.

Assigning attribute data to mesh objects in mesh-only models

No import of any attribute data is done when element data is imported, so the mesh objects (the elements and their nodes, edges and faces) must be assigned attributes (materials, properties and thicknesses, loading and supports), prior to carrying out an analysis within LUSAS, in a similar way and as documented for a feature-based geometry model. Rather than selecting points, lines, surfaces or volumes to make an attribute assignment, nodes, edges, faces, or elements are selected instead. To assist in the attribute assignment process the **cursor selection filter** can be used to identify specific mesh objects.

Editing mesh-only models

When a mesh-only model is being edited any main menu items normally used with feature-based models that are not valid are removed or shown greyed-out to prevent selection. Mesh-only menu items (such as Geometry > Element for instance) are added to the main menu. Context menus may contain different menu items also.

Additional elements cannot be added to a mesh-only model. Once imported, the number and shapes of the elements are fixed but the type of element may be changed by use of the **Change Element Type** option on the context menu on the element group name. In doing so, the number of nodes defining the element topology may be reduced but not increased. For instance, an 8-noded brick elements may be defined for use on previously defined 20-noded brick elements.

Restricted / unsupported functionality for mesh-only models

With reference to the user interface menus for feature-based geometry models the following differences exist for mesh-only models:

- The **Utilities > Slice resultants beams and shells** menu item is not currently supported
- The **Utlities > IMDPlus > Moving load** wizard functionality is not currently supported
- The **Bridge > Moving load** menu item is not currently supported.
- The **Bridge > Prestress Wizard > Tendon profile** and **Multiple tendon prestress** menu items are not currently supported.
- The **Bridge > Prestress Wizard > Single Tendon** design code options are not currently supported.
- Single tendon definition is only permissible by importing data from a spreadsheet.
- Reference paths can only be created currently by typing in the coordinates defining the path.

Model Properties

The Model Properties dialog allows many settings relating to the current model to be defined. Model properties may be accessed from the **File> Model Properties** menu item, by right-clicking the model name (top level) in the Treeview and selecting **Properties** from the context menu, or by double-clicking the Model Properties object in the Analyses Treeview.

The model properties are defined on the following tabs:

- **General**
- **Backups**
- **Notes**
- **Geometry**
- **Meshing**
- **Attributes**
- **Solution**
- **Defaults**
- **Solver System Variables**

General

- **Title** Brief description of the current model as entered in the New Model startup dialog.
- **Model Units** Specifies the consistent set of modelling units to be used.
- **Output feet and inches** (only if imperial units are set) ensures that modelling units used to define coordinates and lengths are reported in feet and inches, instead of just feet or just inches. For example, a length may be output as 5'11.5", rather than 71.5" or 5.92'.
- **Timescale units** dictate how time-based values are displayed on dialogs during modelling, and how they are output when processing results. Units can be specified as seconds, minutes, hours or days.
- **Precision shown in dialogs** Controls the number of significant figures or decimal places displayed in the dialogs.
- **Decimal marker** specifies how decimal points should be handled. Options are: Follow Windows, UK/US decimal point, and European comma formats.
- **Pens** Sets the Pens that are used to draw Points, Lines, Surfaces and Volumes. Click on the **Choose Pen** button to use a different pen, or to modify the Pen Library. Changing a Pen allocation can be applied to existing features and/or new features using the two check boxes.

Backups

Controls where and how many backup versions (revisions)of the model are saved.

- **Auto backup** Maintains another (current) copy of the model file in a specified location.
- **Older revisions** Maintains a specified number of older copies (versions) of the model file in the Associated Model Data folder for that model name. Whenever a model is saved, the previous revision is saved, and if the number of older revisions stated has been exceeded the oldest revision is deleted.

Notes

- **Notes** relating to a model can be typed in the Notes panel of the Model Properties dialog and stored with the model.

Geometry

The method by which the geometry is displayed is controlled using the **Geometry** drawing layer.

- **Merge Options**

    - **Action** controls the criteria that must be satisfied before features sharing a common definition will be merged.
    - **New geometry unmergable** sets the merge status of all new features to Unmergable rather than the default which is Mergable.
    - **Tolerance** controls the distance within which Point features must lie before they will be considered for merging. Note: The merge tolerance should only be changed with extreme caution because changing it from its default value can lead to instability of the underlying geometry engine.

    See **Merging Features** for more details.

- **Default faceting**

    The default faceting controls the number of facets used for shading lines and surfaces. Increasing the number of facets improves the shaded geometry visualisation but takes longer to display. See **Facet Density** for more details

- **Active Local Coordinate**

    Sets the coordinate system as either the Global coordinate or any defined local Cartesian, cylindrical or spherical coordinate.

    If a **local coordinate** is set activate then all subsequent geometry definition is carried out in transformed coordinates.

Advanced Geometric Properties

- **Splitting Defaults**

    The state of the splitting defaults may be set from the advanced geometry dialog for all operations involving splitting operations. The defaults control the check box state and may be overridden during geometry creation.

- **Creation Defaults**

    Process objects in selection order forces objects to be processed in selection order rather than Modellers best fit.

    Allow hollow volume creation allows **hollow volumes** to be created. This option is automatically set true when IGES files are imported. Once set the create volume button will try to create a closed hollow volume when it is not possible to create a solid volume. In addition, extra menu items will appear under the **Geometry> Volume** to enable hollow volumes to be defined.

- **Hole Removal Defaults**

  The state of the holes removal defaults may be set from the advanced geometry dialog for all operations involving hole removal operations. The defaults control the check box state and may be overridden during geometry creation.

- **Merge Defaults**

  The state of the merge defaults may be set from the advanced geometry dialog for all operations involving merging. The defaults control the check box state, and may be overridden during geometry creation.

Meshing

- **Equivalence** Defines the default nodal equivalence tolerance used in a equivalence attribute. If **automatic** is switched on equivalencing is carried out automatically for all nodes in the model and no equivalence attribute assignment is required. See **Nodal Equivalencing** for details.
- **Line Mesh Defaults** Sets the default number of mesh divisions on a line and the maximum subtended angle per element for an arc or splines. If an element exceeds the max subtended angle the number of divisions on the arc or spline will be increased.
- **Irregular tet meshing** specifies the number of the passes and attempts to be made by the tetrahedral mesh generation when attempting to mesh a volume.
- **Create a group of objects that failed to mesh** creates a group named **$failedToMeshObjects** which contains all features which failed to mesh.

Advanced meshing parameters

- **Draw failed parts of mesh only** - draws those parts of the mesh that failed to mesh
- **Linearise element edges**
- **Constrain adjacent linear/quadratic edges** forces mid-side nodes on lines or surfaces to be averaged between the corner/end node positions. This option is turned on by default.
- **Element edge collapsing** invokes **edge collapsing** which removes elements/faces with short edges and small subtended angles by merging them with neighbouring elements.
- **Script to run after meshing** description here

Attributes

- **General Options**

  - **Apply concentrated loads in cylindrical coordinates for Fourier elements** (LUSAS Solver option 202).

- **Body force given as acceleration** This option is turned on by default. Turning it off converts body forces to global loads per unit volume. (LUSAS Solver option 48).

❑ **Slideline Options**

- **Suppress stringent slave search** For simpler geometries, such as flat surfaces in contact, a slight reduction in processing time may be achieved by suppressing the "stringent" local node search but this is not usually recommended. (LUSAS Solver option 184). See the *Theory Manual* for more details.
- **Suppress initial slide-surface stiffness check** Slideline stiffnesses are automatically scaled at the beginning of an analysis if the average master/slave stiffnesses differ by a factor greater than 100. This is to account for contact between bodies that have significantly different material properties. See the *Theory Manual* for more details.
- **Suppress initial penetration check** The coordinates of all contact nodes that have penetrated prior to the commencement of an analysis, are reset back to the closest point on the contacted surface. This option should be set if the node resetting is to be suppressed, such as when performing an interference fit analysis. See the *Theory Manual* for more details.

❑ **Allow modification of LUSAS generated attributes** overrides the default behaviour of not allowing any changes to be made to Modeller-generated attributes such as loading attributes generated as a result of carrying out cable tuning analysis, vehicle load optimisation, prestress calculations, and rail track analysis.

Solution

Solver options can be specified to be common to all analyses or be set individually for each analysis. Element and Draping options apply to all analyses.

Solver Options

When the type of solver selected is set to Default the fast multi-frontal solver will be used if this option is included in the licence agreement. This may be overridden by selecting the solver required. For further information see **Selecting a Solver**.

❑ **Optimiser Options** When using the standard frontal solver the **frontwidth** of the problem may be reduced by optimising the order in which the elements are presented to the frontal solver. The type of **optimiser** to be used is selected from an options dialog. No optimisation is required when using the fast multi-frontal solver. For further information see **Selecting a Frontal Optimiser**.

- ❑ The **Advanced...** button enables a new Modeller option to be specified or particular default values or settings to be modified. Advanced settings should generally only be modified with the assistance of LUSAS technical support.

When specified separately, a Solver options object is added to the Analyses Treeview for each analysis entry. Double-clicking this object provides access to specify the Solver type and optimisation options.

Element Options

See the *Element Reference Manual* for details of which elements can be used with these options.

- **Assign 6 degrees of freedom to all thick shell element nodes** By default this option is on. It has the effect of adding a rotational spring stiffness to the drilling rotation of the thick shell elements making the analysis more stable. The value of the spring stiffness can be adjusted using the system parameter STFINP. For problems in which geometric nonlinearity (Option 87) is being used more accurate results may be obtained by switching this option off and letting LUSAS automatically establish the need for 5 or 6 degrees of freedom at a node. (LUSAS Solver option 278).
- **Axisymmetry about Global X axis** When selected, LUSAS considers the line of axisymmetry in an analysis to be about the global X axis and not the default, which is the global Y axis. (LUSAS Solver option 47).
- **Lumped Mass Matrix** Formulate lumped mass matrix instead of consistent mass matrix for elements. (LUSAS Solver option 105).
- **Write strains to output file** causes element strains to be written to the Solver output file.
- **Preserve loading whilst elements deactivated** retains the loading assigned to all elements in model (activated and deactivated) until a subsequent load case is applied. Typical usage would, for example, be when carrying out a staged construction analysis. Self weight would be assigned to all elements (activated and deactivated) in the loadcase representing stage 1 of the analysis. As stage 2 and subsequent loadcases are activated the loading initially applied to elements in stage 1 would automatically be applied as the elements become active. Note that when using this option no additional loading should be applied to any loadcases following the one that contained the load assignments. If done so, the initial applied loading for those loadcases will be lost.

- ❑ **Integration options**

  - **Fine integration for stiffness and mass** Invokes a finer numerical integration rule for elements. (LUSAS Solver option 18).

- **Fine integration for mass HX16 and HX20** Formulate mass matrix with fine integration. (LUSAS Solver option 91).
- **Coarse integration for semi-loof shells** Invokes coarse numerical integration rule for semiloof elements. This option under-integrates the semi-loof shell element which may have the effect of removing low energy mechanisms when the element is very thin and/or pressure loaded. (LUSAS Solver option 19).
- **Newton-Cotes Integration for beam elements** Newton-Cotes is a form of numerical integration or quadrature. It is often used for through-thickness integrals since sampling points are located at the extremes of the range. (LUSAS Solver option 134).

Element options are accessed via the Model Properties dialog or the Model Properties control object  in the Analyses  Treeview. Element options can only be specified to be common for all analyses.

Draping Options

The draping options control the composite lamina draping process. The draping process works by effectively draping a square mesh of pinned bars over the structure. The length of the bars is considered to be the drape mesh size. A smaller bar size provides a more accurate drape at the expense of computer time and memory usage. Typically the drape mesh size should be about half the element size.

- **Drape by mesh size** - Desired drape cell size. If the desired drape cell size would generate more than the specified maximum number of drape cells then the drape mesh size would be adjusted to generate the maximum number of drape cells. Likewise, if the desired drape cell size would generate less than the specified minimum number of drape cells then the drape mesh size would be adjusted to generate the minimum number of drape cells.
- **Drape by number** - Desired number of drape cells on draping surface. This must be between the maximum and minimum values stated below.
- **Drape by face multiplier** - A number that is used to multiply the actual number of mesh divisions on a draping surface to arrive at a desired drape cell number. This must be between the maximum and minimum values stated below. A default value of 4 is entered.
- **Maximum number of drape cells** - Maximum number of drape cells to be generated.
- **Minimum number of drape cells** - Minimum number of drape cells to be generated.
- **Extend drape grid one row** - Ensures the edges of the component are fully enclosed by the draping grid. Note that the grids for LUSAS draped solids and shells are automatically trimmed at Surface boundaries.

Draping options can be accessed via the Model Properties or Draping options objects in the Analyses Treeview. Draping options can only be specified to be common for all analyses.

Defaults

Sets the defaults for symbols, arrows and text which are used when visualising attributes. Actual settings are controlled from the Attribute Visualisation layer for the current window.

The value and units for **Gravity** that are currently in use may be checked on this tab of the dialog.

The **Advanced...** button enables a new Modeller option to be specified or particular default values or settings to be modified. Advanced settings should generally only be modified with the assistance of LUSAS technical support.

Solver System Variables

By default the LUSAS Solver is set up to operate efficiently for a wide range of modelling and analysis problems. In some cases it may be occasionally necessary to adjust the Solver system variables. Variables that can be modified inside Modeller are accessed via the **File> Model Properties> Solver System Variables** dialog. Making changes to **Solver system variables** should generally be done only after seeking advice from LUSAS Technical Support.

Using Windows

Each Modeller window can display a unique view of a model. View layers inside each window hold model information that can be added or removed from the current window as required. Multiple windows can be opened using the **Window> New Window** menu item. Closing the last window will close the model.

Windows consist of the following components:

- **Drawing Layers** By default the first panel of the tree frame displays each model window currently open, and shows the view **layers** contained in that window.
- **View Properties** Double-clicking in a window or right-clicking in a window with no selection, will display the view properties of that window. The **View Properties** can be altered as required.

Saving a View

A view of a model may be saved with the current settings for future use using the **Window> Save View** menu item. Any new window is based on the **default** view. Therefore, if a view is saved with the name **default**, all new windows are based on this view. The view name **Factory default** cannot be changed and is the default view when the system is installed. The following settings are saved:

- **Rotation** The current rotation consists of a vector and an origin as specified on the View tab of the View properties

- **Layers** The window layers contained in the Treeview.
- **Colours** The colourmap as used for plotting results contours.
- **Page size & borders** The page size and border setting as defined from the File> Page Setup menu item.
- **Pen Library** The pen library referenced every time a item is drawn.

All the above settings are saved. The choice of which to apply to a new or current window is made when a view is loaded.

Pen Library

The choice of colour for various operations is linked to the Pen Library. The Pen Library contains twenty pens, each numbered, each with a colour, style and thickness that may be set if desired.

Pens are used in a number of dialogs. Whenever a pen is used a **Choose pen** button allows access to the Pen Library to specify a different pen, or to change the pen colour or style. If a particular pen style or colour is changed then this will affect every operation that references that pen.

Loading a View

A previously saved view may be loaded into a new window, or into the current window using the **Window> Load View** menu item. Any of the saved view settings can be chosen.

Copying Windows

A window can be copied by selecting and copying the window name in the Treeview, then pasting the window back into the Treeview. The window layers and settings are also copied.

Using Layers

Selective viewing of model and results data is done using separate pre-named layers in the Layers Treeview. Layers can be added to each view window by right-clicking on a blank part of that view, or by right-clicking the window name in the Layers Treeview, or by selecting the **View> Drawing Layers** menu item.

The following layers can be generally manipulated from the Layers Treeview:

General modelling layers

- **Geometry**
- **Mesh**
- **Attributes**
- **Labels**
- **Annotation**

- **Utilities**

When an influence analysis has been carried out the following additional entry will be added to the Layers Treeview. This layer cannot be added manually.

- **Influence shape**

Results processing layers

- **Contours**
- **Vectors**
- **Deformed mesh**
- **Diagrams**
- **Values**

Manipulation of Layers

The order in which the layers are drawn to the view window is defined by the order or the layers in the Layers Treeview. The top layer in the Treeview is drawn first. If the display of one layer is eclipsed by another layer, (i.e. the mesh is eclipsed by the contours, or the annotation is eclipsed by the deformed mesh), the eclipsed layer can be moved down in the Layers Treeview by selecting it, then dragging and dropping it to a new position to cause it to be drawn following a particular layer.

A context menu (accessed by right-clicking on each layer name in the Layers Treeview) provides additional manipulation options:

- **Copy** (and **Paste**) allows a layer (and its corresponding layer settings) to be copied and pasted between different view windows of the same model.
- **Delete** removes the layer from the view window
- **On/Off** controls the display of the layer in the view window. Turning a layer off retains the layer in the Layers Treeview but removes the display of that layer from the view window.
- **Reset** returns the layer to its default settings
- **Move up** / **Move down** moves the selected layer to be drawn before or after the preceding or following layer.
- **Properties** Double-clicking on a layer name (whether or not is it On or Off) will display a Properties dialog where the style of the viewed layer can be changed. Clicking OK will turn On a previously turned Off layer.

Layer symbols explained

A symbol adjacent to each layer name in the Layers Treeview shows the display status of each layer:

A coloured layer symbol indicates that the display of a layer has been turned 'on'.

A greyed-out layer symbol indicates that the layer has been turned 'off'.

A red circle with a line through it indicates that no results are loaded or currently available for this layer, or inappropriate settings are currently set.

View properties: options and settings

Selecting Model Features

Items displayed in the view window may be selected with the cursor by clicking on them individually or by rectangular, circular or polygonal area selection. Following selection, items may be added or removed from the initial selection by carrying out further selections based upon either menu choices or upon particular keystrokes used.

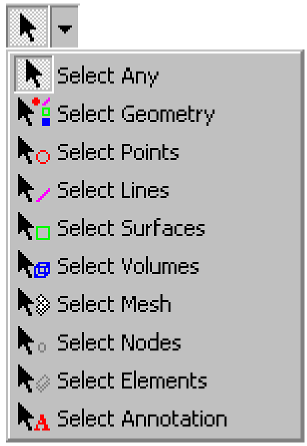
Items in a current selection may be viewed in the Browse Selection window which can be displayed from the **View> Browse Selection** menu item or using a right-hand mouse click in the Selected area of the status bar at the bottom of the graphics area. By means of checkboxes the selected items may be individually unselected.

Cursor-based selections

The standard cursor can be used to select any object, i.e. Points, Lines, Surfaces, Volumes, Nodes, Elements, Annotation according to the model type in use. It can also be used to make rectangular area selections.

Greater control over what is selected can be achieved by changing the cursor selection filter. The cursor options listed change according to the type of model in use.

Selection filters for feature-based models



- The selection cursor will select any object, i.e. Points, Lines, Surfaces, Volumes, Nodes, Elements, Annotation
- The selection filters allow only specific objects to be selected, i.e. with the Line filter only Lines can be selected and with the Surface filter only Surfaces can be selected.
- Selection filters can be activated either from the drop buttons to the right of the cursor button, or using keyboard shortcuts. To display the drop buttons click on the down-arrow to the right of the cursor button. Alternatively hold down the appropriate key whilst selecting using the normal cursor as follows:

G – Geometry selection filter

P – Point selection filter

L – Line selection filter

S – Surface selection filter

V – Volume selection filter

M – Mesh selection filter

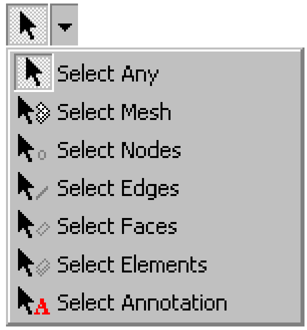
N – Node selection filter

E – Element selection filter

A – Annotation selection filter

When a specific selection option has been chosen the on-screen cursor will show a graphical representation of the chosen option.

Selection filters for mesh-only models



- The selection cursor will select only Nodes, Elements or Annotation
- The selection filters allow only specific objects to be selected, i.e. with the Edges filter only edges of elements can be selected and with the Faces filter only Faces can be selected.
- Selection filters can be activated either from the drop buttons to the right of the cursor button, or using keyboard shortcuts. To display the drop buttons click on the down-arrow to the right of the cursor button. Alternatively hold down the appropriate key whilst selecting using the normal cursor as follows:

M – Mesh selection filter

N – Node selection filter

B – Edge selection filter

F – Face selection filter

E – Element selection filter

A – Annotation selection filter

When a specific selection option has been chosen the on-screen cursor will show a graphical representation of the chosen option

See **Appendix C - Model Selection Shortcuts** for a complete listing of all the selection keys available.

Area selections

Regions or areas of the model may be selected as a rectangular, a circle, or a polygon either using the appropriate toolbar buttons or the keyboard short cut.

To select a rectangular area click to define one corner, hold the mouse button down, and drag the cursor to the opposite diagonal corner. This can also be achieved using the standard cursor.

Pick the circle selection tool or hold down the **C** key then select the centre of the circle and drag the edge of the circle to the required radius.

Pick the polygon selection tool or hold down the **X** key then select each corner of a polygon and either double click to close the polygon or select **Close Polygon** from the context menu. It is invalid to define a vertex which would cause two lines on the perimeter to cross. Because more than one click is needed to define a polygon, individual items may not be selected whilst in polygon select mode.

Cursor and Area Selection Modifiers

Items can be added to, toggled, or removed from a selection by the use of specific keys. The default keys used are Windows defaults:

- Holding down the **Shift** key whilst selecting items will *Add* the newly selected items to those currently selected.
- Holding down the **Control** key while selecting items will enable the selection state of the item selected to be *Toggled*.
- Holding down the **Shift** and **Control** keys while selecting will *Remove* items found from the current selection
- By default all items completely enclosed in a selected area will be selected. By holding down the **Alt** key, items intersecting the selection perimeter will also be selected. The **Alt** key may be used with, or independently from, the **Shift** or **Ctrl** keys. The **Alt** key can also be used with feature selection shortcuts e.g. **Alt + Shft + L** adds lines to the current selection.
- All visible items can be selected together using the **Select All** command which can be invoked from the **Edit> Select All** menu item, from the right-click context menu or using the **Control + A** keyboard shortcut.

Mapping of Keyboard Modifiers for Cursor and Area Selection

The Cursor drop-down menu contains a **Keyboard Mapping...** menu item which allows the default Windows-based cursor and area selection keyboard modifiers to be changed to suit those that are generally used with CAD modelling systems or be user-defined.

- ❑ **Windows** - Uses the defaults stated in the Cursor and Area Selection Keyboard Modifiers section.
- ❑ **CAD** - Uses the defaults shown in the following table.

Keyboard modifier actions when selecting items using Windows defaults and those generally used by CAD systems:

Key type or sequence	Windows	CAD systems (generally)
No keys	Set	Toggle selection
Ctrl	Toggle selection	Set
Shift	Add to selection	Remove from selection
Ctrl+Shift	Remove from selection	Add to selection

User-defined modification of the keyboard modifiers requires the Intersection key to be defined initially, followed by drop-down menu selections for the No keys, Ctrl, Shift, and

Ctrl+ Shift key types/sequence, where each has to be uniquely set to be Set, Add, Toggle or Remove.

Selecting coplanar neighbours

After selecting a surface or element face the **Select Coplanar Neighbours** menu item can be selected from its context menu. This provides a quick way to select a number of surfaces or element faces according to their alignment in relation to the selected surface or element face. By specifying an angular tolerance all surfaces or element faces where the angle between the normals of adjacent surfaces or element faces lies within that tolerance will be added to the selection. This is of particular use when defining a draping surface as used in composites analysis. An option to ignore internal faces helps to ensure that when an element has faces that both lie within the angular tolerance, only the external face is selected.

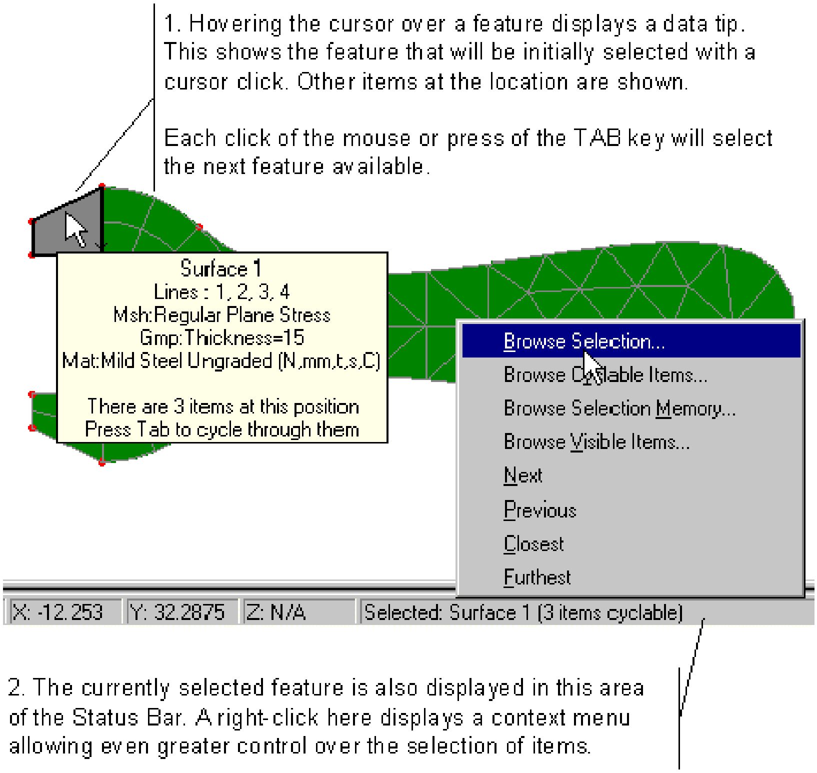
Advanced Selection Filter

The advanced selection filter accessed via the context menu item **Advanced selection...** for a view window enables geometry, nodes or elements to be selected based on a number of different criteria. Items may be selected by number or range or numbers, for example 1T5I2 (representing 'one to five in increments of 2') will select item 1,3,5). When a results only file is loaded the geometric and material attributes reflect the numbers in the solver data file. Geometry can by selected according to the connectivity of the feature to surrounding features; End Points, Free Lines and External Surfaces can be highlighted in this manner.

Cycling through the Selection

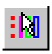
When features lie close together or overlap it can be difficult to select the required feature first time. In these circumstances, each separate press of the **Tab** key or click with the mouse at the same position selects a different feature. The currently selected feature is displayed in the Status Bar at the bottom of the screen.

Alternatively, click in the Status Bar to cycle the selectable features. Or right-click in the Status Bar to display a context menu from which **Next**, **Previous**, **Closest** or **Furthest** may be selected

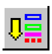


Even closer control over the selection of items at the same cursor location can be attained using the Browse Cyclable Items window which is available on the right-click in the status bar or from the **View> Browse Cyclable** Items menu.

Associate Selection Downwards

 **Associate selection downwards** is an option to automatically select lower order features when selecting objects in the model. For example, selecting a Surface will also select the Lines and Points that define the Surface. This option may be invoked from the **Edit> Associate Selection Downwards** menu item. Associate selection downwards is set off by default.

Associate Operations Downwards

 **Associate Operations downwards** is a option to operate on lower order features when a command is issued. For example, deleting a Surface will also delete the Lines and Points which define that Surface. This option may be invoked from the **Edit> Associate Operations Downwards** menu item. Associate operations downwards is set on by default.

Selection Memory

Some operations require features to be stored in selection memory prior to other features being selected in order to complete an operation. One example of use would include the setting of a single point in selection memory prior to the normal selection of a number of other points prior to selecting the Geometry> Line> By joining menu item to create a fan of lines from the selected point. Another example would be to define the master and slave members when making joint material assignments to sets of multiple lines. Here, the set of lines representing the master would be set in selection memory and the set of lines representing slave items would be selected normally prior to the material being assigned.

The selection memory commands are available from the **Edit> Selection Memory** menu item or from the context menu enabled by right-clicking in the graphics area. The selection memory commands are:

- **Set** - clears the selection memory and set the contents to the items in the selection.
- **Add** - adds the selected items into selection memory.
- **Remove** - removes the selected item from selection memory. (Only available if the selected item is already in selection memory)
- **Recall** - places all objects in selection memory into the main selection. Objects are not removed from selection memory.
- **Clear** - clears the selection memory.
- **Browse**- displays the Selection Memory browse window.

Selection Colour

The Pen used to draw items in the selection or the selection memory may be changed from the View properties. To display the View properties double-click (without anything selected) in the View window and choose Properties from context menu.

Groups

The Groups Treeview is used to store selected user-defined collections of objects (geometry, nodes or elements) under a collective name. For example, a certain set of geometry might be grouped together with the name **Nut**, whilst another set might be grouped and named **Bolt**.

LUSAS also automatically creates groups as part of the general modelling process as slice sections are created, or as a result of an analysis when slidelines are present in the model. With slice sections, groups are created with the group name **Slices** with each slice section having a group name of **Slice 1**, **Slice 2** etc. With slidelines, a group named **Slideline Results** is created containing master and slave group names for each defined slideline.

Groups are also automatically created when importing data files from other supported third-party software applications to create mesh-only models. In this case groups are named after element or with material references, if present in the data file.

When using LUSAS HPM software, groups are automatically created to simplify the modelling of the composite parts, as well as the interface surface and other items.

Uses of Groups

- ❑ Enabling unique components to be identified, manipulated, hidden, or have results plotted only on those features.
- ❑ Allowing the assignment of attributes to a group in one step. The appropriate geometry features will be used if attributes can not be assigned to all geometry types.
- ❑ Identifying all the features that failed to mesh during any command that invokes meshing. See **Fixing Meshing Problems**
- ❑ When defining slice sections through a model in order to view the internal arrangement or to plot results.
- ❑ To allow easy manipulation of master and slave results following a slideline analysis.

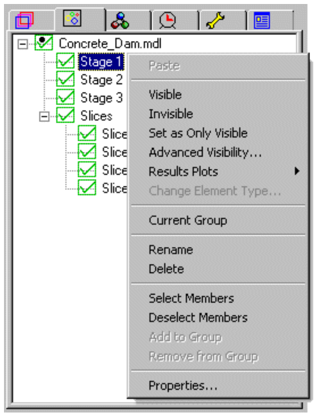
Defining Groups

First, select the model features to be grouped together and then select either the icon on the main toolbar or use the **Geometry> Group> New Group** menu item.

All current group definitions are listed in the Group panel of the Treeview. Manipulation of Groups can be carried out using the context menu of the Group panel, and/or using the sub-menu entries under the main **Geometry> Group** menu item.

Naming Groups

Groups can be given a meaningful name at the time they are defined. They may later be renamed in the Treeview or from the group properties accessed from the context menu.



Manipulating Groups

From a group's context menu the following commands will act on that group in the current window:

- ❑ **Visible** Makes all members of the group visible.
- ❑ **Invisible** Makes all members of the group invisible.
- ❑ **Set as Only Visible** Makes the whole model invisible and then sets all the members of the group to be visible.
- ❑ **Advanced Visibility** Enables the visibility of the higher and lower order features of the members of a group to be manipulated.
- ❑ **Results Plots** permits results to be selectively plotted for the chosen group. For details see **plotting results for groups** in the results viewing section.
- ❑ **Change element type** (mesh-only models) permits changing the element type of a group of imported elements by description, or by entering a specific known element name.

- **Current Group** Set the group to be the **current group**.
- **Rename** Enables a group to renamed.
- **Delete** Deletes the group. The group is deleted but the contents of the group are not.
- **Select Members** Adds the members of the group to the current selection.
- **Deselect Members** Removes the members of the group from the current selection.
- **Add to Group** Adds the currently selected items to the group.
- **Remove from Group** Removes the currently selected items from the group.
- **Properties** Displays a list of the member items of the group.

When the group defines a slice section the otherwise generally available Rename, Delete, Add to group, and Remove from group options are unavailable but additional slice-related options are include on the context menu:

- **Draw axes** Draws the local axes for the slice section. Note that options on the Contours properties dialog allow for plotting of results on slice sections and at slice axis directions.
- **Print Properties** Displays information about the slice including cross-sectional area of the slice, centroid and section property data.
- **Print Forces** Displays force and moment information for the slice.

Group symbols explained

By default a symbol adjacent to each group name in the Groups Treeview shows the visibility and status of each group.

When modelling:

A black dot next to a symbol denotes the current group into which all new geometry will be added when created.

(green tick) All of the objects in this group are visible.

(blue tick) Some of the objects in this group are visible.

(red cross) None of the objects in this group are visible.

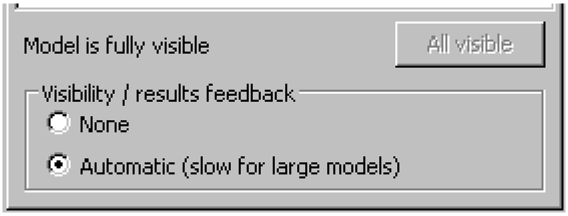
When a results file is loaded:

(green tick, green border) All of the objects in this group are showing results.

(blue tick, blue border) Some of the objects in this group are visible and some are showing results.

(green tick, blue border) All of the objects in this group are visible but only some are showing results

Group visibility / symbol feedback panel



The group visibility / symbol feedback panel at the bottom of the Groups Treeview reports on whether any groups have been made invisible, or whether the model is fully visible. The visibility / results feedback settings control whether the group symbols are displayed to show the status of the objects in the group, or whether no group visibility symbols are to be used. For particular sizes of model, and numbers of objects within a number of groups, it may be necessary to turn-off the automatic group feedback option to retain optimum display speeds.

Changing the Visibility of Features

By default all model geometry is visible (providing the geometry layer is present and turned 'on' in the Layers Treeview, and providing it is not hidden by another layer) but as models get larger it is convenient to temporarily turn-off the display of parts of the model.

Features may be made invisible using any of the following methods:

- By selecting features and choosing **Invisible** from the context menu activated from the view window.
- By selecting an attribute from the Treeview and choosing **Invisible** from its context menu.
- By creating a group from selected model features and choosing **Invisible** from the group name's context menu to make the members of the group invisible.

Features may be made visible by using any of the following methods:

- By choosing **All Visible** from the context menu activated from the View window (used if any features have been made invisible)
- By choosing **Visible** from an attribute's context menu to re-display the features assigned to the attribute
- By choosing **Visible** from a group name's context menu to make the members of the group visible again.

In addition the advanced visibility dialog activated from the View window's context menu allows fine control on the visible / invisible items by controlling the visibility of higher order and lower order features. i.e. This allows all Lines attached to a Point to made visible or all Surfaces connected to a Line to be made invisible.

Notes:

- All visible features will always have their lower order features visible. i.e. If a line is visible its defining points will also be visible.
- When a feature is made visible any associated elements or nodes will also be made visible. Use the Advanced Visibility dialog to override this behaviour.
- An element can only be made invisible if its defining feature is invisible. To make a chosen element invisible, select the element and then use the Advanced Visibility dialog to make the element invisible by selecting the **Also apply to higher order** option. This makes the selected elements and the defining features invisible without making the unselected elements invisible.

Rotating, Zooming and Panning

A number of tools are available to manipulate the view of the model.

- ❑ **Dynamic rotation** is carried out by selecting the dynamic rotate button, or by pressing the scroll wheel button and either the left or right mouse button or by holding down the **R** key, while moving the mouse in normal cursor mode. The model will be rotated about the centre of the model unless any part of the model is selected in which case the model will rotate about the centre of the selection. The model can additionally be rotated around any of the screen axes by pressing additional keys. See **Rotating the Model**.
- ❑ **Dynamic zoom** is carried out by selecting the dynamic zoom button, or by scrolling the mouse wheel or by holding down the **Z** key while moving the cursor in normal cursor more. If any part of the model is selected the location of the centre of the selection will remain fixed.
- ❑ **Dynamic pan (drag)** is carried out by selecting the pan button, or by depressing the scroll wheel button or holding down the **D** key while moving the mouse in normal cursor mode.

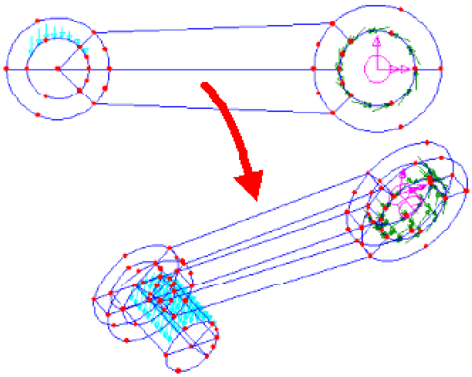
Notes:

- Rotation, zoom and pan model manipulations are applicable for all cursor input modes including normal cursor selection of features, defining lines by cursor or when section slicing.
- For larger models when the refresh time is significant the model display will reduce to an outline view when using rotate, zoom and pan.

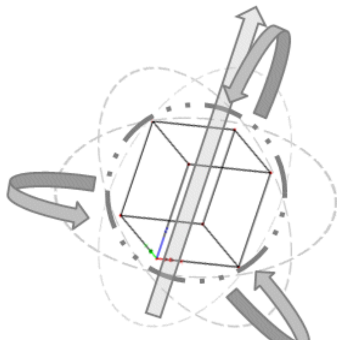
Rotating the Model

Three methods of rotating the view of the model are available, each one is a button on the View toolbar:

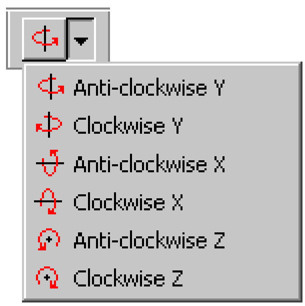
-  **Dynamic rotate** Allows the model to be rotated dynamically using the cursor. The model rotates around various multiple axes when the cursor is moved. By holding down the **Control** key, the **Control** and **Shift** keys or the **Shift** key while using the dynamic rotate the model can be rotated independently about the screen Z, Y and X axis respectively. Click on the normal cursor to return to selection mode.
  Note also that holding down the **Alt** key provides a one-key option for rotating about the the screen Y axis.

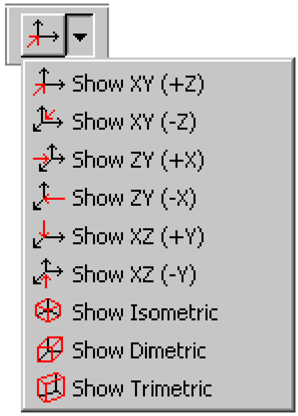


Note. Models are dynamically rotated using the 'model ball' method. With this method, a model can be imagined to be surrounded by a sphere such that a mouse-click and a drag of the cursor on the screen represents clicking on the surface of the sphere and dragging to rotate it to a new position. In doing so, it is important to note that the model rotation is restrained to rotate only around the model's vertical axis (as defined on the Vertical Axis dialog) - unless any **model viewing shortcuts** are being used at the same time. A benefit of this approach is that, no matter where on the screen you click to start the rotation of your model, if you return the mouse pointer to the same spot, (whilst you are dynamically rotating the model), the model will return to its original position and orientation on the screen.



- **Incremental rotate** Allows the model to be rotated by an specified rotation about a chosen global axis. The specified rotation may be modified by adjusting the rotation increment on the window's properties.





- **View along axis** Views the model along a chosen global axis.

Note that this toolbar button is not provided as part of the standard user interface but it can be added by selecting the View > Toolbars > Customise > menu item.

Note that this facility is also available by clicking in the X, Y or Z boxes in the Status Bar (holding-down the Shift key, if needed, to obtain the views along the negative screen axes.)

Zooming in or out

Zoom tool This tool works in two ways:

1. By dragging a box around part of the model the view will zoom into that area.
2. Clicking on part of the model will zoom in progressively for each click. Hold the **Control** key to zoom out.

Dynamic Zoom is similar to dynamic rotate. Using the cursor the model can be visually enlarged or reduced in size.

Panning (Dragging) the Model

Dynamic Pan allows the model to be dragged into position on the screen.

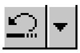
Reseting the View

Home Restores the view to a scaled to fit view in the XY plane.

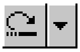
Resize When the resize button is depressed the model will fit into the available screen area.

See Appendix C - **Model Viewing Shortcuts** for a complete listing of all model viewing facilities and keyboard shortcuts available

Undo/Redo



The **Undo** button allows any number of actions since the last save to be undone. If more than the last action is to be undone then the actions to be undone may be selected from the undo history list by clicking on the down arrow at the side of the undo button.

 **Redo** is available immediately after choosing an undo event to enable the undone action to be reinstated.

Notes:

- The undo facility works by replaying the session file from the last save. Because of this, it is advisable to save the model frequently to speed-up the undo facility.
- Undo is only available when a model file is loaded. (i.e. undo is not available when a results file is loaded without a corresponding model file)

Page Layout Mode

Two viewing modes are available, both accessed from the **View** menu:

- ❑ **Working Mode** is useful for model generation. In working mode annotation is scaled and moved so it is always visible.
- ❑ **Page Layout Mode** enables the model to be viewed as it would appear on a printed page. This makes annotation easier to position and allows pictures to be created to a specified scale.

Scaling

In Page Layout Mode the model is scaled and positioned within the margins defined using **Page Setup> File** menu item. This behaviour may be modified by toggling the scale to fit window  button.

If a picture is to be created to a specified scale the page size should first be set using the **File> Print Setup** menu item. The desired scale and position should then be set on the **View** tab of the **View Properties** accessed from the context menu. In this dialog the **scale to fit window** option should be switched off and the **scale** and **origin position** defined.

Annotating the Model

The View window may be annotated using the **Utilities> Annotation** menu item. Annotation can be placed by either cursor positioning or by specifying a coordinate location in Frame or Model coordinates. Annotation added to the model is displayed in the Annotation layer in the  Treeview.

Cursor positioned annotation

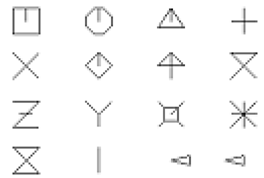
- ❑ **Line** Single lines may be added in a selection of colours and line styles.
- ❑ **Polygon** Filled or unfilled polygons may be annotated on the screen in a selection of colours. Left click to indicate successive polygon vertices. When at least three vertices have been indicated, right click and select Cancel or Close from the context menu.
- ❑ **Bitmap** Adds a bitmap from a selected file

- ❑ **Banner** Adds the LUSAS banner
- ❑ **Arrow** Defines an annotation arrow of a default size and colour

Coordinate positioned annotation

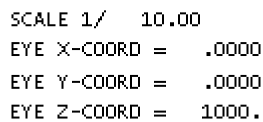
Several types of annotation are available:

- ❑ **Text** Any number of lines of text may be plotted in a selection of fonts, character heights, angles and colours. Requires text setting-out point to be defined.
- ❑ **Line** Requires start and end points of line to be defined. Single or multiple lines may be added in a selection of colours and line styles.
- ❑ **Polygon** Requires points to be defined for each vertex.
- ❑ **Bitmap** Adds a bitmap from a selected file at a specified location point
- ❑ **Banner** Adds the LUSAS banner at a specified location point
- ❑ **Arrow** Defines an annotation arrow at specified start and finish points in chosen pen colour.
- ❑ **Symbol** A selection of symbols may be plotted in a selection of sizes, angles and colours.



Other Annotation

- ❑ **Window border** Displays an annotated frame around the Window containing the LUSAS version number in use with the model name, the date, the model title, and the model units.
- ❑ **Window summary** Window summary annotation is added in the form of an automatically assembled text block. It displays information about the model such as its view scale and orientation, and if a results file is loaded a summary of key values for a particular loadcase.



- ❑ **Window summary position** The location of the summary block of text with reference to the left, right, bottom or top of paper. Note that the Window summary can be moved easier graphically by selecting it and then dragging it to a new position.

Notes

- If the annotation is eclipsed by other model data drag the layer (in the Treeview) to the bottom of the stack so that it is drawn last.

- An annotation toolbar is available but by default is hidden. It can be displayed from the **View> Toolbar** menu item. From this toolbar coordinate positioned text and bitmaps can be defined, and cursor positioned lines, boxes, polygons and arrows added.

Editing Annotation

After being added to a model window, both cursor positioned and coordinate position annotation can be easily moved by selecting it with the cursor and dragging to a new position. In addition all defining parameters such as location, style and in some cases content can be edited.

To modify the properties of a piece of annotation select the annotation, right-click, then choose Properties from the context menu. The following properties apply to most annotation types:

- ❑ **Anchor point** Annotation can be located with respect to the model (model coordinates), or to the window frame (frame coordinates).
- ❑ **Visibility** The annotation can be made visible on all windows or just the current one.
- ❑ **Pen** The pen used to draw lines or polygons can be selected from the pen library. Line thicknesses can be edited. The font used for text annotation can be modified by clicking on the **Font** button.
- ❑ **Name** The identifying name of the annotation item. If no name is entered an automatic numeric identifier is allocated.

Notes

- By default the anchor point of all annotation is positioned in frame coordinates from the bottom left corner of the page. This enables the annotation to be positioned separately from the model. If negative values are specified for the anchor coordinates then the annotation is positioned from the top right corner of the page.
- Annotation may be tied to the model by specifying the anchor point in model coordinates.
- Annotation positioned in frame coordinates can be moved by selecting the annotation in the graphics window and dragging to the required location.
- Annotation lines are used where 2D graphing results are to be recreated at defined locations. See **Results on Sections Through a Model**
- Polygons are used when a section through a 3D model is to be recreated at a defined location. See **Results on Sections Through A Model**

Saving a Model

When a model is saved using the **File> Save** menu item all model properties, views and associated values are saved also. If a results file was open when the model was saved this is not re-opened automatically when the model is re-opened later.

Customising the Environment

Various user-definable settings and facilities allow the interface to be customised.

- **View properties** contain options relating to the current window.
- **Startup templates** can be used to pre-load the Attributes Treeview of the interface with selected attributes for a particular analysis, set default mesh or material types, or define preferred colour schemes - to name just a few uses.
- **Toolbars and toolbar buttons** can be customised and user-defined toolbar buttons can added to the user-interface either to sit on a new toolbar group or alongside existing buttons in an existing group.

View Properties

The View Properties dialog allows general information relating to how both the model and results are viewed for a selected window.

View properties may be displayed in various ways: by double-clicking inside a window (with no features selected); by right-clicking a window name in the Layers Treeview and then selecting Properties from the context menu, or by double-clicking the view properties entry for a particular Window in the Layers Treeview.

View properties are defined on the following tabs:

- **General** Show or hide the screen ruler, selection tolerance, and enables the window background and selection colours to be modified.
- **View** Shows the view rotation vector and rotation increment. Add or removes a window border or window summary. Controls if elements activated in the active loadcase are displayed or not. Enables the scale and origin position to be set and allows the current view settings to be saved.
- **View axes** Controls whether views axes are displayed and at what position. The style of axes can also be set.
- **Deformations** Specify the amount by which the nodal deformations are scaled.

General

- **View name** is the name of the current window
- **Show Rulers** Shows, (or hides), the X,Y,Z axis rulers of the current window.
- **Selection tolerance** Sets how close the cursor has to be to a feature to be able to select it.

LUSAS uses standard Microsoft Windows colours to define the screen colours by default. By deselecting the **Use Windows colours** option the following colours can be changed:

- **Background colour** Sets the window background colour.
- **Selection Pen** Sets the pen colour used to draw model feature when they are selected.
- **Selection memory Pen** Sets the pen colour used to draw model features when they are in selection memory.

View

- **Scale to fit window/page** When enabled automatically adjusts the zoom level to fit the model to the view window/page.
- **Scale** (with Scale to fit window/page disabled) allows the scale ratio of the view to be set manually
- **Origin position** Defines the origin of the view.
- **Rotation Vector** An equivalent eye position coordinate. Entering (0, 0, 1) views the model from the Z axis, and (1, 2, 3) gives a three-dimensional view.
- **Rotation increment** Sets the rotation increment used for incremental rotation.
- **Triangle sort** Defines the triangular sort algorithm to use when shading (For OpenGL with GDI drivers only).
- **Border annotation** Displays an annotated frame containing the LUSAS version number in use with the model name, date, title, and the model units.
- **Summary annotation** An automatically assembled text block that displays information about the model such as its view scale and orientation and, if a results file is loaded, a summary of key values and settings for a particular loadcase.
- **Location** Opens a dialog to allow the location of the summary block of text to be set with reference to the left, right, bottom or top of paper. Note that the Window summary can be moved more easily graphically by selecting it and then dragging it to a new position.
- **Show only activated elements** Controls whether only the elements activated in the active loadcase are displayed or not.
- **Save View** Saves the current view, including the view properties, pen library, colour map and window layers. Note that when a view is loaded into a window there is a choice of what to reload e.g. colours, layers, etc.

View Axes

- **Visualise coordinate system** Visualises the active coordinate system in the graphics area using the position and style specified. The active coordinate system may be set from the **Geometry** tab of the model properties dialog.
- **Anchor point** Anchors the coordinate axes position to either a model coordinate, or a frame (window) coordinate.

- **Styles** Defines the style used to draw the coordinate axes and optionally sets the font used for axes labelling.

Deformations

- **Specify magnitude** Sets the maximum nodal deflection for the mesh as seen on the screen (in mm).
- **Specify factor** Sets an exaggeration factor for the maximum nodal deflection for the mesh.

Customise Startup Templates

Startup templates can be used to pre-load the Attributes Treeview with selected attributes for a particular analysis, set default mesh or material types, or define preferred colour schemes - to name just a few uses. User-defined startup templates are created by recording the setting of a variety of selections and then associating the recording with a template name.

Case Study. Creating a Startup Template

In this example a startup template will be created to define a material type, set it as the default, and also set user-defined display colours for a View window.

1. Run LUSAS and create a new blank model of any filename.
2. Select **File > Script > Start Recording** and enter **my_defaults** as the filename.
3. Select **Attributes > Material > Material Library**, choose **Mild Steel** and click **OK**.
4. In the Attributes Treeview, right-click on **Mild Steel Ungraded** and select **Set Default**.
5. In the Graphics Area, right-click and select **Properties**
6. Deselect **Use Windows colours**, select a **Black** background colour, choose a **Yellow** selection pen and click **OK** when done.
7. Select **File > Script > Stop Recording**
8. Exit from LUSAS.

Using the Startup Template

1. Run LUSAS and on the New Model dialog select the ... button adjacent to the Startup template drop-down list.
2. On the Customise Startup Template dialog select the ... button adjacent to the Script field. Enter **My defaults** into the Name field. Press the **Add** button to add the script and name to the table. Click **OK**.
3. On the New Model dialog, select **My defaults** from the Startup template drop-down list and click **OK**.
4. The startup template script will run setting all values to those previously chosen.

Toolbars and Toolbar buttons

The toolbars used on the Modeller user interface can be adjusted by using the **View > Toolbars** menu item. Toolbar groups can be turned on or off, new toolbar groups can be defined, customised toolbar groups can be created and user-defined toolbar buttons can added to the user-interface either to sit on a new toolbar group or to sit alongside existing buttons in an existing group. Toolbar manipulation is provided by a third-party and incorporated into LUSAS Modeller for general use.

Turning toolbar groups on and off

Toolbar groups are listed in the Toolbars dialog and may be turned on or off by checking each item in the list. The following options are also available:

- **Show Tooltips** shows a temporary description of the toolbar button when the cursor is moved over the button.
- **Cool Look** removes the raised button style to leave a 'flat' button.
- **Large Buttons** are not implemented in LUSAS modeller

Creating a new toolbar group

Favourite toolbar buttons (and any user-defined toolbar buttons) can be grouped together into a new toolbar group.

- Use the **New** button on the Toolbars dialog to create a new, named toolbar group such as 'Personal'. An empty 'Personal' button group will be added to the Modeller user interface.

Customising toolbar groups

Toolbar buttons can be added to exiting toolbar groups or placed on new toolbar button groups.

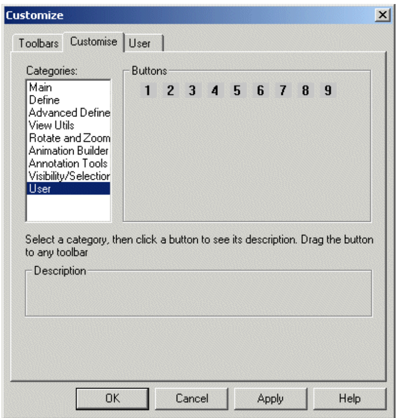
- Use the **Customise** tab to select a toolbar category and then, from the arrangement of buttons shown for that category, drag and drop a toolbar button into an empty part of the user interface. This can be repeated for as many buttons are necessary. As each button is added the button group will enlarge to accommodate it. Button groups can be 'docked ' alongside other button groups on the user interface by dragging and dropping into place.
- Buttons may be removed from toolbar groups by holding down the Alt key and then dragging the button into the View window.

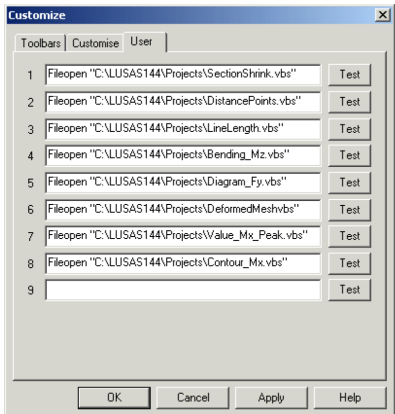
Creating user-defined toolbar button actions

Nine user-definable toolbar buttons are provided for linking to a specified action. With the Customise tab selected these can be seen if the User entry is selected in the Categories list.

Scripts can be recorded and specified to be played when a particular user-defined toolbar button is selected.

- In Modeller, use **File > Script > Start Recording** to record an action to be taken (such as the adding a Contours layer to the Treeview and the selecting a particular entity such as Force / Moment, and a component such as Mz, for example). Then use **File > Script Stop Recording** to save the script with a name such as Bending.vbs to a folder.
- Then use the **User** tab to define the action that a particular user-defined toolbar button should take when pressed. This involves inserting a text string to reference the script that was recorded. A typical entry would read: `Fileopen "C:\LUSAS150\Projects\Bending.vbs"`





Changing the user-defined toolbar button images

The default numbered user button images as supplied are held on a single bitmap image that is144 pixels wide and 15 pixels high. Each toolbar button image is created in sequence and occupies a region that is 16 pixels wide and 15 pixels high. The bitmap is named userToolbar.bmp (case-sensitive) and can be found in <LUSAS Installation Folder>\Programs\Config folder.

User toolbar button bitmap image as supplied

Example of user-defined toolbar button bitmap image

It is recommended that the supplied file is copied and renamed to userToolbar_supplied.bmp prior to making any changes to this supplied file. Changes made to the button images will be seen when LUSAS Modeller is next run.

Chapter 3 : File Types

LUSAS File Types

LUSAS can create or use a significant number of different file types. The main file types are summarised below.

- **Model Files** (.mdl) are created by LUSAS Modeller and are used to store all model definition information.
- **Solver (Analysis) Data Files** (.dat) are created by LUSAS Modeller during the tabulation phase. They contain the data required by LUSAS Solver to perform an analysis.
- **Solver Output Files** (.out) are text files which are created by LUSAS Solver. They contain an echo of the input data, details of any errors or warnings which have occurred during the analysis and tabulated results if requested.
- **Solver Results Files** (.mys) are created by LUSAS Solver and contain all of the analysis results for access by LUSAS Modeller. Results files are also occasionally referred to as plot files.
- **Solver Restart Files** (.rst) enable a nonlinear, dynamic, transient or viscous problem to be restarted after it has been chosen to stop an analysis during a solve.
- **Modeller Results Files** (.mrs) are created by LUSAS Modeller and are used to store the results cache when the model is saved. These files save assembled results and speed up the results processing of combinations. If necessary mrs files may be deleted to save disk space.
- **History Files** (.his) are created by LUSAS Solver and contain specified analysis results for access by LUSAS Modeller.
- **Script Files** (.vbs) contain a collection of LUSAS Modeller commands so that, when they are replayed, a sequence of operations may be carried out automatically. Script files can be recorded by LUSAS Modeller or edited directly using a text editor.
- **Session Files** (.ses) are created automatically every time LUSAS Modeller is run. They contain a record of all commands issued during a session.

- **Interface Files** (.dxf, .igs, .stp, .stl, .def, .nf) allow graphical structural information to be exchanged between LUSAS Modeller and external packages.
- **Command Files** (*.cmd) are used to import and export models from and to version 13 of LUSAS Modeller where only geometry features supported by version 13 Modeller will be exported.
- **Library files** (.csv) are created by LUSAS Modeller to store user-defined cross-sections. In addition to this library files, LUSAS Modeller also uses supplied libraries of pre-defined materials and section properties that cannot be edited by users. Another library file (with a .lmd extension) allows users to store and transfer attribute data (such as mesh, geometric, material, loading, and supports) and associated utilities data between models. See **Transferring Data Between Models** for details.
- **Picture Files** (.pic, .bmp, .jpg, .wmf) allow the contents of the Graphics Area to be saved in a standard file format. Picture files are used to subsequently display the information or, in conjunction with the LUSAS Expose program, to create files which may be printed or plotted. In addition to proprietary LUSAS picture files, screen content can be saved in BMP, JPG or WMF file formats.

All file types assume the default extensions that are given in brackets. When specifying filenames it is good practice to simply supply the filename without the file extension. LUSAS will then supply the correct extension for the file type being written which will ensure that existing files are not inadvertently overwritten by specification of the wrong file type.

Model Files

Model files contain all the information relating to the model, its current database and settings. The information is stored in an binary form and may only be accessed using LUSAS Modeller. A model file is not saved automatically, but LUSAS Modeller prompts on exit as a reminder to save changes to a model file. The following are accessed from the **File** menu:

- **New** Prompts to close an existing model file, and creates a new model file.
- **Open** Opens previously saved models. Modeller will prompt for confirmation before the currently loaded model is closed. See notes for opening models created prior to Version 15.0
- **Close** Closes the currently open model file.
- **Save** Saves the current model to disk at any time. See **Model file locations** for more details
- **Save As** Allows specification of an alternative filename. Selecting Save as for an existing model with results will create a new model file and, if necessary, a new folder but the existing results (and other intermediate files) will not be copied.

Opening a LUSAS model will automatically offer to open available results files, if found and if listed in the **Manage Results Files** dialog.

Opening pre-version 15.0 model files

Opening models created prior to version 15.0 which reference more than one results file will result in a different File Open dialog being presented from that displayed for models with one (or no) results file referenced.

See **Opening pre-Version 15.0 models that do not reference multiple results files** and **Opening pre-Version 15.0 models that reference multiple results files** for more details.

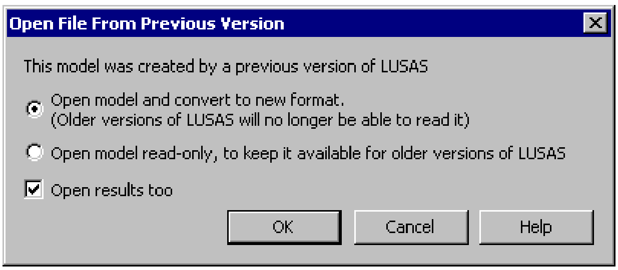
Notes

- When using Version 14 releases of Modeller, old Version 13 models can be opened if Version 13 is installed but this may take longer than usual as the files require conversion to the Version 14 format.
- When saving a model, disk space may be saved by deleting the mesh and faceting data using the **Advanced** button on the **File> Save As** menu item. This data will be regenerated when the model is reloaded.
- Section property data, that may be referenced in a model is saved in a section property library that may have a different folder location.

Opening pre-Version 15.0 models that do not reference multiple results files

In Version 15.0 the locations of all files referenced by a model are saved by specifying **model file locations**.

Opening models created prior to Version 15.0 (that do not reference multiple results files) will result in this dialog being displayed:



On the File open dialog displayed:

- **Open model and convert to new format** updates the model to the latest format. As a result, when it is saved older versions of LUSAS will no longer be able to read it. The filenames of the results files are updated to include a reference to the analysis name, and all results and associated model data files are saved according to settings made on the Modeller settings page of the **LUSAS Configuration Utility.**
- **Open model read-only** keeps the model available for viewing / editing in older versions of LUSAS. Filenames of the results files are unchanged.

- **Open results too** opens all associated results files for the model. Results files for models created prior to Version 15.0 will be listed in the Analyses Treeview beneath the Analysis 1 entry.

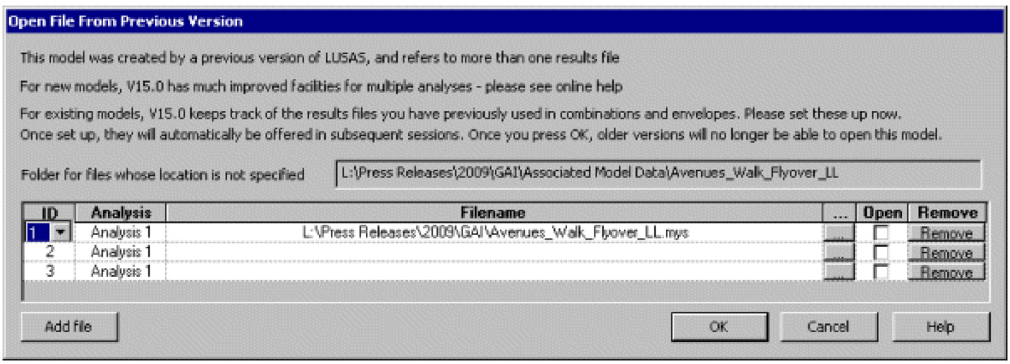
Opening pre-Version 15.0 models that reference multiple results files

In Version 15.0 the locations of all files referenced by a model are saved either with, or beneath the model folder by specifying **model file locations**.

Models created prior to Version 15.0 allowed combinations and envelopes to be created that referenced loadcases from more than one results file. Combination and envelope data was saved if the model file was saved. On re-opening such a model, the results files used in the combinations or envelopes had to be loaded on top of the model, in the order that they were first loaded, for the combinations and envelopes to be valid. This was because within the combination or envelope itself the results file was referenced by a file number (File: 1, File 2 etc) rather than by its name.

When opening pre-Version 15.0 models the path and filename of all results filenames associated with the model need to be specified if the files are to be automatically accessed by LUSAS Modeller each time the model is opened. To not specify a filename for the associated results files listed may invalidate the results obtained from a combination or envelope depending upon the loadcases included in them.

Opening a model created prior to Version 15.0, that contains combinations and envelopes that, between them, refer to more than one results file, will result in this dialog being displayed:



On this example dialog the model opened has three results files associated with it: results file **ID** 1, 2 and 3. The location of results file 1 is assumed to be the same folder as the model file location because prior to Version 15.0 a results file was (by default) written to the same folder and with the same name as the model filename. Results files 2 and 3 are automatically assigned to Analysis 1 so that they will be saved with that analysis when the model is saved. However, the location of any of the results files referenced by a combination or envelope saved in the model is, for historical reasons, unknown, hence the empty path for results files 2 and 3 in the Filename column. To ensure that any combinations and envelopes defined using these files remain valid in Version 15.0 and above, the location and filename of these 'unknown' results files need to be stated. To do this requires a knowledge or a record of the filenames and the order in which the results files were loaded in earlier versions.

So in summary:

- **ID** is the identification number of a results file that is referenced in an envelope or combination.
- **Analysis** The drop-down list selection made here determines whether a results file entry (that is added to the Analyses Treeview) is saved in that treeview when the model is saved.
    - Choosing **Analysis 1** (the default option) saves the results file entry in the Analyses Treeview when the model is saved.
    - Choosing **None** does not save the results file entry in the Analyses Treeview when the model is saved. These results files will have to be re-attached each time the model is read.
- **Filename** defines the location and filename of a referenced results file.
- The ... button can be used to correct any filename assumptions made by Modeller and to specify the location and filename of any results files that do not have a path specified.
- The **Open** check box (when checked) indicates that the results file is to be opened when the OK button is pressed.
- **WARNING:** The Remove button will remove the results file ID, Analysis and filename entries for that row from the grid and break the link to the results file in all combinations and envelopes that make use of that results file ID. All loadcase entries for the results file will be removed from all combinations or envelopes that refer to them.

The **Folder for files whose location is not specified** is set to be that of the created Associated Model Data folder for the model. The entering of relative paths and model names in the empty cells in the Filename column (as opposed to using the ... button which specifies full paths) mean each entry is evaluated with respect to this folder location.

Once links are established they are retained within Modeller and can be accessed and modified via the **File** > **Manage Results** files menu item which provides the same functionality.

Notes

- Results files added to the grid but not connected to any particular analysis (those assigned 'None' in the Analysis column of the grid) will be listed in the Analyses Treeview in sequence beneath the other analysis entries. This effectively provides the Version 14.7 working environment for old models within the new multiple analysis capability of Version 15.0, but means that the results files included in the Analyses Treeview can only be updated by opening and solving each relevant model (as was the case in pre-Version 15 releases).

Model file locations

The **LUSAS Configuration Utility** controls where files relating to a model (such as results files and intermediate files) are saved.

For each model the location of files that are created in the course of a building it and running the various analyses associated with it can be specified as being:

- **Within a folder for each model, within a folder called "Associated Model Data"** This is the default setting. Note that the model file itself does not reside within the folder created.
- **Within a folder with the same name as the model** Note that the model file itself does not reside within the folder created.
- **Within the same folder as the model** (This is Version 14.7 behaviour)
- **To a specified folder** where the location can be defined by internal LUSAS tokens and ordinary text. For example, the token**%ModelLocn%** is substituted with the working current folder for the model; and **%ModelName%** is substituted with the model's file name. Relative links (e.g. ..\ to use the parent folder) can alternatively be specified.

Model Folders explained

- **Associated Model Data** is the default folder name and container for all files (apart from temporary files) that are created by LUSAS in the course of modelling or solving an analysis. This will include results files (.mys), solver output files (.out) and solver data files (.dat) for instance. Within this folder two additional subdirectories are created called 'Backups' and 'Sessions':
  - **Backups** A copy of the original model is preserved in this folder when the model is loaded for editing. The number of revisions of a model that can be saved to this folder can be set on the Backups tab of the Modeller Properties dialog. The default number of revisions saved is two, but this can be modified. Whenever the model is saved, the previous revision is saved here and (assuming the number of revisions has been reached) the oldest revision is deleted.
  - **Sessions** All session files (.ses) are stored in this subfolder. The model recovery file (.rcv) is also stored here.

Notes

- In addition to the backup file locations stated it is also possible to maintain another (current) copy of the model file in another location. See the Backups tab on the Modeller Properties dialog.
- The folder location of temporary files, created in the course of modelling with LUSAS, is specified on the **Temporary Files** page of the LUSAS Configuration Utility.

- The Windows operating system limits the length of a path to a file to 260 characters. See **File and folder naming in LUSAS**.

File and folder naming in LUSAS

The Windows operating system limits the length of a path to a file to 260 characters. For most modelling situations this should not prove to be a problem but for some modelling and analysis applications within LUSAS, notably when carrying out **vehicle load optimisation** (VLO) analysis, this limit may conceivably be met. This will typically occur if very long model names are used in conjunction with the use of very long names for attributes and analysis runs that are used in VLO analysis.

In the most extreme case files relating to a model can be saved beneath a model folder in a folder named 'Associated Model Data' (see **Model File Locations** for details), in a folder named after the model name. VLO folder and filenames based upon user-defined file names are then created beneath that folder. This adds a VLO output folder (based upon the model name), and a concatenation of the VLO run name, the influence attribute name, and the coordinate value of the position on the model to which the influence attribute has been assigned, to make up the full path to a particular file.

A typical 'longest' path name for a file would therefore be constructed from:

<userspath> / Associated Model Data / <usersmodelname> / <usersmodelname>_VLOoutput / <usersVLORunname> _ <usersInfluenceName> - (0.00000, 0.00000, 0.00000) - (element 99999, element 99999).dsp

Allowing for those parts of a path that are fixed in nature (such as the predefined named folder structure and the coordinates and element name) leaves approximately 150 characters with which to define a model name (noting that the model name appears twice in the path), a VLO analysis run name, and an influence attribute name.

Solver Data Files

Solver data files contain the data required by LUSAS Solver to perform an analysis. Solver data files (.DAT) are automatically created by LUSAS Modeller during the tabulation phase which is triggered either by pressing the Solve  button, or by selecting any of the other Solve or Solve now options available for the analysis entries in the Analyses Treeview.

A Solver Data file can also be created manually by using the **File> Export Solve datafile...** menu item.

All methods produce a data file in readable ASCII text format and the file may, if necessary, be modified with a standard text editor. The format of the analysis data file is described fully in the *Solver Reference Manual*

Solver Output Files

When an analysis is performed by LUSAS Solver it creates a text output file which has an echo of the input data, details of all errors diagnostics and warnings and tabulated results.

Solver Restart Files

When an analysis is performed by LUSAS Solver, data can be written to a restart file to enable a nonlinear, dynamic, transient or viscous problem to be restarted after it has been chosen to stop an analysis during a solve.

Solver restart file generation needs to be specified in Modeller prior to running an analysis by choosing the File> Export Solver datafile... menu item, accessing the Advanced Solution Options dialog and selecting Generate restart file.

The restart output facility enables failed or terminated analyses to be restarted from the last saved restart output dump. This is particularly useful for a number of situations:

- When the termination of the analysis was due to a failure of the solution process rather than a problem with the modelling of the structure the solution may be restarted from the last converged increment with a different or modified solution strategy. For example, a failed increment may be restarted under either constant load or arc-length control.
- To break up larger, longer runs of 100s or 1000s of increments or time steps into several sequential runs to reduce the size of each results file created. This may also be done simply to safeguard against any potential interruption to the solve that may otherwise mean starting the analysis again right from the beginning.
- To solve to a certain point or stage in an analysis, and then continue from this point with several different further loading configurations that each be considered in separate parallel restart runs.

Restarts are not currently supported by LUSAS Modeller and hence must be defined directly in a LUSAS Solver data file. See the *Solver Manual* for more details.

Solver Results Files

Results files When the Solve button is pressed in LUSAS Modeller one or more analyses are carried out by LUSAS Solver and a results file will be produced in a specified Model folder. LUSAS results files, or plot files as theyare sometimes referred to, have a **.mys** extension.

The information in a Solver results file is stored in a binary format and may only be accessed using LUSAS Modeller. The results file will contain the results of the analysis and sufficient model information to process the results. Full details of the finite element mesh (nodes and elements), material and geometric property numbers, support positions and equivalent nodal loads are stored in the results file so that results processing can be carried out without a model file if desired.

Opening available results files for a model

When a model is opened, Modeller will attempt to open all available associated results files for that model. The **File > Open Available Results Files** menu item also exists for the same purpose. Results files are loaded on top of the current model and results loadcases are added to the relevant Analysis entries in the Analyses Treeview.

Results files that are associated with a model can be seen by selecting the **File > Manage Results files** menu item.

Opening a results file without a model being open

A results file can be opened in isolation from a model by using the **File** menu:

- ❑ **Open** By changing the Files of type drop-down list, results files (.mys) can be selected. This opens a results file in isolation from a model. Modeller will prompt for confirmation before the currently loaded model is closed.
- ❑ **Close** Closes the currently open results file.

Opening a results file in isolation from a model does not provide access to any generated post processing loadcases such as envelopes, combinations etc that are saved with the model file.

Closing results files

- All open results files can be closed at the same time using the **File > Close All Results Files** menu item.
- Results files can also be closed individually by right-clicking on a loadcase name in the Analyses Treeview and selecting the **Close file** menu item.

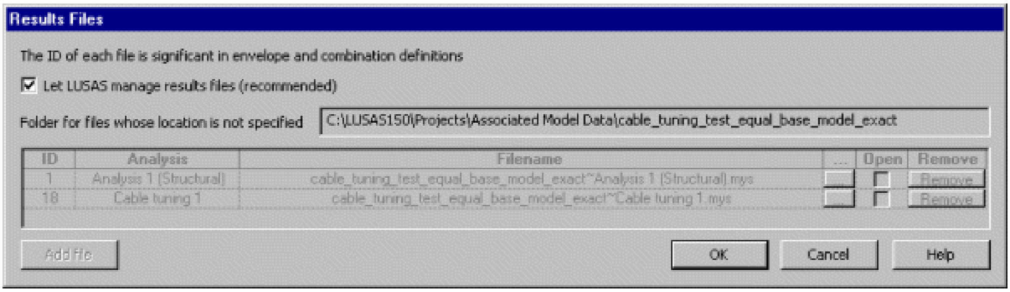
Notes

- If a model has been re-meshed since the results file for that model was created the cannot be loaded. Instead the model must be re-solved to create and automatically open the results file.
- If a model has been modified in same other way (than just being re-meshed) an option to load or not load the results is made available.
- For transient and nonlinear analyses the frequency that LUSAS Solver writes results to the results file is specified when defining the **analysis control**. If this is not specified results are saved in the results file on every time step or load increment.

Manage Results Files

In Version 15.0 of LUSAS the locations of all intermediate and results files created during the building and solving of a model can be specified to be created either within a folder for each model, in the same folder as the model, or a specific location dependent upon the **Model File Location** used.

The **File > Manage Results Files** menu item displays a grid that lists all results files that are associated with the open model.



It is recommended to always let LUSAS manage results files.

The **Folder for files whose location is not specified** is set to be that of the created Associated Model Data folder for the model. The entering of relative paths and model names in the empty cells in the Filename column (as opposed to using the ... button which specifies full paths) mean each entry is evaluated with respect to this folder location.

In the results file grid:

- **ID** is the identification number of a results file.
- **Analysis** This column shows the name of the analysis for which the results file is applicable. The drop-down list of analyses is populated according to the analysis types present in the model.
  - Changing an analysis name associates the results file with a particular analysis in the Analyses Treeview. It is not recommended to do this.
  - Choosing **None** does not save the results file entry in the Analyses Treeview when the model is saved.
- **Filename** defines the location and filename of a referenced results file.
- The ... button can be used to correct any filename assumptions made by Modeller and to specify the location and filename of any results files that do not have a path specified.
- The **Open** check box (when checked) indicates that the results file is to be opened when the OK button is pressed.
- **WARNING:** The Remove button will remove the results file ID, Analysis and filename entries for that row from the grid and break the link to the results file in all combinations and envelopes that make use that results file ID. All loadcase entries for the results file will be removed from all combinations or envelopes that refer to them.

If a results file is to be added to the grid the **Let LUSAS manage results files** check box should be unchecked and the **Add file** button used to add an extra row to the grid to enable an Analysis reference, file location and other settings to be made.

Notes

- Results files added to the grid but not connected to any particular analysis (those assigned 'None' in the Analysis column of the grid) will be listed in the Analyses

Treeview in sequence beneath the other analysis entries. This effectively provides the Version 14.7 working environment for old models within the new multiple analysis capability of Version 15.0, but means that the results files included in the Analyses Treeview can only be updated by opening and solving each relevant model (as was the case in pre-Version 15 releases).

Modeller Results Files

Modeller results files have a **.mrs** extension and are saved whenever the model is saved. Modeller results files contain loadcase combination and envelope component results that are calculated by LUSAS Modeller.

A Modeller results file speeds-up the assembly of the selected results within LUSAS Modeller since the component results are only calculated once for the selected combination and envelope results components. This means that when setting each combination or envelope active for viewing results the software does not have to re-calculate the results for that results component. However, selecting a combination or envelope result component that is not pre-calculated will cause the results for all envelopes of envelopes and combinations to be re-calculated.

The information in a Modeller results file is stored in a compressed binary format and may only be accessed using LUSAS Modeller.

History Files

History files are used to output the named variables, and selected node and element results from LUSAS Solver in an ASCII format. Specification of the node and element numbers to be output to this file is defined from the **File> Export Solver datafile...** menu item. The output frequency for incremental analyses is controlled using **analysis control**. The results stored in the time history file can be accessed for **graphing**.

The history file format consists of a header section with a title, list of named variables, type of nodal results and type of element results, followed by the results for each time/increment number. The format is shown below. **Note.** Due to space limitations, the number format has been adjusted. Standard history files will contain accuracy to machine precision.

The named variables, selected nodal results and selected element Gauss point results will be output for each time step or increment as specified in the analysis control.

Script Files

Script file may be created and used to store a sequence of commands for later playback. Script files are created in Visual Basic Script (.VBS) format and are particularly useful for storing combinations of commands which are used frequently. Uses include consistent reproduction of screen images for use in reports and use with **startup templates** to pre-load the Attributes Treeview with selected attributes for a particular analysis, set default mesh or material types, or define preferred colour schemes.

Scripts are normally run by opening the file in a file browser but user-defined **toolbar buttons** can be set-up to run scripts.

Script file manipulation is controlled from the **Files> Script** menu item. The following functionality is available:

-  **Run Script** An existing script file is replayed by choosing it from the **Open** dialog.
- **Start Recording** creates a script file. If a non-default file extension is specified or if the file already exists you will be prompted for confirmation before proceeding. Existing script files can be appended to if required. While recording all attempted commands are logged to the script file using the LUSAS scripting language.
- **Stop Recording** closes the script file.

Session and Recovery Files

Each time Modeller is run or the model is saved a new recovery file is created in the current working folder. This recovery file is named after the model name with the **.rcv** extension. Every attempted command, whether entered from the user interface or via the command line, is logged in this file using the scripting language. When the model is saved or the user exits Modeller the recovery file is renamed to a session file with an incrementing version number.

Picture Files

About Picture Files

LUSAS picture files may be used for storing graphical information for subsequent conversion to an alternative file format using the LUSAS picture file utility program, Expose.

LUSAS picture files are stored in readable ASCII text format. The individual picture file records use the following general format:

```
code, r1, r2, r3, r4, i1, i2, i3
```

The information is stored in packets of data as defined in the following table.

Code	Function	Parameters	Description
1	Move	r1, r2	Moves to the drawing location specified by the x (r1) and y (r2) coordinate (mm).
2	Draw	r1, r2	Draws a line from the current position to the drawing location specified by the x (r1) and y (r2) coordinate (mm).
3	Symbol	r1, r2, r3, r4, i1	Plots a LUSAS built-in symbol at a specified screen position. (0-Square,1- Circle, 2-Triangle, 3-Double Triangle, 4-Diamond, 5-Cross, 6-Boxed Cross, 7-Asterisk, 8-Horizontal Arrow (origin at apex), 9-Horizontal Arrow (origin at base), 10-Vertical Arrow, 11-Vertical Line, 12-X, 13-Y, 14-Z, 15-Barred X
4	Character	r1, r2, r3, r4, i1	Plots an ASCII character at a specified screen position with: x coordinate (r1), y coordinate (r2), rotation angle in degrees (r3), character height in mm (r4) and ASCII character code (i1).
5	Colour	r1, r2, r3	Percentage colour content with: red % (r1), green % (r2) and blue % (r3).
9		r1, r2, r3, r4, i1	Starts a colour-filled multi-sided polygon with: number of vertices (i1). Real numbers (r1-r4) are not used.
0		r1, r2	Creates a polygon vertex with: x and y coordinate (r1-r2). Must be used in conjunction with and appear immediately after code 8 or 9 above.
10	Clipping Rectangle	r1, r2, r3, r4	Sets current clipping rectangle x1-r1, y1-r2, x2-r3, y2-r4
20	Multi-Line Text	r1, r2, r3 ,r4 ,i1 ,i2	Defines mutli-line text located at x-r1, y-r2, rotation-r3 (degrees), size-r4 (mm), alignment-i1 (0-top left, 2-top right, 6-top centre, 8-bottom left, 10-bottom right, 14-bottom centre, 16-middle left, 18-middle right, 22-middle centre, 24-baseline left, 26-baseline right, 30-baseline centre), nLines-i2 (number of subsequent lines of text)

Saving Picture Files

- Pictures may be saved using the **File> Picture Save** menu item.
- Note that views of a LUSAS model can also be saved for use in other applications as BMP, JPEG, or WMF files using the **File> Picture Save** menu item. For more information see **Printing and Saving Pictures**

Print Files

When using the print result wizard the output may be re-directed to a **Print File**. A print file has a **.prn** extension. The opening and closing of print files is controlled using the **Files> Print File** menu item. The following facilities are available:

- **File> Print File> Open** The print file is opened by specifying a valid filename. LUSAS will prompt for confirmation to proceed if the specified file already exists or if a non-default file extension is used.

- **File> Print File> Close** The print file may be closed at any time. With no print file open, printed output will be directed to a text output display window.

Notes

- Output to the text window can be directed to a log file. See **Text Window** for more details.

Interface Files

Interface files are used to transfer external modelling or material data into and out of LUSAS Modeller. The full model or a selected portion of a model can, dependent upon the file format chosen, be exported to a chosen interface file.

The currently supported list of interface file formats is:

- **CMD** (**.cmd**) Format for import of LUSAS Modeller model files saved as command (CMD) files in previous versions of LUSAS.
- **Solver Data Files** **(.dat)** LUSAS Solver data files (used to import or node and element data)
- **DXF** (**.dxf**) AutoCAD Drawing eXchange Format.
- **IGES** (**.igs**) Initial Graphics Exchange Specification. Format for import and export of geometry data.
- **LMS CADA-X** (**.nf**) Model description and modal data exported to a file that can be read by the LMS software.
- **NASTRAN Bulk Data files** **(.bdf, .dat)** (used to import node and element data)
- **ANSYS cdb files (.cdb)** (used to import node and element data)
- **Abaqus input files (.inp)** (used to import node and element data)
- **PATRAN** (**.def**) Neutral file format for inputting phase I geometry information and outputting phase II mesh information
- **STEP** (**.stp**) STandard for the Exchange of Product data.
- **STL** (**.stl**) Stereolithography data files.

In addition to these interface files, search area topology files (.inf) can be imported into LUSAS for graphical cross-checking. These files, which contain details of search areas used by Autoloader, are converted into geometric line and surface data in LUSAS

- **Search area topography** (**.inf**) Autoloader generated search areas.

Summary of Interface File Import / Export Capability

Interface file name and extension	Import file into LUSAS?	Export file from LUSAS?
CMD (.cmd)	**YES**	**YES**
SOLVER Data File (.dat)	**YES**	NO
DXF (.dxf)	**YES**	**YES**
IGES (.igs)	**YES**	**YES**
LMS CADA-X (.nf)	**YES**	**YES**
NASTRAN Bulk Data Files (.bdf, .dat)	**YES**	NO
ANSYS (cdb)	**YES**	NO
ABAQUS (.input)	**YES**	NO
PATRAN (.def)	**YES**	NO
STEP (.step, .stp)	**YES**	**YES**
STL (.stl)	**YES**	**YES**

Notes

- See **Importing Geometry Data** for details of how to import interface files.
- See **Importing Mesh Data** for details of how to import finite element data files created either by the prior running of an analysis in LUSAS or by importing interface files from other supported third-party software applications
- DXF, IGES and STEP files often contain much more detailed information than is required to create a finite element model, so a certain amount of model tidying should be expected after carrying out an import.

Transferring Data Between Models

The library browser allows users to transfer attribute data (such as mesh, geometric, material, loading, and supports) and associated utilities and report data between models. Selected data is saved to a LUSAS Model Data file (*.lmd). The browser is accessed using the **File** > **Import/Export Model data** menu item. Any number of library model data files may be created.

Transferring data

The library browser displays a list of the attributes and utilities for the currently open model, alongside a list of attributes and utilities that have been saved in the selected LUSAS model data library file.

- Selected data can be transferred from the model to the library by using the check boxes on the Model side of the dialog to select the attributes for transfer and then pressing the **Export from model to library** radio button.
- Selected data can be transferred from the library to the model by using the check boxes on the Library side of the dialog to select the attributes for transfer and then pressing the **Import from library to model library** radio button.

Clash detection when transferring

Name clash detection checks are carried out to ensure that data is not unknowingly over-written.

Clashes will occur when the same attribute or utility name (with the values or settings) is used in the model as in the library file. If clashes occur during the transfer process the attribute name or utility in the library file can either be overwritten or the offending attribute(s) can be renamed in the Attributes Treeview panels of the model prior to a subsequent re-transfer to the library being attempted.

When transferring data from a model data library to the model, clashes will occur when the same attribute or utility name (with the values or settings) is used in the library as in the model file. If a clash is detected the following name clash resolution options are provided:

- ❑ **Overwrite model item with the one from the library**
- ❑ **Do not import this item** keeps model and library items as they were
- ❑ **Append unique name extension** appends a user-defined extension to the attribute name
- ❑ **Prepend unique name prefix** gives the attribute name a user-defined prefix
- ❑ **Use this choice for all other name clashes** applies one of the previous selections to all other clashes of names.

Deleting items

If any model attributes, utilities or reports are selected using the check boxes available pressing either the **Delete from model** button or the **Delete from library** button will delete those items from their respective sources. As a result, the Library Browser dialog also provides the means to carry out multiple deletions of attribute/utilities data from a model (something not possible currently via the main Attributes and Utilities Treeviews).

Notes

- Note that whilst the Attributes and Utilities Treeview panels may show data sorted by id number (default) or by name (optional) the listing of entries in the File Library browser is always by name. If necessary, model treeview contents can be sorted to match by selecting the menu item **Sort by name** from the context menu for a Treeview tab.

- If one attribute (such as a composite definition) relies upon another attribute (such as a set of material properties) it is not possible to transfer one without the other(s).
- If variations have been used in the definition of an attribute then transferring that attribute from the model to the library (and vice-versa) will also transfer the relevant utilities datasets describing the variations used.
- Attributes should not be edited in the library file.

Importing Interface File Data

Geometry and mesh data from supported interface files can be imported using the **File> Import...** menu item. When a file is selected the **Advanced** button can be used to specify import parameters. After all feature entities have been imported, a feature merge will be carried out according to the merge setting defined under Model properties.

File Import Options (Advanced)

Only those options applicable to the file being imported will be available for selection.

Option	Description	Default
Translate annotation type geometry entities	Include entities marked as annotation.	False
Translate blanked entities	Include entities marked as blank.	False
Merge trimming lines	Attempts to merge the trimming lines of trimmed surface entities.	False
Delete dependent geometry	Will delete geometry objects created from entities marked as dependent.	True
Delete points not defining lines	All points not connected to a line are removed.	True
Delete lines not defining surfaces	All lines not connected to a surface are removed.	True
Delete unconnected lines	Deletes unconnected lines that do not define any surfaces.	True
Use domain space trimming curves	Use domain space trimming curves in preference to model spacing trimming curves.	False
Lock the mesh post import	Locks the mesh following import to ensure it is not changed unintentionally.	False
Model is solid volumes	If selected, fills in any missing data to create a solid volume.	False
Coalesce surfaces	Removes similar lines from adjoining surfaces to simplify the model.	False

Option	Description	Default
Coalese volumes	Removes similar surfaces from adjoining volumes to simplify the model.	True
Create material groups	Create named groups for features in the data file having the same material property	False
Maximum number of groups	Maximum number of groups permitted to be created from material property types	500
Merge geometry post import	Merges geometry within the general specified merge tolerance.	True
Minimum line length	Facets containing lines of less than this specified length will be ignored.	False
Minimum angle degrees	Facets containing lines of less than this specified length will be ignored.	False
Pre-translation scale	Scaling factor applied to all entities before translation.	1.0
Radius of curvature to length ratio	Minimum allowable radius of curvature to line length ratio in surface trimming.	1%
Parsing error limit	Import terminate after specified number of error (0 indicates no limit).	2
Entity types to exclude	List of entity type numbers to ignore (if checked), entities of these type numbers will not be translated unless they define entities that are to be translated.	Ignore 106 (copious data) and 108 (plane surface)
Drawing layers to process	Allows selection of named layers when importing DXF or IGES data.	None

Importing Solver datafile data

Geometry and Mesh data that has been written to a Solver data file may be imported into Modeller for particular types of analysis.

- Point, Line, Surface and Volume geometry data can be imported to create a feature-based geometry model using the **File> Import...** menu item.
- Element data can be imported to create a mesh-only model by using the **File> Import** menu item. When a file is selected the **Advanced** button can be used to specify import parameters. See **Model Types** for more information.

Geometry and mesh data from supported interface files can be imported using the **File> Import...** menu item. When a file is selected the **Advanced** button can be used to specify import parameters. After all feature entities have been imported, a feature merge will be carried out according to the merge setting defined under Model properties.

Importing Mesh Data

Mesh-only models can be created by importing finite element data files created either by the prior running of an analysis in LUSAS or, more usually, by importing data files from other supported third-party software applications.

During the mesh import process, Modeller creates separate Groups for each element type encountered. For models created from LUSAS data file these will be familiar LUSAS element names. For models created from other software they will be the names used within that system, whatever they may be. See **Mesh-only models** for more information.

After import the **vertical axis** for the model may need to be defined to ensure correct isometric viewing and loading of the model.

File Import (Advanced option)

Only those options applicable to the file being imported will be available for selection.

Option	Description	Default
Create material groups	Create named groups for elements in the data file having the same material property. A maximum number of groups can be specified and if more groups are created the number specified only the most common element groups in the model will be created. This option is for use with LUSAS Solver data files and Nastran bulk data files only.	False

Exporting Model Data

Model data can be exported to a chosen interface file format by using the **File> Export** menu item.

Export options

Depending upon the export file format chosen all, or some of the following export options will be available for a selected analysis:

- **Current window**
- **All**
- **Visible**

For exporting to a LUSAS V13 CMD file an analysis entry must additionally be chosen.

And the following features may be exported:

- **Geometry and Mesh (excluding volumes)**
- **Geometry and Mesh (including volumes)**
- **Geometry Only**
- **Mesh only (excluding volumes)**
- **Mesh only (including volumes)**
- **Nodes Only**
- **Deformed mesh factor** (only accessible when a results file is loaded) The mesh may be tabulated with node coordinates computed from the deformations in the active loadcase multiplied by a specified factor. This is useful when the deformed mesh caused by one analysis is to be used as the starting point for a further analysis.

DXF Interface Files

The AutoCAD Drawing eXchange Format or DXF file, as it is more commonly known, can be used to import and export data to and from LUSAS.

DXF Import

DXF files are imported using the **File> Import...** menu item.

DXF entities supported by the LUSAS DXF import facility are listed in the table below.

DXF Entity	Imported as LUSAS Feature
POINT	Point.
LINE	Straight Line.
3DLINE	Straight Line.
ARC	Arc Line.
CIRCLE	Two arc Lines.
POLYLINE	Spline Line.
SOLID	Straight-edged Surface.
3DFACE	Straight-edged Surface.
TRACE	Straight-edged Surface.
POLYGON MESH	Multiple straight-edged Surface.
POLYFACE MESH	Bicubic Surface.
EXTENDED ENTITIES	Not supported.

Tip. Units and entity orientation can be modified by defining a local coordinate and making this active before importing. For example, the units may be changed from mm to m during conversion by defining a scale local coordinate with X, Y and Z scale factors of 1e-3. The entity orientation may be changed from landscape to portrait with the aid of an XY rotation local coordinate with an angle of 90 degrees.

Notes

- The amount of information which may be transferred via the DXF file is limited due to limitations in the DXF file format (for example, a volume cannot be expressed in standard DXF data).

- AutoCAD version 13 uses DXF extended entities for some items. LUSAS does not support import of extended entities and will warn to this effect if an AutoCAD version 13 DXF file is detected.

DXF Export

A DXF interface file may be created from LUSAS for use in an external program using the **File> Export...** menu item.

LUSAS attributes are converted into their equivalent DXF entity. Control over the amount of information exported is provided, i.e. All or Visible features and/or mesh may be specified. This is valid for both pre-processing model files and results files. The following parameter is available on the export dialog to control creation of DXF files:

- ❑ **Level Indicator** indicates whether **Geometry Only**, **Mesh Only** or **Geometry and Mesh** are to be exported. The level indicator is only required when a model file is open and features are active. When no model is loaded, such as during post-processing, only the mesh is exported. Additional options are available to include Volume mesh entities in the export process. Only element faces are exported when exporting Volume feature mesh records.

LUSAS feature types supported by the DXF export facility are listed in the table below:

LUSAS Feature/Mesh	Exported as DXF Entity
Feature LINE (straight)	3DLINE
Feature LINE (arc)	ARC
Feature LINE (spline)	POLYLINE
Feature SURFACE (straight-edged)	3DFACE
Feature SURFACE (general curved)	3DLINE/ARC/POLYLINE
Mesh LINE (linear or quadratic edge)	3DLINE
Mesh SURFACE (linear or quadratic face)	POLYFACE MESH
Mesh VOLUME (linear or quadratic face)	PLOYFACE MESH

Notes

- The exporting of models generates DXF files containing structural information only. This facility is not intended for exchanging graphical information, for this purpose picture files should be used.

- Only the element faces are exported when exporting volume feature mesh records
- For further information on the DXF file format, users are referred to the AutoCAD © Reference Manual.

IGES Import / Export

IGES files are imported from the **File> Import...** menu item. When a file is selected the import process may be controlled from the **Advanced** button by specifying import parameters.

LUSAS Model geometry may be exported to IGES using the **File> Export...** menu item.

Notes:

- IGES data is made up of a number of discrete surfaces. These need to be merged together to create volumes which can then be meshed.
- When IGES import is used the option to crate hollow volumes is automatically invoked.
- Any active local coordinates will be ignored.
- All IGES annotation lines and font data is ignored.
- The IGES interface only supports fixed length ASCII IGES files.
- All curve and surface geometric type entities are translated into LUSAS Modeller.

Supported IGES Entities:

Entity	Description
100	circular arc
102	composite curve
104	conic arc
106	copious data
108	plane surface
110	straight line
112	parametric spline curve
114	parametric spline surface
116	point
118	ruled surface
120	surface of revolution
122	tabulated cylinder
124	transformation matrix
126	rational B-spline curve
128	rational B-spline surface
130	offset curve
140	offset surface
141	trimming line of bounded surface
142	trimming line of parametric surface
143	trimmed bounded surface
144	trimmed parametric surface
186	B-rep volume
502	Manifold Brep vertex list
504	Manifold Brep edge list
508	Manifold Brep loop
510	Manifold Brep face
514	Manifold Brep shell

Note. Only those found in the selected IGES file are displayed in the exclusions list.

LMS CADA-X Files

This data format is used to export data from LUSAS Modeller to the LMS Modal Analysis Suite of Software.

- When exporting modal data or element matrices, a check for a results file is made. If no results are available, the `moddat` parameter is deselected. The modes for export dialog is only displayed if modal data is requested.
- The mode shapes input dialog will expect the mode shape numbers to be entered in the same manner as the SET MODAL MODES command.
  The format is: *resultfileID:mode1;mode2;mode-a**T**mode-b**I**increment* ... etc.
- Additional system parameters are required to deal with problems encountered when reading the neutral file into LMS. These variables should be set as shown below in the Modeller start-up file. The variables are defined as follows:
- **LMSTKV** when set forces the THICKV element property keyword to accept the #NO_DEF purpose code to describe the property type instead of MEMBRANE, PLANESTRAIN, SHELL, PLATE and SHEAR. This is due to an error in the LMS parser.
  LMSTKV should be set to 1 to have #NO_DEF output to the neutral file.
  LMSTKV should be unset to have normal codes output to the neutral file.
  The default setting is 0.
- **LMSDMP** forces the export routines to only output SPRING and MASS elements and properties for joints instead of SPRING, MASS and DAMPER elements and properties. This is to work around a limitation in the LMS parser which will not interpret the DAMP or DAMPER keywords.
  LMSDMP should be set to 3 to have 3 properties/elements per joint output.
  LMSDMP should not be set to have 2 properties/elements per joint output.
  The default setting of LMSDMP is 2 (2 properties/elements per joint).
- **LMSPRC** allows the user to specify single or double precision real number output in the neutral file. Using this parameter can reduce the size of the neutral file. The differences between the two types of precision is as follows:

  Single precision: +n.nnnnnnE+ee
  Double precision: +n.nnnnnnnnnnnnnE+ee
  Currently LMSPRC can take the following values:
  LMSPRC = 1 Single precision format (default)
  LMSPRC = 2 Double precision format
  LMSPRC = 3 Double precision format on a single precision machine.

LMS Export

- **Option 290** must be set before tabulating a model to ensure element matrices are transferred from LUSAS.
- **Option 290** allows you to instruct LUSAS to output the element STIFFNESS and MASS matrices to the .mys plot file. The volume of data transferred can be substantial so this option is turned off by default.
- **Nodal Freedoms** LMS supports only six degrees of freedom: X, Y, Z, Rx, Ry and Rz. If unsupported freedoms are encountered a warning message is issued. If this section is not present it will **not** affect the translation of the neutral file into LMS as long as no node statement contains a reference to a #Frnnn freedom label.

- **Node Coordinates** The node co-ordinates are written with the co-ordinate system omitted implying the use of the global co-ordinate system. The Modeller node labelling scheme is preserved during the export process.
- **Material Properties (mdl file)** Isotropic, 2D Anisotropic and 2D Orthotropic materials are supported. 3D material properties are exported as ANISO3D, however the values are read but not used by LMS. Joint properties are output in the element property section of the neutral file.
- **Material Properties (mys file)** Material properties are transferred from LUSAS to Modeller. If no material properties are detected in the LUSAS mys file, then dummy material properties are set-up and a warning is issued. Materials can be used in LMS to group common elements together. The dummy properties allow the other model description entities (element topology) to be read by the LMS parser.
- **Element Properties (mdl file)** The LMS element properties supported are PBAR, STIFF, PMASS, BEAMG and THICKV. The corresponding Modeller geometric properties are mapped into the expected LMS format. Supported Modeller property types are as follows: Bar/Link, Beam, Membrane/Plate/Shell. Beam and joint eccentricities are ignored. Checks for unsupported element properties are made and a warning is issued if any are found. Unsupported element properties are not output.
- **Element Properties (mys file)** Modeller geometric properties from the mys file are transferred and output into the LMS neutral file. The same constraints as for the mdl file apply.
- **Element Topology** Elements are output in element type order. Material properties must be specified for all elements, but LMS element properties for solid elements are optional. Checks for unsupported elements are made and a warning is issued with the unsupported elements not written to the neutral file. Beam elements which have end freedoms released are output to the neutral file, but the node freedoms are not transferred as they may not be valid for all connections to a node. Supported elements are shown in the table below.
- **Eigenvectors and Frequencies** Node displacements for all nodes specified in the model description are output to the neutral file in the global co-ordinate system. When mode shapes are read by LMS, the mode shape numbers will not necessarily be the same unless all eigenvectors are exported. Modeller mode shape numbers are preserved during an export, but are not preserved when read into the LMS software.
- **Element Matrices** The element matrices (stiffness and mass) are output in element type order. There are three sections required to define a matrix. These are MATSHP, MATVAL and MATDEF.

By definition elements of the same type have the same matrix shape, therefore for each element type there is only one MATSHP keyword. However, each element matrix contains different values and hence gives rise to one MATVAL and MATDEF statement per element matrix. To reduce the amount of matrix data output to the neutral file, and to keep its size to a minimum, only the non-zero (active) columns of the element matrices are processed.

Supported Elements

The following elements are supported by LMS.

Bars	Beams	Plates, Shells, Membranes		Solids			Joints
		Triangular	Quadrilateral	Tetra	Penta	Hexa	
BAR2	BEAM	TPM3	QPM4	TH4	PN6	HX8	JF3
BAR3	BMS3	TPM6	QPM4M	TH4E	PN6E	HX8M	JPH3
BRS2	BRP2	TPM3E	QPM4E	TH15	PN15	HX20	JRP3
BRS3	GRIL		QM8				JNT3
		TF3	QF4				JNT4
		TTF6	QSC4				
		TRP3	QTF8				
			RPI4				
		TS3	QSI4				
		TTS3	QTS4				
		TTS6	QTS8				
		TSM3	SMI4				

NASTRAN BDF and DAT Import

Data from NASTRAN Bulk Data Files (.BDF) or DAT (,dat) files can be imported to create a feature-based geometry model using the **File> Import...** menu item or a mesh-only model by using the **File > Import Mesh** menu item. When a file is selected the **Advanced** button can be used to specify import parameters. See **Model Types** for more information.

ABAQUS Input File Import

Data from Abaqus Input Files (.inp) can be imported to create a feature-based geometry model using the **File> Import...** menu item or a mesh-only model by using the **File > Import Mesh** menu item. When a file is selected the **Advanced** button can be used to specify import parameters. See **Model Types** for more information.

ANSYS CDB File Import

Data from ANSYS CDB Files (.CDB) can be imported to create a feature-based geometry model using the **File> Import...** menu item or a mesh-only model by using the **File> Import Mesh** menu item. When a file is selected the **Advanced** button can be used to specify import parameters. See **Model Types** for more information.

PATRAN Interface Files

About PATRAN

The PATRAN neutral file contains the full finite element model information. The Neutral file is split into two data categories: Phase I contains the definition of the geometric entities, and Phase II contains all of the finite element (node and element) information.

PATRAN Import

PATRAN files are imported using the **File> Import...** menu item.

Phase I data (geometric entities) is read from the PATRAN neutral file. Phase II data is ignored. The following table shows the supported Neutral file packet types for import into LUSAS:

Packet	Title	LUSAS Equivalent
25	Title	Used for information purposes only.
26	Time/Date/Version	Used for information purposes only.
31	grid	Point.
32	line	Spline Line defined by 2 Points.
33	patch	Bicubic Surface defined by 4 spline Lines.
34	hyperpatch	Volume.
47	trimmed surface	Bicubic Surface and spline Lines defining the trimmed regions.

Tip. Imported PATRAN data is particularly suited to tidying, since all defined geometry is spline data. See Tidying Imported Lines and Surfaces for more details.

PATRAN Export

Export of LUSAS data to PATRAN was last supported in LUSAS V14.3

Exporting a Solver data file

The **File> Export Solver datafile...** menu item is intended to export Solver datafiles for solution on another computer. It provides the means to select which analyses and loadcases are written to a solver data file for all or only visible elements, without any solving taking place.

The data written to a Solver data file can be controlled according to the following:

- **Analysis** Choose the Analysis type to be tabulated.
- **Loadcases / Influences** Choose **All** loadcases or selected **Specified** loadcases for tabulation
- **Elements** Choose **All** elements, **Selected** elements or only those **Visible** elements for tabulation

Controlling the content of a LUSAS Solver Output File

Advanced export (solution) options are accessed from the **Advanced** button on the Export Solver datafile dialog.

By default no results are written to the LUSAS output file. Results can however be written to the output file for **All** elements and nodes, or for those in the current **Selection** or **Selection Memory**. The following options allow the results written to the output file to be specified.

- ❑ **Element** results such as stress and strain (as controlled by the LUSAS Solver options set from the **File> Model Properties** menu item) can be written to the output file at Node and/or Gauss points and also written to a history file if required.
- ❑ **Node** displacements and reactions can be written to the output file and a history file can also be written if required.
- ❑ **Generate plot file** If selected configures the LUSAS Solver data file to create a plot file (mys). (default)
- ❑ **Generate restart file** If selected configures the LUSAS Solver data file to create a **restart file** (rst).

Notes

- Prior to Version 15.0 this menu item was named **File> LUSAS Datafile**
- During the tabulation process progress will be reported to the Text Window. If problems are encountered warnings and/or error messages will be displayed in the same window. Such warnings and errors can be caused by inconsistencies in the model data which may produce erroneous analysis data files. These errors should be acted on before continuing with an analysis.
- LUSAS is configured to run the majority of analyses without the need to adjust the system parameters. In some circumstances however it may be necessary to adjust one or more of these parameters. System parameters may be modified from within the **File> Model Properties** using the **Defaults** tab. Modified parameters will be tabulated in a SYSTEM chapter at the start of the LUSAS Solver data file.

STEP Import / Export

STandard for the Exchange of Product data (STEP) files are imported according to Part 42 of the Geometric and Topological Representation by using the **File> Import...** menu item. When a file is selected the import process may be controlled by clicking the **Advanced** button and specifying appropriate parameters.

LUSAS model geometry cannot currently be exported to a STEP file.

STL Import / Export

STL files are used by Stereolithography software. They hold information needed to produce 3D models on Stereolithography machines.

STL files are imported using the **File> Import...** menu item. When a file is selected the import process may be controlled from the **Advanced** button by specify the parameters.

LUSAS Model geometry may be exported to STL format from the **File> Export...** menu item.

Notes:

- STL data defines vertices of triangles that define the shape of a surface.

Chapter 4 : Model Geometry

Introduction

There are four geometric feature types in LUSAS to which attributes such as a mesh type, geometric properties, material, supports and loading can be assigned:

- ❑ **Points** define the vertices of the finite element model.
- ❑ **Lines** define the edges of the finite element model. (**Combined Lines** define edges built from a series of continuous lines).
- ❑ **Surfaces** define external faces or internal construction surfaces of a model.
- ❑ **Volumes** define simple solid components of a model.

Features are defined hierarchically , i.e. Points define Lines, Lines define Surfaces, Surfaces define Volumes.

If higher order features are created using techniques which do not involve lower order features, for example, by specifying coordinates, Modeller will automatically generate the lower order features from which to define them. Furthermore, due to feature associativity, when a lower order feature, for example a point, is moved, the higher order features defined by it, for example a line, is also moved.

Features may be deleted from the model provided they are not referenced by a higher order feature. For example, a Line may not be deleted if it is used in a Surface definition.

Geometry creation

Geometry can be created by using a relevant Geometry menu item for each geometric feature type, or be imported from other systems. **See Importing Geometry Data**.

Notes

- Attributes are assigned on a feature basis, therefore the positions of geometric and material discontinuities, supports and loads must be carefully considered when defining the features.

- By default, coordinates are expressed in terms of a global Cartesian axis system. Local coordinates may be used by setting a pre-defined local coordinate active. This is achieved by defining a local coordinate and choosing the **Set Active** command on the context menu. See Local Coordinate Systems.
- Modelling units used to define coordinates and lengths can be input and reported in feet and inches, instead of just feet or just inches. For example, a distance can be input and output as 5'11.5", whereas in previous versions it had to be either 71.5" or 5.92'.

  On dialogs with grid cells expecting dimensional input, the units expected are displayed as a data tip when the cursor is hovered over the input cell.

Visualising Geometry

There are a number of ways to visualise geometry all of which are controlled from the **Geometry** layer properties activated from the Geometry layer context menu. The Geometry layer controls the display of all geometry. If the Geometry layer does not exist or is hidden by another layer the geometry will not be drawn.

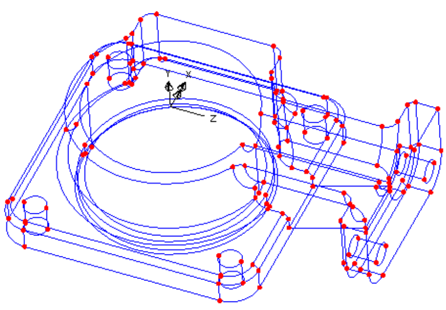
Show Geometry

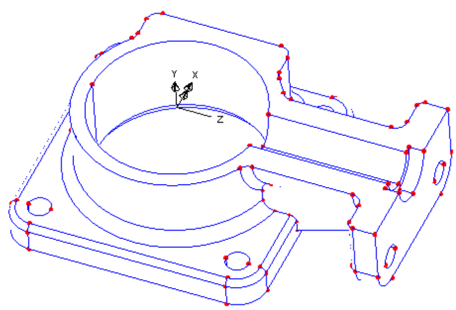
The Geometry layer properties may be set to not show certain geometry types. To aid visualisation Points, Lines, Combined Lines, Surfaces and Volumes can all be independently not shown as required.

Note. Not showing a geometry type is not the same as making all items of that type invisible as the presence of that geometry will affect the visibility of lower order features. (i.e. A Line can not be made invisible if a Surface using that line is not shown (visible but not drawn) because the drawing of Surfaces has been suppressed in the Geometry layer properties).

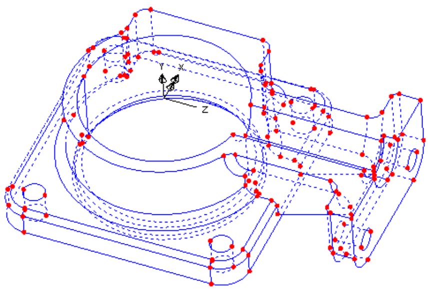
Display Style

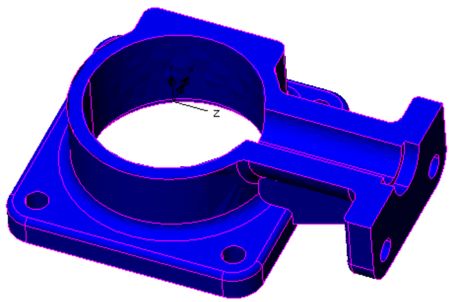
Geometry can be displayed in a number of styles. By default Geometry is viewed in wireframe mode with hidden parts shown but a wide variety of styles can be obtained by mixing the options for wireframe with and without hidden line and solid plots. Some examples follow:





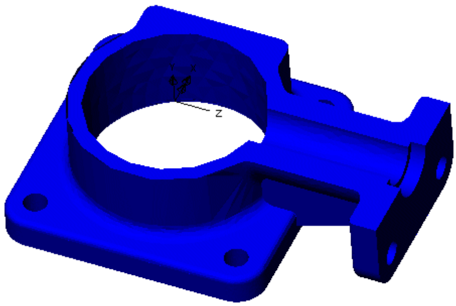
Default wireframe geometry visualisation **Wireframe with hidden Parts removed**





Wireframe with hidden parts draw dashed

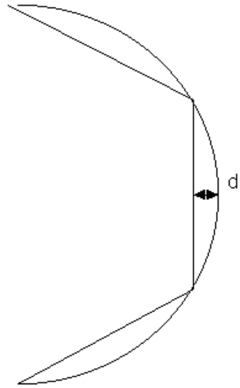
Solid fill with wireframe and hidden parts removed



Solid fill with wireframe (hidden parts removed) and Points hidden

Facet Density

By default lines and surfaces are assigned a facet density which is used in visualisation. Facet density effectively controls how smooth a line or surface will look when drawn to the screen. Straight lines, arcs and splines are all drawn using facets of a particular line length. Surfaces are drawn using facets that are triangular. The default facet density may be changed prior to geometry definition from the **Geometry** tab of the **Model Properties** dialog or after geometry definition by selecting the appropriate Lines and Surfaces and modifying the facet density from the **Geometry> Surface> Facet density** or **Geometry> Line> Facet density** menu items.



The facet density may be specified either as:

- Maximum facet length (in model units).
- Maximum deviation **d** (in model units).

or for Lines as:

- Minimum adjacent angle (in degrees).
- Minimum number of facets for straight line, full arc line and spline line.

or for Surfaces as

- Minimum number of facets for planar surface, surface with seam and other surfaces.

Note that facet density only affects the display of the geometric feature and not the actual geometric accuracy of the model.

Notes

- The display speed is inversely proportion to the number of facets used to define the geometry.
- The facet density can be visualised by selecting the **Facet** option on the **Geometry Layer** properties dialog. The facet density display can be restricted to only selected Lines and Surfaces using the **Facet only if selected** option.

Using Colour For Geometry

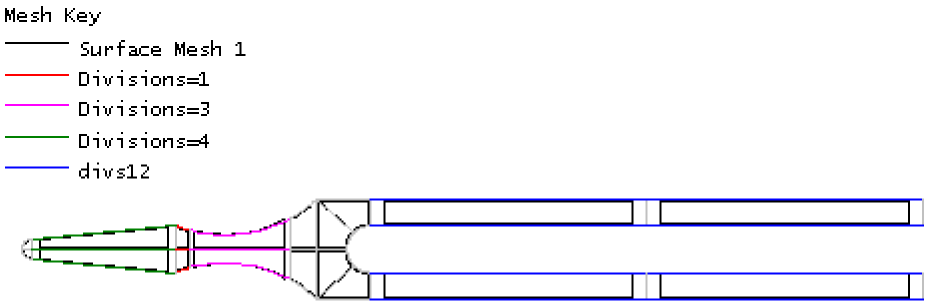
The colour in which geometry is drawn may be specified in many ways.

From the Geometry layer properties specify **Colour by:**

- ❑ **Own colour** An individual feature may be drawn in a pen different from the default geometry type pen. The pen is specified on the properties dialog. Select the single feature and right click then choose Properties to display the feature properties. Until a pen is set for an individual feature, that feature will be drawn using the default pen.
- ❑ **Normals** Surfaces are coloured according to whether they are orientated showing the top or bottom of the surface.
- ❑ **Assignment** Features are coloured according to which attribute is assigned to them. Features with no attribute assigned are drawn in grey. The picture below shows an example of this
- ❑ **Group** Features are coloured according to which group they are in. Features not in a group are drawn in grey
- ❑ **Type** Each geometry type has a default pen, associated with it. This option causes all geometry to be drawn in that pen. The colours may be set from the Model Properties.

By default the settings are red for Points, magenta for Lines, orange for Combined lines, green for Surfaces, blue for Volumes.

- **Line / Surface Connectivity** Features are drawn in colours according to the number of higher order features connected to them and areas of the model that have not been merged together correctly after import are highlighted by being drawn in a different colour. One example of use is for checking models created from the import of 3D CAD data where the use of this option would enable any surfaces that were not correctly forming volumes to be seen. Use of the **merge** facility would correct any unmerged and isolated features.



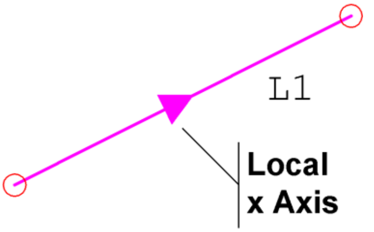
Orientation Axes

Orientation axes may be viewed as a local axis set for Lines, Surfaces and Volumes . The local x, y and z axes are shown, with a double arrowhead on the x axis and a single arrowhead on the y axis and no arrow head on the z axis.

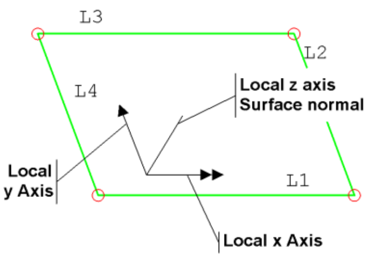
When features are meshed, the orientation of the feature determines the orientation and spacing of the elements. Therefore the orientation of Lines, Surfaces and Volumes can be changed by reversing or cycling the features. See **Changing Geometry Orientation** for more details.

In all of the following illustrations the local axes are orthogonal.

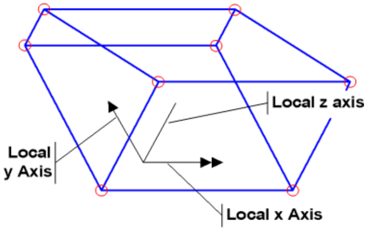
- **Line** directions can be drawn to indicate the local x direction of Line or axes can be displayed.



- **Surface** axes or surface normals can be displayed. Axes are positioned adjacent to the first Line in the Surface definition. In the example shown the axes are orthogonal but viewed from an angle to show the z axis orientation.



- **Volume** axes can be displayed. The origin or the axis is closest to the first point in the first Surface of the Volume definition.



Tip: To aid axes visualisation on larger models choose the **Orientations only if selected** option. This will display axes only on selected features.

Labels

Labels can be added to the view of a model from the **View> Insert Layer> Labels** menu item, or from the window context menu. The label options are controlled from the labels property dialog.

Labels may be added to geometry features as follows:

- Name
- Position
- Mesh
- Geometry
- Material
- Supports
- Loading
- Transformed freedoms
- Composite
- Slideline
- Constraints
- Thermal surfaces
- Retained freedoms
- Damping
- Activate
- Deactivate
- Equivalence
- Search area
- Influence
- Age
- Tendon

Node, element and Gauss point labels may also be displayed.

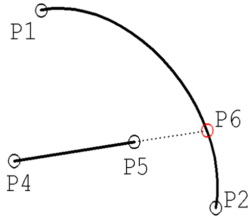
Notes

- Line labels for standard Lines are drawn at 3/4 distance from the start of the line. This can be a useful indication of the line orientation.
- If a Line is used as part of a Combined Line definition, the Line label is located at 8/10 distance and the corresponding Combined Line label is located at 6/10 distance along the Line segment. This is to avoid the labels overwriting each other.
- For complex models labels may be displayed only on selected features by choosing the **Label selected items only** option on the label properties dialog.

Points

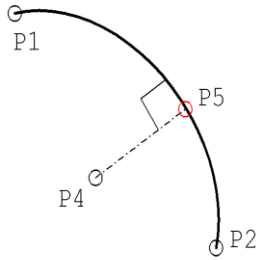
Points define the vertices of a model and are used in the definition of lines. Point geometry is defined from the **Geometry**> **Point** menu item, or by the use of the New Point toolbar icons. Line types available are:.

- ❑ **Coordinates** Defines a Point by entering the X, Y and Z coordinates (Z is optional). If a non-Cartesian **local coordinate system** is in use the coordinates are specified in the coordinate system of that local coordinate set. The dialog box labels will be updated to reflect the required coordinate input.
- ❑ **Cursor** Allows definition of a series of Points on the screen with the cursor. The Points can snap to a grid in the XY, YZ or XZ plane. The out of plane ordinate can be specified as non-zero if desired. This facility is useful for positioning Points on Lines or Surfaces which will be used for splitting that feature later.
- ❑ **From Mesh** Defines a Point at the position of every node of the selected mesh. **See Geometry From Mesh**. This is useful for defining a Point feature to which loads or supports can be subsequently assigned. The Point must be **equivalenced** with the underlying meshed feature in order for the Point's assigned attributes to be transferred to the underlying nodes. Subsequent re-meshing of the structure with different mesh spacing characteristics may result in movement of the underlying nodal positions.



- ❑ **By Intersection** Defines a Point or a number of Points at the intersections of two or more selected Lines. When the Lines selected do not physically intersect Points may be created at the nearest intersections. These nearest intersections are controlled by the following options.
  - ❑ **All point pairs** creates Points at all possible intersections.
  - ❑ **Nearest point pair only** creates a Point on each line where the projection of the Lines is at its nearest.
  - ❑ **Nearest pair to reference position** defines Points on each line at the intersection nearest to the defined reference Point.

- ❑ **Limit distance between points** only creates Points at intersections that are within a specified distance
- ❑ **Allow extended lines** will create Points at the intersection of the extension of the selected Lines.



- ❑ **By Projection** Defines a Point at the (perpendicular) projection of a selected Point onto a selected Line or Surface.
- ❑ **By Extension** Defines a point at the extension of a selected Line. The extension may be defined as a parametric or actual length.
- ❑ **Make Planar** moves the selected points onto a plane defined either as an offset from an orthogonal plane, or onto a plane defined by 3 coordinates, or as a best fit to the selected points. A local coordinate may be used to define the orthogonal axes if required.

Case Study. Editing Point Properties

By selecting a single Point, then right-clicking to display the context menu, properties relating to that Point can be displayed. From an individual Point’s properties the coordinates can be altered, and attribute assignments may be manipulated.

Lines

Lines connect point features and are used in the definition of surfaces. Line geometry is defined from the **Geometry**> **Line** menu item, or by the use of the New Line toolbar icons. Line types available are:

- ❑ **Line** defined by two Points.
- ❑ **Arc/Circle** defined by two Points and a Line map (a means of selecting the meaning of a third definition point)
- ❑ **Spline** defined by two or more Points.
- ❑ Lines may also be created by **Splitting** lines, and **Intersecting** with other lines

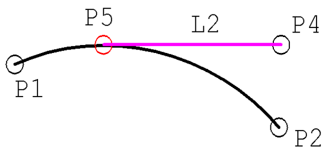
Additionally, these other lines may, during the modelling process, be created as a result of editing the geometry:

- ❑ **Elliptical Arc/Ellipse**
- ❑ **Composition Line** defined by a Line and Surface map.
- ❑ **Intersection** defined by two Surface maps.
- ❑ **Isoparametric**

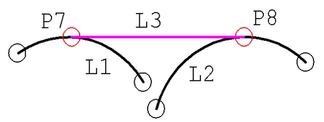
Line geometry is defined from the **Geometry**> **Line** menu item.

General Line Definition

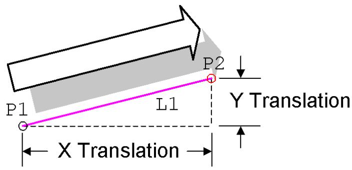
- **Coordinates** Defines a Line by entering the X, Y and Z coordinates. Entering more than two coordinates will define linked Lines. If a non-Cartesian local coordinate system is in use the coordinates are specified in the coordinate system of that local coordinate.
- **Cursor** Allows definition of a series of straight Lines on the screen with the cursor. The Lines can snap to a grid in the XY, YZ or XZ plane. The out of plane ordinate can be specified as non-zero if desired.
- **Points**Defines a Line from selected Points. A dialog is displayed to specify the Line type as either Straight Line(s), Arc, or Spline.
- **Tangent > Point to Line** Defines a Line between a selected Point and the tangent to a selected Arc. An error will occur if no tangent is possible. In this example Line 2 is created by specifying Line 1 and Point 4. Point 5 is automatically created.



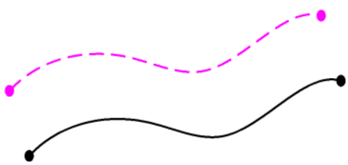
- **Tangent > Line between Arcs** Defines a Line which is tangent to two coplanar Arcs. The new Line can be defined as an inside or outside tangent. Options are available to split the Arcs at the new Points, and then delete the original Arcs.



- **By Sweeping** Defines a Line by sweeping a Point through a transformation (translation, rotation, mirror or scale). In this example Line 1 is created by sweeping Point 1 through a *translation* in X and Y.

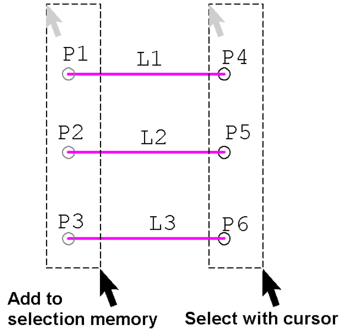


- **By Offsetting** Defines a Line or series of Lines offset from a selected Line or series of Lines. If multiple Lines are selected a new set of Lines can be offset in one operation. The direction of the generated lines can be controlled by selecting a point in the direction of the new lines required, prior to selecting the lines to be used for offsetting.. If multiple Lines are selected the outside fillets may be created with arcs or straight Lines.

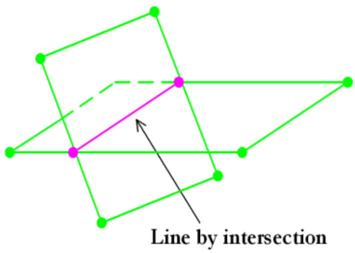


- **By Joining** Defines a number of Lines by joining two sets of Points. Points put in selection memory define the start of each Line, and Points selected in a subsequent 'normal' selection define the end of each Line. The Points must pair up equally. Lines are joined according to the order in which the Points were selected, (or Point number when boxing a selection), i.e. first point in selection memory joins to first point in the selection, etc. In this example Lines 1 to 3 were defined, by first adding Points 1 to 3 to selection memory, then selecting Points 4 to 6, and then using this command.

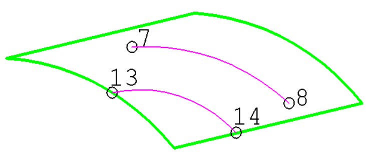
  Note also that a fan of lines from one point to many others can be created by setting one point only in selection memory prior to selecting the other set of points in a 'normal' selection. This is useful for creating lines to represent the cables radiating from the top of a tower to the deck on a cable stayed bridge for example.



- **By Intersection** Lines may be defined by intersecting two or more Surfaces. Intersects all Surfaces within a single selection with all other Surfaces within that selection. If no intersection is found a warning will be issued.



- **By Manifolding** (via projection) Existing Lines may be projected or laid onto an existing Surface. A Surface to be projected onto is selected, followed by the Line to be projected and the **Geometry> Line> By Manifolding** menu item is used to create the new manifolded Line. The new Line is created in the plane of the surface.
- **By Manifolding** (via Point creation) Lines can also be created directly onto a Surface by creating points lying on a surface (use the Geometry> Point> Cursor menu item to snap points, as for instance to a curved surface as in the example shown) prior to using the **Geometry> Line> By Manifolding** menu item to create the manifolded line. Points created prior to choosing the command



can be, but do not have to be, on the boundary of the Surface. In this example Points 7 and 8 lie within the Surface boundary, while Points 13 and 14 lie on the boundary.

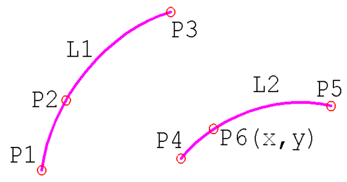
- **From Mesh** Defines a Line based upon points at the position of every node of the selected mesh elements. **See Geometry From Mesh**.

Arc and Circle Definition

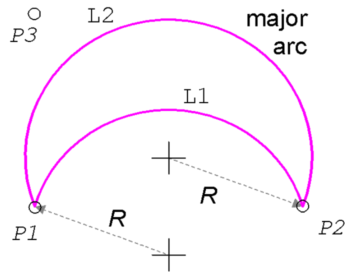
Arcs and circles are defined from the **Geometry> Line> Arc/Circle** menu item.

- ❑ **From Coords/Points** Coordinates can be entered manually or taken from selected Points. The coordinates define the arc in one of three ways:

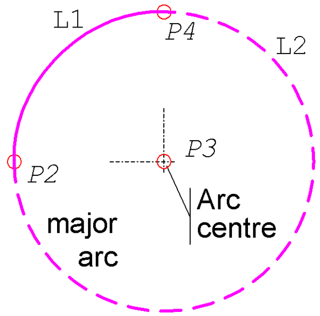
  - **Start Point, Bulge Point and End Point** Defines an Arc or Circle which passes through three coordinate points. In this example Line 1 is defined by selecting Points 1, 2, 3 and specifying Point 2 as the 'bulge' point. Line 2 is defined by selecting P4 and P5, then entering the coordinates to define a 'bulge' Point, P6 (which is not actually created)



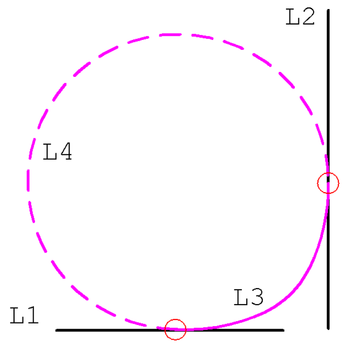
  - **Start Point, Directional Point, End Point, and Radius** Defines an Arc or Circle between two coordinate points, to a specified radius. A third coordinate point is required to indicate the direction in which the arc or circle bulges. In this example arcs are created from Point 1 to Point 2 with a radius of *R*, using Direction Point 3. Choosing Minor Arc creates Line 1, while choosing Major Arc creates Line 2. A Major Arc subtends an angle greater than 180 degrees at the arc centre, while a Minor Arc subtends an angle less than 180 degrees.



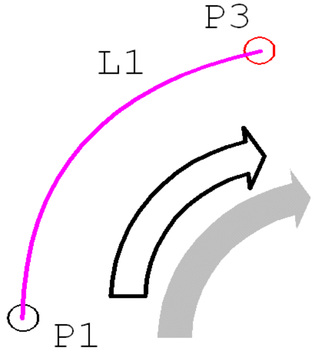
- **Start Point, Centre Point and End Point** Defines an Arc or Circle between two coordinates, with a third coordinate defining the centre of the arc or circle. The centre must be equidistant from the start and end points. In this example arcs are created by selecting Points 2, 3, 4 with Point 3 as the arc centre. Choosing Minor Arc defines Line 1. Choosing Major Arc defines Line 2 (dotted line for clarity). Note that the centre point is not linked to the line, so the geometry will not update if it is moved.



- ❑ **Tangent to Lines** Inserts an Arc or Circle with a specified radius, tangent to two selected Lines. The selected lines must be straight Lines or Arcs. In this example Line 3 is created by selecting Lines 1 and 2, then specifying a radius and Minor Arc. Alternatively, Line 4 is created by selecting Lines 1 and 2, then specifying a different radius and Major Arc.



- ❑ **By Sweeping Points** Defines Arcs by sweeping selected Points through a specified rotation. In this example Point 1 is swept into Line 1 using a rotational transformation and choosing the Minor Arc option.



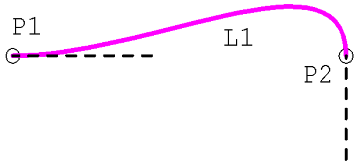
Spline Definition

Spline Lines are defined from the **Geometry**> **Line**> **Spline** menu item.

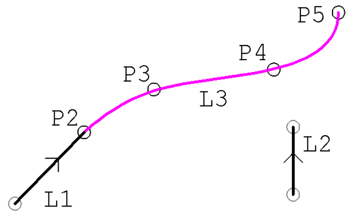
Note that the inetrmediate points used to create a spline are not linked to the spline, so the geometry will not update if these points are moved.

- ❑ **By Points** Defines a spline from three or more selected Points.

- **Points and End Tangents** Defines a Spline passing through two or more selected Points. The end directions are defined by entering end tangent vectors. In this example Line 1 is defined by specifying Points 1;2 and inputting start and end tangents by vectors (1, 0, 0) and (0, -1, 0) respectively. Both the direction and length of the end tangents control the spline shape. In this example, a different shape of spline would be defined, passing through the same Points, if the end tangents were changed to (3, 0, 0) and (0, -0.5, 0) respectively.



- **Tangent to Lines** Defines a Spline passing through two or more selected Points. The Spline end vectors are taken from the directions of two selected Lines. The tangent Lines do not have to connect with the Spline. In this example Line 3 is defined by selecting Points 2, 3, 4, 5 to define the path, and Lines 1 and 2 as end tangents.



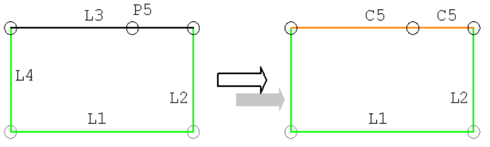
Splitting Lines

Lines may be split at a Point, or split into a number of equal or unequal divisions to form new Lines. The splitting Lines dialogs contain options to automatically split and delete the original Lines, and to replace the split Lines with Combined Lines. Note that when splitting Lines the original Line can only be deleted if it did not define any Surfaces, or if the 'Use in dependent Surfaces' option is selected. As a further option a combined line may be created. This is useful if a regular mesh is required.

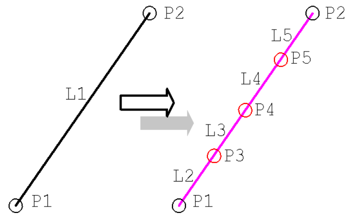
Any attributes assigned to a feature that is split will be automatically assigned to the new features created.

Line splitting commands are accessed from the **Geometry> Line> By Splitting** menu item.

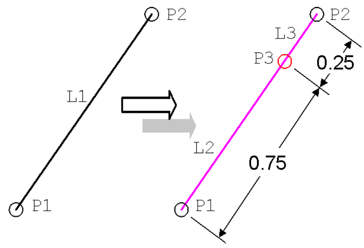
- **At a Point** Splits an existing Line into two new Lines at a selected Point on the Line, Spline or Arc. A splitting tolerance can be set to increase the allowable distance between a Point and the Line being split. In the example here Line 3 is split at Point 5 into 2 new component Lines defining Combined Line 5. Line 3 is also replaced in the definition of Surface 1.



- **In Equal Divisions** Splits an existing Line into a specified number of equal divisions. A new Line is defined at each division. In the example shown Line 1 is split into 4 new Lines of equal length, and the original Line is deleted.



- **Parametric or Actual Distance** Splits an existing Line into a number of divisions based on specification of parametric distance values along the Line. The direction of the existing line is used to calculate the splitting positions. A new line is created at each division. In this example Line 1 is split at a parametric distance of 0.75. Line 2 is created at 3/4 of the original length and Line 3 at 1/4. Specification of a parametric divisions list as 0.1;0.5;0.75 will split a line at 1/10, 1/2 and 3/4 distance into 4 new lines.

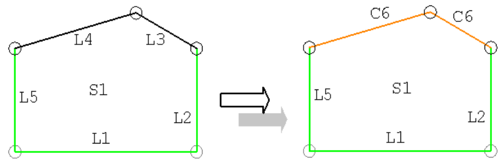


Combined Lines

A Combined Line is a Line which is composed of several continuous individual Lines. Combined Lines may be used in exactly the same was as other Lines to allow Surfaces, to be meshed using regular mesh patterns. This is especially useful for meshing surfaces defined by more than four Lines.

Combined Lines are defined from the **Geometry**> **Line**> **Combined Line** menu item.

- **Lines** Defines a Combined Line from two or more selected Lines that are continuous but do not form a closed loop. Any number and type of existing Lines can be used to define a Combined Line. In the example here Lines 3 and 4 are used to define Combined Line 6. Lines 3 and 4 are replaced in the definition of Surface 1.

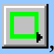
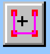


Notes

- Automatic numbering uses the next highest available number for Lines or Combined Lines.
- Mesh attributes may not be assigned to Combined Lines, but must be assigned to the Lines defining the Combined Line.
- Using Combined Lines in the definition of Surfaces provides an additional means of producing transition meshes.

Case Study. Using Combined Lines

Combined lines may be used in a surface definition in order to use a regular mesh.

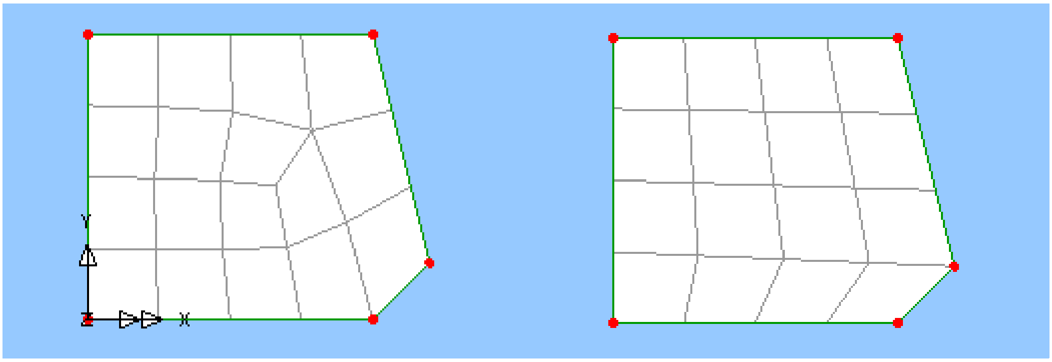
1. Define a Surface using the New Surface button , and enter the coordinates (0,0,0), (100,0,0), (120,20,0), (100,100,0) and (0,100,0).
2. Define a Line mesh attribute (using **Attributes> Mesh> Line**) with of element type None with mesh divisions of 1 and assign it to the shorter Line on the left hand side of the surface.
3. Define a Line mesh attribute (using **Attributes> Mesh> Line**) with of element type None with mesh divisions of 3 and assign it to the longer Line on the left hand side of the surface.
4. Define a Surface mesh attribute using (using **Attributes> Mesh> Surface**) with Plane Stress, Quadrilateral, Linear elements and assign it to the surface. This will automatically select an irregular mesh for a five-sided Surface as shown below.
5. Now define a Combined Line by selecting the Lines on the left hand side of the model and press the  button from the Line menu item.
6. Since the Surface definition has been altered the Surface will be remeshed. A regular mesh will now be adopted because the line divisions on the Combined Line match those on the opposite Line.

Irregular Mesh

A regular Surface mesh using Plane Stress, Quadrilateral, Linear elements is assigned to the Surface with the resulting mesh shown. (An irregular mesh is used as surface does not have 3 or 4 sides).

Regular Mesh Using Combined Lines

The lines on the right hand side are used to define a Combined Line. A remesh occurs because the surface has been redefined and a regular mesh is generated because the surface is now defined with 3 Lines and 1 Combined Line.



Surfaces

Surfaces define the faces of a model and are used in the definition of volumes. Surface definition commands are found under the **Geometry> Surface** menu item. Surface types are:

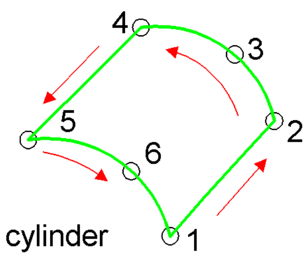
- **Regular** defined by 3 or 4 Lines.
- **Irregular** defined by 5 or more Lines.

Note. For meshing purposes an irregular surface can be considered to be a regular surface using Combined Lines.

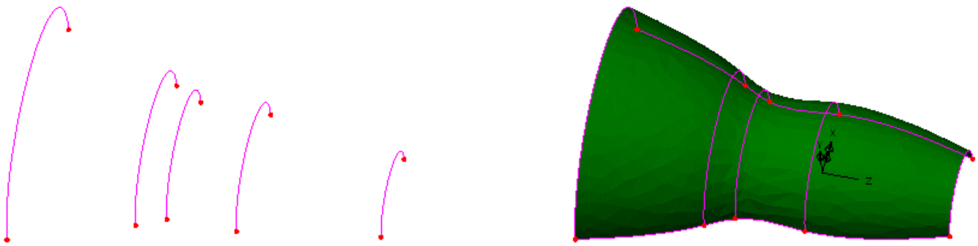
Surface Definition Commands

The following commands are for defining Surfaces directly:

- **By Coordinates** Defines a Surface by entering a list of X, Y and Z coordinates (Z is optional). If a non-Cartesian local coordinate system is in use the coordinates are specified in the coordinate system of that local coordinate. Coordinates can be entered using global (default) or local coordinate systems.
- **By Cursor** Allows definition of a series of flat rectangular Surfaces on the screen with the cursor. The Surfaces can snap to a grid in the XY, YZ or XZ plane. The out of plane ordinate can be specified as non-zero if desired.
- **By Points** Defines a Surface from three or more selected Points. A Surface can be defined from any number of Points.
- **By Lines** Defines a Surface from the selected Lines. When defining Surfaces in this way, any number of Lines may be specified in any order and a Surface will be formed from the greatest number of lines that form a closed loop. If a number of disconnected Lines are selected a **Lofted Surface** will be created. A lofted Surface is defined as Surface with a smooth transition between 2 or more Lines.
- **From Mesh** Defines a Surface based upon lines and points at the position of every node of the selected mesh. **See Geometry From Mesh**.



Surfaces generated from selected Lines



Lofted Surface generated from selected Lines

Note: The direction in which the Surface is defined is used to define the **Surface orientation**, The surface orientation is used at a later stage to define element normals and local loading directions.

Coalescing Surfaces

Two or more surfaces can be reduced to one surface defined by lines or combined lines if the surfaces share a common line.

Creating Holes in Surfaces

Geometry> Surface> Create Holes

- **Create** Defines a hole or number or holes in a Surface. Closed loop(s) of Lines or the Surface(s) representing the hole(s) and the Surface to be holed should be selected prior to choosing this menu option.
- **Move** Moves or modifies selected hole(s) within a Surface. To move a hole select the Lines and/or Points forming the perimeter of the hole. If only some Lines or Points are selected the hole will be stretched by only moving the selected features. **Note.** Moving a hole actually deletes the original hole and recreates a new hole. This means the feature numbers of the Points and Lines will not be maintained.
- **Copy** Creates multiple holes within a Surface.

- **Delete** Deletes the selected hole(s). The hole boundary lines may be deleted, retained as Lines or used to create new Surfaces.
- **Delete All** Deletes all holes from a selected Surface. The hole boundary lines may be deleted, retained as Lines or used to create new Surfaces.

Case Study. Creating a Surface with Holes

It is often convenient to create a Surface with embedded holes rather than create a number of Surfaces around a hole. To create a single Surface with a number of holes carry out the following:

1. Create the outer Surface and a defined a separate 'inner' Surface for each hole.
2. Select all Surfaces.
3. Use the **Geometry> Surface> Holes> Create** menu item to create a single Surface with holes. Ensure that the **Delete geometry defining holes** option is chosen to remove the original 'inner' Surfaces defining the holes. The lower-order points and lines of those surfaces will be re-created because they are required define the inner boundaries of the enclosing surface.

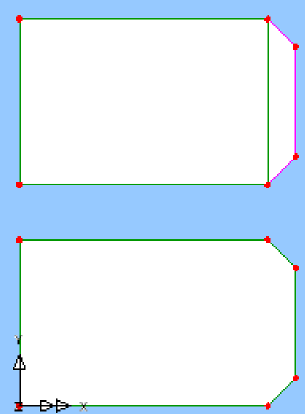
Redefine Perimeter

A Surface perimeter may be modified by selecting the Surface and the new boundary Lines. The new Surface perimeter will be created from a closed loop of Lines formed from the old Surface perimeter and the new boundary Lines. The perimeter of the Surface will be defined from the closed loop of Lines with the maximum number of segments. If two possible loops of Lines have the same number of segments a additional Line from the existing Surface boundary should be selected to resolve the ambiguity.

Case Study. Redefine Surface Perimeter

To redefine the perimeter of a Surface use the following steps.

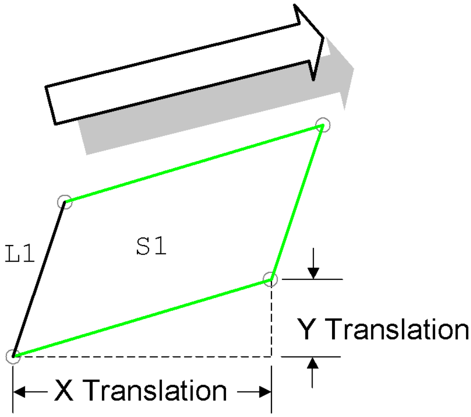
1. Create a Rectangular Surface.
2. Define the new perimeter with Lines which start and finish at existing Points on the Surface.
3. Select the Surface and the new perimeter Lines.
4. Use the **Geometry> Surface> Redefine Perimeter>Redefine** menu item to define the new Surface perimeter.



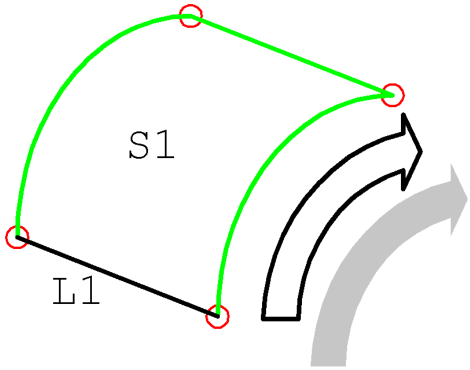
Sweeping Surfaces

- **By Sweeping** Defines a Surface by sweeping a selected Line through a transformation (translation, rotation, mirror or scale).

Line 1 is swept using an X and Y translation to create Surface 1:

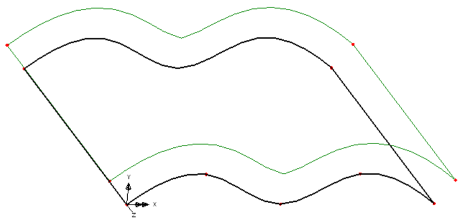


A cylindrical Surface is defined by sweeping Line 1 through a rotation:



Offsetting

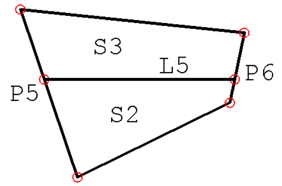
- **By Offsetting** Defines a Surface at a specified distance normal to a selected Surface. The positive offset direction is defined by the Surface normal unless an additional Point is selected.



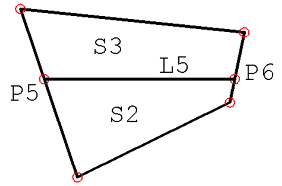
Splitting Surfaces

Surfaces may be defined by splitting an existing surface in a number of ways using the menu items under **Geometry> Surface> Splitting**. Surface splitting commands contain an option to automatically delete the old Surfaces and Lines which have been split. Attributes assigned to the split feature will automatically be assigned to features of the same type created during the split process. The following splitting methods are supported.

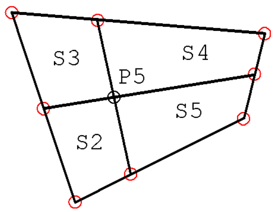
- **By Line** Splits a selected Surface at an existing Line. The end Points of the Line must lie on different boundary Lines of the Surface. It is advisable to split curved Surfaces using Lines that are manifolded over the Surface. In the example shown here, the original Surface is split at Line 5. Points 5 and 6, defining Line 5, must lie on the original Surface boundary Lines.



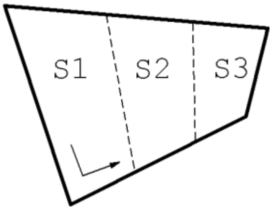
- **By Points** Splits a selected Surface at two boundary positions indicated by Points. A new Line will be manifolded onto the existing Surface and the Surface will be split at this Line. In the example shown, Surfaces 2 and 3 are created in the parametric space of the original Surface using the Points on the boundary. The new Line is manifolded onto the original Surface.



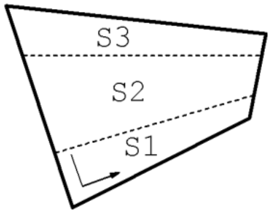
- **At a Point** Splits a selected Surface at a single Point defined on the surface inside the boundary. Four new Surfaces are created by manifolding straight Lines or arcs onto the existing Surface using the relative position of the splitting Point. In this example, the original Surface is split at Point 5. The resulting Surfaces (2-5), use parametric space to calculate boundary Point positions.



- **In Equal Divisions** Splits a selected Surface into separate Surfaces at a specified number of equal divisions. The direction of the split is expressed using local Surface axes. In the example shown here, the original Surface is split in its local x direction into equal divisions forming 3 new Surfaces.



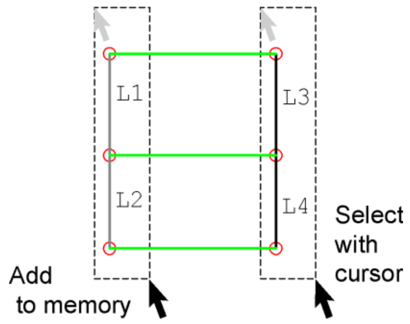
- **At Parametric Distances** Splits a selected Surface into separate Surfaces at specified parametric divisions. For example, specifying the parametric distances as 0.25 and 0.75 will split a Surface at the 1/4 and 3/4 position in parametric space in the specified local direction. In this example, the original Surface is split in its local y direction at parametric divisions 0.25;0.75.



Joining

- ❑ **By Joining** Defines a number of Surfaces by joining two sets of selected Points or Lines.

  The first set of Lines should be added to selection memory, the second set should be selected. The Lines should pair up equally. Surfaces are joined according to the order in which the Lines were selected, i.e. first Line in selection memory joins to first Line in selection, etc.



  In the example shown Surfaces 1 and 2 are defined by first selecting Lines 1 and 2, then adding them to selection memory, then selecting Lines 3 and 4 and using the Surface by joining command.

Other surface joining examples

- A single line can be put in selection memory and a set of lines selected in a normal selection and surfaces created .
- A single point in selection memory and a set of lines in normal selection can be used to created a pyramid shape of triangular surfaces.

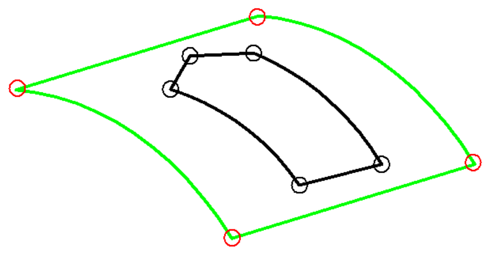
Intersection

- ❑ **By Intersection** Defines a Surface at the intersection of two selected Volumes.

Manifolding

Manifolding is the process of creating geometry which lies on the surface map of an existing Surface.

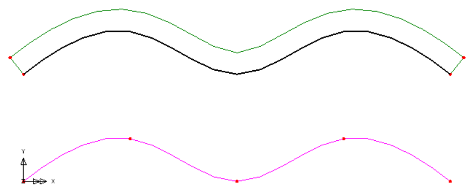
- ❑ **By Manifolding** Existing Surfaces may be projected or laid onto an existing Surface. A Surface to be projected onto is placed into Selection Memory and the Surface to be projected is then selected prior to choosing the **Geometry**> **Surface** > **By Manifolding** menu item. The new Surface is created normal to the selected item (in selection memory) and will lie on the map of the underlying Surface.
- ❑ **Manifold by Lines** Defines a Surface positioned on an existing underlying Surface with its boundary specified by edges or vertices. In this example a Surface is created by Joining any combination of the Points and Lines shown. Any new Lines created will automatically be manifolded onto the



underlying Surface.

Extrusion

- **By Extrusion** Defines Surface by extruding selected Lines a specified distance. The positive direction may be defined by an additional Point if selected. For Arcs the default direction is assumed to lie in the plane of the Arc.



Volumes

About Volume Features

Volumes define the solid geometry of the model and come in two forms.

- **Solid Volumes** are defined by a number of connected Surfaces. These are recognised by the geometry engine which allows Boolean operations to be performed. Solid Volumes are the default form of Volume and are created when geometry is defined in Modeller.
- **Hollow Volumes** are defined by a number of disconnected Surfaces. These are not recognised by the geometry engine. Hollow Volumes are usually only of use when the geometry has been imported from CAD.

For meshing purposes volumes may be split into the following categories.

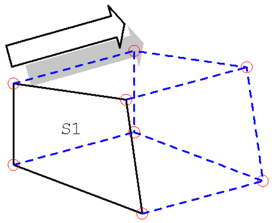
- **Regular** volumes may be defined as **Tetrahedral** (4 sided volume with all faces defined by triangular surfaces), **Pentahedral** (5 sided prism with top and bottom faces defined by triangular surfaces and side faces defined by quadrilateral surfaces), **Hexahedral** (6 sided cuboid with all faces defined by quadrilateral surfaces).
- **Swept Irregular** are defined by 2 identical irregular Surfaces joined at equivalent positions on the boundaries by quadrilateral Surfaces. The Lines joining the two irregular Surfaces must be either straight Lines or arcs with a common centre.
- **Irregular** are defined by any number and type of surface but can only be meshed with tetrahedral elements.

Defining Volumes

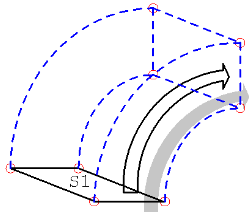
The following commands are for defining Volumes directly:

- By using the **Shape Wizard** to create regular volumes.
- By selecting **Surfaces** to define a Volume from four or more selected connected Surfaces. The Surfaces may be entered in any order. When defining Volumes in this way any number of Surfaces may be selected to define a Volume. Duplicate and unconnected Surfaces will be filtered out.

- ❑ **By Sweeping** to define a Volume by sweeping a selected Surface through a transformation (translation, rotation, mirror or scale).

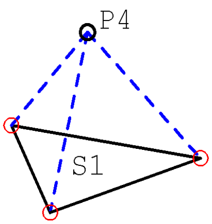


A Surface is swept through a *translation* to create a Hexahedral:

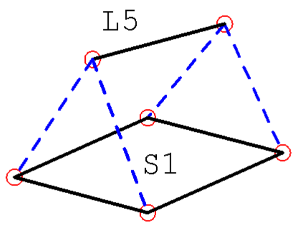


A Surface is swept through a *rotation* to create a Volume:

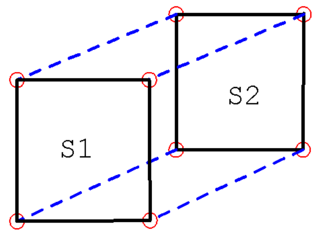
- ❑ **By Splitting** Splits a selected Volume by a selected Surface.
- ❑ **By Joining** Joins two selected features to form a Volume, either a Point to a Surface, or a Line to a Surface, or two Surfaces. The features are joined by straight lines.



Surface 1 is joined to Point 4 to form a pentahedral.



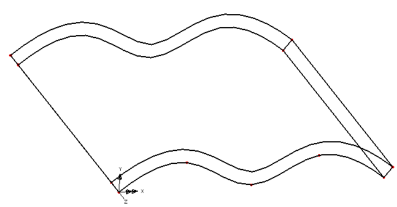
Surface 1 joined to Line 5.



Surface 1 is joined to Surface 2 to form a hexahedral. Two triangular Surfaces joined this way would form a pentahedral.

Groups of Volumes may be defined by joining two sets of selected Surfaces. The first set of Surfaces should be added to selection memory and the second set of surfaces should be selected. The Surfaces will pair up equally. i.e. Volumes will be joined according to the order in which the Surfaces were selected. The first set of Surfaces in selection memory joins to the first set of Surfaces in selection etc.

- ❑ **By Extrusion** Enables a Volume to be defined by extruding a specified distance normal to a selected Surface. The positive direction is defined by the Surface normal unless an additional Point is selected. Extrusion can be towards or away from the specified reference Point.



Coalescing Volumes

The internal Surfaces from a Volume model may be removed by coalescing Volumes. This will result in a model with fewer Volumes which may be meshed using Tetrahedral elements.

From Mesh Defines a Volume based upon the position of every node of the selected mesh. **See Geometry From Mesh**.

Case Study. Coalesce Volumes

Sometimes a results file exists without a corresponding model file. The coalesce Volumes feature allows the results mesh to be converted to a Volume model so it can be modified and used in a subsequent analysis.

1. Open results file and use the **File> Save As** menu item to save it as .mdl file.
2. Use **Control-A** to select the mesh and convert it to Volumes using the **Geometry> Volume> From Mesh** menu item and select the **Coalesce Volumes** option.
3. Create a loadcase.

Delete Holes

When geometry is imported from CAD it may have small holes defined which are of no significance in the analysis. These holes may be removed using the **Geometry> Volume> Delete Holes** menu item.

Delete Voids

Voids are cavities in a Volume which do not penetrate the defining surfaces. Voids are removed using the **Geometry> Volume> Delete Voids** menu item.

Orientating Volume Axes

In some analysis types the Volume axes is used to define the material direction. The Volume axis may be orientated by the following methods:

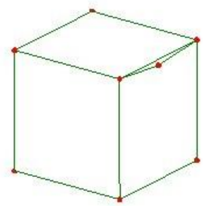
- ❑ Axis to Surface
- ❑ Cycle Axes
- ❑ Cycle Relative

See **Changing Geometry / Element Orientation** for details.

Hollow Volumes

Hollow volumes, just like normal solid volumes, are defined by a set of surfaces. However, unlike normal solid volumes, the defining lines and points of those surfaces do not need to be perfectly merged together. Hollow volumes are mainly used when geometry has been imported from CAD systems, when such merging may be difficult or impossible. Once defined, hollow volumes behave in almost exactly the same way as normal solid volumes. For example, they may be meshed and have attributes assigned and deassigned in the usual way. The main difference is that in a geometric Boolean operation a hollow volume will behave like a set of surfaces, hence the name.

Hollow volumes may be open or closed. To understand the difference consider just one defining edge of the volume between two defining surfaces. If that edge consists of exactly one line, or exactly two lines of very similar length and shape, then those lines can be considered a matching pair. If all the defining edges of the volume can be matched in this way, the volume is said to be closed. However, if one or more edges of the volume cannot be matched in this way then the volume is said to be open.



For the example shown most of the lines form matching pairs. However, one of the defining surfaces has an edge that is defined by two lines, with a point in the middle. This pair of lines cannot be matched to the corresponding single line on the adjacent surface, and so this volume is open.

The distinction between open and closed hollow volumes is only important when it comes to meshing. A closed hollow volume can be meshed in exactly the same way as a normal solid volume. However, a regular mesh cannot be assigned to an open hollow volume, and so these must be meshed using an irregular mesh.

The user interface to enable the creation of hollow volumes may be invoked from the Advanced Dialog of the Geometry Properties dialog. This option is automatically invoked when geometry is imported from CAD. When this option is chosen the volume definition tools will try to automatically create a solid volume as normal. However, if this process fails, a further attempt will be made to create a closed hollow volume from the selected surfaces using the defined closure tolerance.

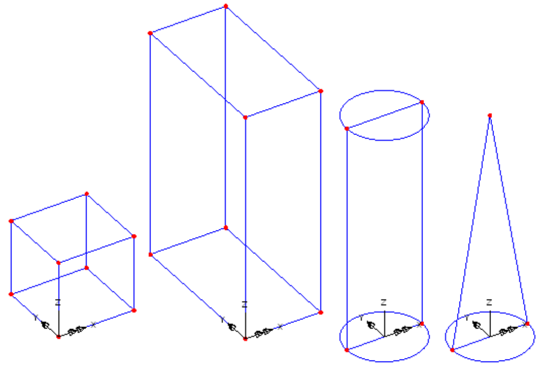
Open hollow Volumes can only be created via the **Geometry> Volume> Hollow Volume> Create** menu item. Surfaces may be added or removed from an open hollow volume definition using the **Geometry> Volume> Hollow Volume> Add** and **Geometry> Volume> Hollow Volume> Remove** menu items.

Once a hollow volume has been defined its status (open or closed) can be determined from its properties dialog. In some cases an open hollow volume may be changed to a closed hollow volume by simply increasing the closure tolerance on the volume properties dialog. When meshing a closed hollow volume, any nodes closer together than the volume's node merge tolerance (defined on the volume properties dialog) are merged. For closed hollow volumes this defaults to the volume closure tolerance.

Shape Wizard

The shape wizard defines analytical shapes which may orientated with a local coordinate system and positioned by a user defined origin. If a point is selected the selected shape's origin will default to the coordinates of the selected point. Shapes may be defined using Lines, Surfaces or Volumes.

The following shapes are supported; cube, cuboid, cylinder, and cone. The origin of each shape is indicated by the axes.



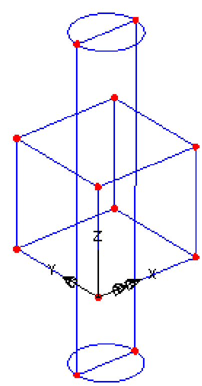
Notes.

- Specifying a negative length or height results in that dimension being defined along the negative axis direction.
- For the cylindrical shape there is an option to create the cylinder with a seam which effectively creates a cylinder using only one surface rather than the default two surfaces.

Boolean Geometry Construction

Boolean operations allow complex geometry to be defined by combining, subtracting or intersecting existing Surfaces or Volumes.

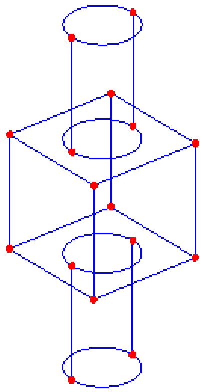
- **Union (with simplify internal geometry)** enables a Surface or Volume to be defined by union of any number of selected Surfaces or Volumes.
- **Union (without simplify internal geometry)** enables a number of Surfaces or Volumes to be defined by union of any number of selected Surfaces or Volumes.
- **Subtraction** enables a Surface or Volume to be defined by subtracting one Surface or Volume from another Surface or Volume.
- **Intersection** enables a Surface or Volume to be defined as the intersection of two selected Surfaces or Volumes.
- **Slice** Enables a selected Volume to be sliced by a plane and the resulting geometry to be deleted either side of the slice if required. The slice plane may be defined in any global or local plane which may be visualised prior to the slice



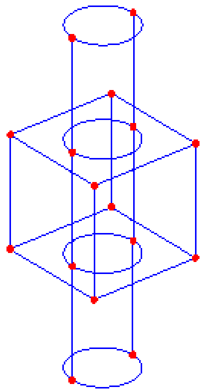
Separate (unconnected) cube and cylinder =

operation. A Surface of any shape may be used to slice a Volume by Subtracting the Surface from the Volume.

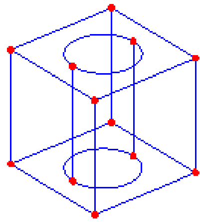
2 volumes



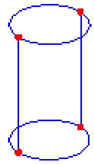
Union of cube and cylinder (simplify internal geometry) = 1 volume



Union of cube and cylinder (no simplify internal geometry) = 4 volumes



Subtraction of cylinder from cube = 1 volume



Intersection of cube and cylinder = 1 volume

Geometry From Mesh

An existing finite element mesh defined in a Solver results model file (.mys) file can be converted into features such that each element is converted into a single feature. Use of this facility will produce a similar model to that created by the use of the **File** > **Import** menu item which permits the import of a Solver data file(.dat) to create a model.

In using the Geometry from Mesh facilities, lower order features are automatically created, and the command describes the highest order of feature type that is to be created. For example, if a Line convert command is used on a single HX8 solid element mesh, then 8 Points would be defined at the node positions and 12 Lines would be defined from the edges of the HX8. As a result, be warned that very large models can be produced .

The currently selected elements are converted using the following menu items:

- ❑ **Geometry> Point > From Mesh** creates Points from nodes.
- ❑ **Geometry> Line > From Mesh** creates Points from nodes and Lines from beam elements, surface and volume element edges.
- ❑ **Geometry> Surface > From Mesh** creates Points from nodes, Lines from beam elements and Surfaces from surface elements and volume element faces.
- ❑ **Geometry> Volume > From Mesh** creates Points from nodes, Lines from beam elements, Surfaces from surface elements and Volumes from volume elements.

To convert from mesh the results database must be saved as a model file with access for writing prior to conversion. Firstly open the .mys using the menu item **File> Open,** then save as a model file using **File> Save As**. Select the elements you wish to convert to geometry and pick the appropriate **From Mesh** menu item.

Deformed mesh factor

The coordinates of all nodes in the mesh (used to define points) can be adjusted based upon a specified deformed mesh factor.

Notes

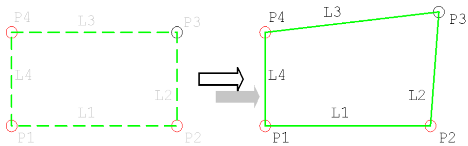
- No attributes will be converted.
- When converting to Volumes to resulting Volumes may be coalesced by removing the internal Surfaces.
- The conversion commands always create new features and cannot be used to edit existing features.
- Quadratic element edges with 3 nodes are converted to spline Lines with an exact match using the end node positions. A Point is defined at the mid-side node position but is not used in the Line definition.

Importing Mesh Data

Moving, Copying and Sweeping Geometry

Geometry may be moved or copied to new positions, or 'swept' to create higher order geometry from lower-order features using transformations. Compound transformations may be used in which a series of transformations are carried out in a specified order. When a feature is moved or copied, features will be merged as defined by the current merge status. See **Merging and Unmerging Features**.

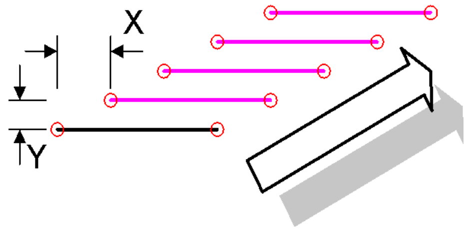
- ❑ **Move** When a feature is moved to a new location its lower order features will also be moved and its higher order features will be updated.. In the example shown here, Point 3 is moved using an X and Y translation. Due to feature associativity the definition of Lines 2 and 3 and of Surface 1 is automatically updated. Moving can be used to separate features on a temporary basis to assist in the manipulation of features, for example



when defining slidelines or joints.

Note: When moving holes the Surface is actually deleted and recreated so the feature numbers will not be maintained.

- **Copy** Features may be copied any number of times. When a feature is copied its lower order features will also be copied using the same transformations. Copied features will inherit the same attribute assignments as the original features. In the example shown here the Line at the bottom is copied 4 times using a transformation in the X and Y direction.



- **Sweeping Geometry** New geometry can be created by sweeping existing lower order geometry into higher order geometry using a transformation, as for example, by sweeping a Point to create a Line, or a Line to create a Surface, or a Surface to create a Volume. To do so a transformation or sweep type needs to be specified.

Transformations

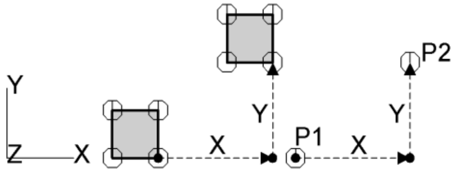
Transformations are used in two ways:

- When moving, copying or sweeping geometry a transformation is specified, and may be saved if required.
- For special applications such as to orient discrete point and patch loads, or to define a reflective mirror plane for thermal analyses.

Transformation attributes are defined using the **Utilities**> **Transformation** menu item, as well as from a move, copy or sweep dialog. Certain transformations can be defined by adding two or three geometric Points to **selection memory** before initiating the transformation command.

The following transformation types are available:

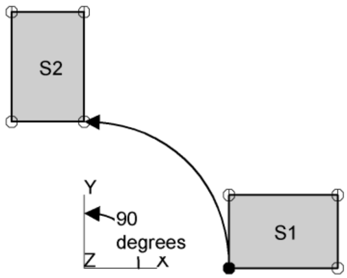
- **Translation** Linear translation along a specified vector coordinate. The vector coordinate will use the **active coordinate set** (See notes for an exception for cylindrical and spherical coordinate sets). Two Points added to selection memory can be used to define the vector coordinates. In this example, the translation is defined using Points 1 and 2, which stores a translation of X and Y. This is then used to copy the



Surface shown.

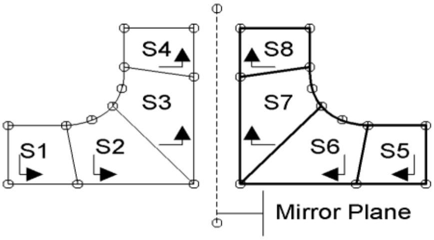
- **Rotation** (in a global plane) A specified angular rotation in either the XY, YZ, or ZX global plane at a specified origin.

  In this example Surface 1 is copied about the global origin through positive 90 degrees. A right hand corkscrew rule is used for rotations. Local coordinate systems can be used to rotate about non-global axes.

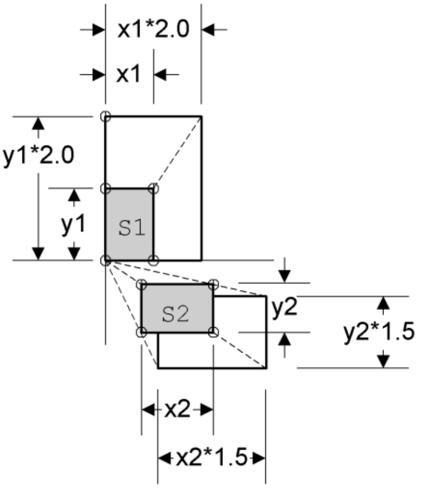


- **Mirror** A mirror plane may be defined by specifying three points in space to define an arbitrary plane, or two points to define a plane parallel to either the Z, Y, or X axis. Three Points added to selection memory can be used to define an arbitrary mirror plane, and two Points added to selection memory can be used to define a mirror line in the XY plane (not the YZ or ZX planes)

  Note:.Care should be taken when mirroring Lines and Surfaces as their orientations may be reversed so some Surfaces may effectively be turned upside down. Lines may also point in the opposite direction. See **Changing Geometry Orientation** to resolve any problems.

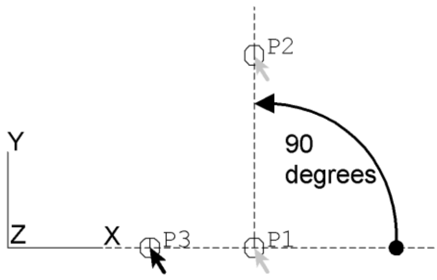


- **Scale** A scale factor and the origin point of the scale is specified .



- **Matrix rotation** A direction cosine either specified directly, or using two or three Points added to selection memory. Points in selection memory define a plane, the rotation of the global XY plane to this new plane defines the transformation. Note. If the determinant of the matrix is not unity then the effects may not be as desired.

  This example defines a rotation by defining a plane relative to the global Cartesian axis set by indicating three Points: at the origin (P1), along the local x axis (P2) and defining the local xy plane (P3). The resulting transformation is a rotation of 90 degrees about the global axes.



Notes

- In a transformation dialog, (including the move, copy, or sweep commands), click on the **Use** button to use a transformation defined from Points in selection memory.
- A transformation is not updated when the points defining it are changed.
- When a cylindrical or spherical coordinate set is active only X,Y and Z transformation input is supported.

Compound Transformations

Saved transformations may be used together to create a compound transformation, i.e. two or more transformations can be performed on selected geometry at once. To carry out a compound transformation (firstly from the **Utilities> Transformation** menu item) define the required transformations and save them with suitable names. Then when copying, moving or sweeping select the Compound option and specify which transformations to use by adding them to the right side box. Single transformations may be used more than once if required. The order in which the transformation is performed is significant, therefore click the up and down buttons to get the correct order, starting from the top.

Merging and Unmerging Features

When a new feature is generated, if its position coincides with an identical feature then by default the two features will be merged (removing one of the features) provided the merge characteristics are satisfied.

In addition to the automatic merging carried out during feature generation, any combination of features can be merged at any time. Care should be taken to merge from the lower order features upwards, as higher order features can only merge if defined by the same lower order features.

Merge/Unmerge Commands

The following merge/unmerge commands are available from the **Geometry** menu, under each feature type.

- ❑ **Merge** Merges all mergable features currently selected subject to the current merge characteristics and tolerance. Lower order features must be merged first. for example two Lines cannot be merged until the Points defining the Lines are themselves merged. When selected at the same time, LUSAS merges lower order features before higher order features.
- ❑ **Make Mergable/Unmergable** Sets the merge status of selected features. Merging can happen unintentionally in the normal course of events when additional features are defined in the same position as existing features. Using this command, it is possible to prevent two coincident features being merged by making one of the features unmergable. The merge status of an individual feature may be viewed, and altered, by displaying the properties of the selected feature, (right-click button), on the Hierarchy tab.
- ❑ **Unmerge** Duplicates or retracts selected features into higher order features which reference them in their definition. When a feature is unmerged from its higher order features, any new features defined are automatically set to be unmergable.

Merge Options

Several merge settings can be set from the **Geometry** tab on the **Model Properties** dialog.

- ❑ **Ask for Confirmation** Configures Modeller to prompt for confirmation before selected features are merged. (Confirmation tab).
- ❑ **Merge Tolerance** Controls the distance within which Point features must lie before they will be considered for merging. (Geometry tab).
- ❑ **Make New Features Unmergable** Sets the merge status of all new features to Unmergable. (Geometry tab).
- ❑ **Merge Characteristics** Controls the criteria that must be satisfied before features sharing a common definition will be merged. (Geometry tab). See below.

Merge Characteristics

Features will be merged only if they share common lower order features, in addition feature merging is dependent upon attribute assignments. By default, identical assignments must be found on two features, for those features to be merged. The merge type parameter controls how LUSAS handles feature merging. The following merge types are supported:

- ❑ **Off** where no merging is carried out.
- ❑ **Exact** where features are merged only if all assignments are identical. This is the default.

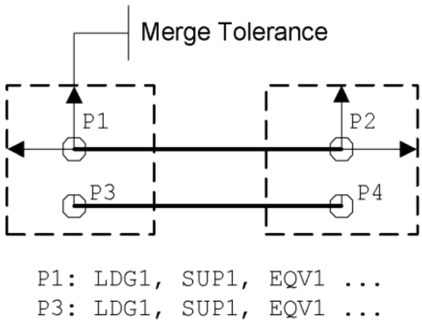
- ❑ **Wild** where features are merged if feature assignments of the same type for both features match. The assignments of both features are retained where the assignment type is unique to one feature.
- ❑ **Ignore Assignments** ignores the assignments when deciding if two objects should merge (this is the opposite of "Exact" where the two objects must have the same assignments to merge). The assignments of the feature merged out will be transferred to the feature retained unless the retained feature already has that particular assignment.

Notes

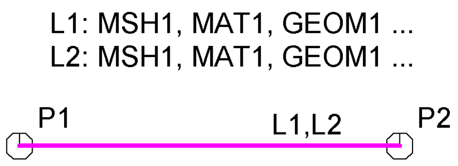
- If several features are merged with merge status set to ignore assignments and the assignments are indeed different, the remaining feature will inherit the assignment of the lowest numbered deleted feature.
- If two coincident features are not merged, two sets of coincident nodes will be generated when they are meshed and the finite element mesh will be unconnected at the feature discontinuity. This may be corrected at the meshing stage by merging the nodes using the equivalence facility.

Using the Merge (and Unmerge) Commands

- ❑ **Merge Point** Coordinates of Points to be merged must lie within the merge tolerance. By default assignments must be exactly the same for a Point to be merged.

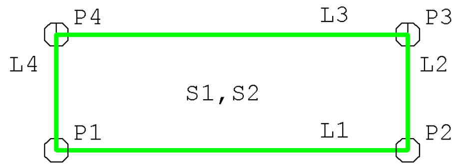


- ❑ **Merge Line** Lower order features must be common for merging to take place. By default assignments must be exactly the same for a Line to be merged.



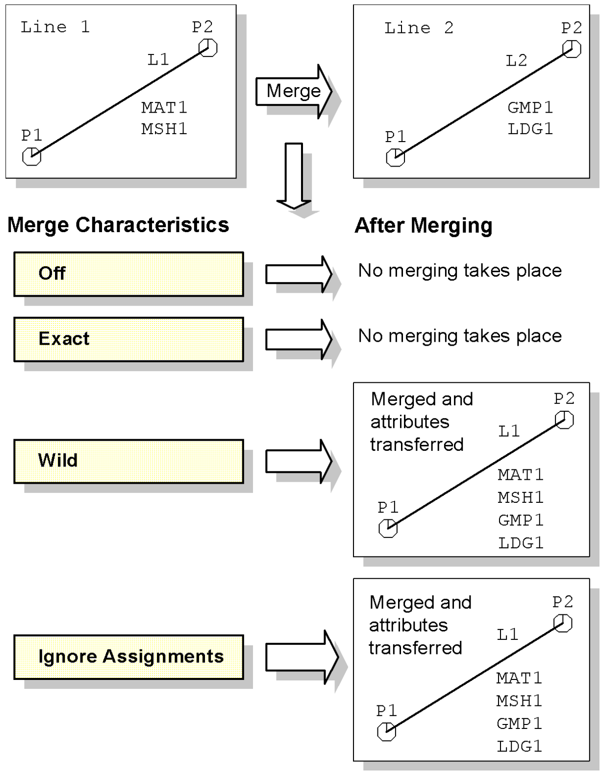
- ❑ **Merge Surface** Lower order features must be common for merging to take place. By default assignments must be exactly the same for a Surface to be merged.

```
S1: MSH1, MAT1, GEOM1 ...
S2: MSH1, MAT1, GEOM1 ...
```



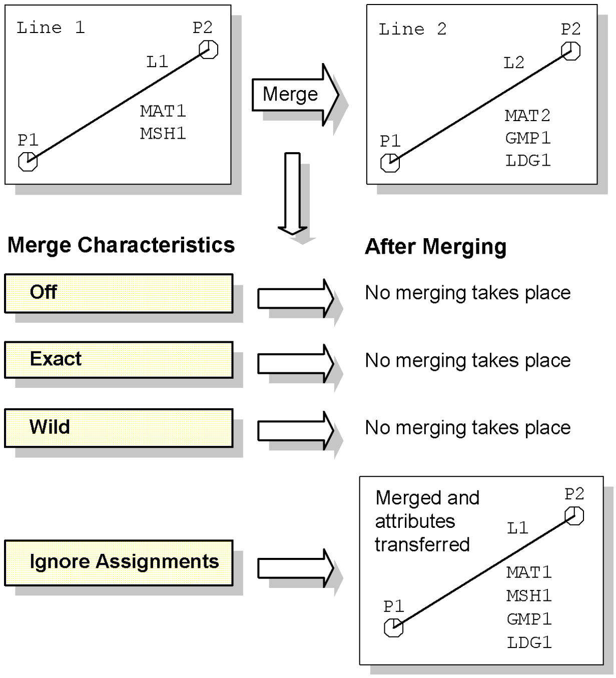
Merge Case Study 1

Merging Lines with different non-zero assignments. Wild and Ignore Assignments will merge Lines successfully.



Merge Case Study 2

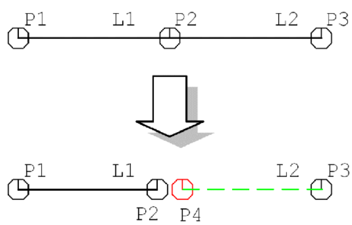
Merging Lines with additional material assignment on Line 2. Material assignments differ, therefore only Ignore assignments will merge successfully.



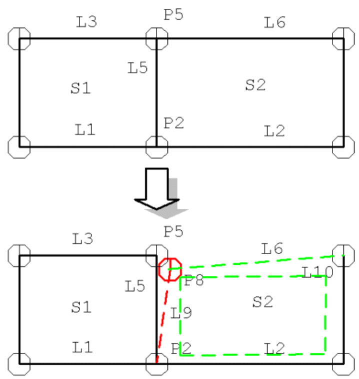
Using the Unmerge Commands

To unmerge a feature from a higher order feature (i.e. a point from a line) select both the feature to be unmerged and the higher order feature from which to unmerge it. In the following examples red represents "New" and green "Modified" geometry.

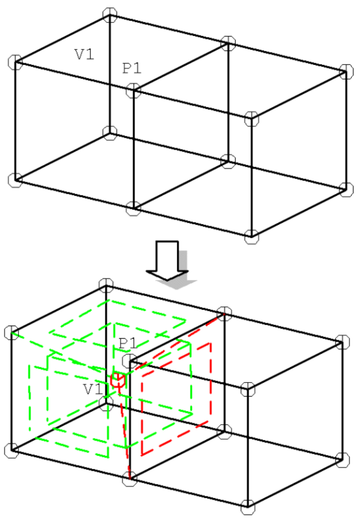
- ❑ **Unmerge Point in Line**Unmerge Point 2 in Line 2. A new Point 4 is copied from Point 2 and Line 2 is redefined using Point 4. Points 2 and 4 are coincident. Point 4 is unmergable.



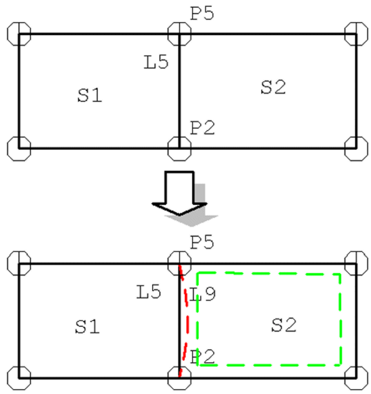
- **Unmerge Point in Surface**Unmerge Point 5 in Surface 2. A new Point 8 is copied from Point 5, a new Line 9 is defined and Line 6 is redefined using Point 8. New features are set to be unmergable. Point 5 and Point 8 are coincident.



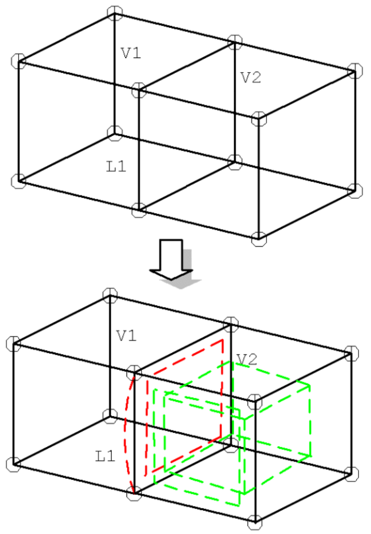
- **Unmerge Point in Volume**Unmerge Point 1 in Volume 1. A new Point, 2 new Lines and a new Surface are defined and affected Surfaces and the Volume are redefined. The new Point is coincident with Point 1.



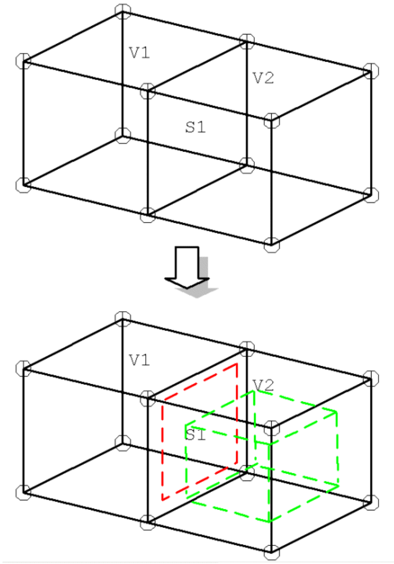
- **Unmerge Line in Surface**Unmerge Line 5 in Surface 2. A copy of Line 5 is defined which replaces Line 5 in the definition of Surface 2. Points are unaffected.



- **Unmerge Line in Volume**Unmerge Line 1 in Volume 2. A copy of Line 1 is defined joining the same end Points The new Line replaces Line 1 in the definition of the affected Surfaces in Volume 2



- **Unmerge Surface in Volume**Unmerge Surface 1 in Volume 2. A copy of Surface 1 is defined which replaces Surface 1 in the definition of Volume 2.



Case Study. Forcing Features to Merge

Sometimes when creating the geometry duplicate features are created. If these have different assignments the default merge setting will prevent coincident features from merging. In this case it is useful to request the merge settings to ignore assignments. LUSAS can be forced to merge duplicate features in the following way:

1. Set the merge status to ignore assignments by choosing the **File> Model Properties** dialog box, **Geometry** tab and pick **Ignore assignments** for the **Merge action**.
2. Select the whole model using **Edit> Select All**. Then merge the model features using the Merge Features button  on the Advanced Define toolbar and pick the option to merge defining geometry.
3. Reset the merge status to **Exact**.

Note: If some features have been previously unmerged these must first have their merge status reset to Mergable using **Geometry>** ***Feature type*****> Make Mergable**.

Changing Geometry / Element Orientation

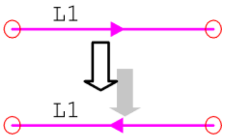
For feature-based models the orientation of the Geometry is generally used to define the local axes of the elements. For mesh-only models the local axes of the elements will be the same initially as those defined in the data file that was used to create the model.

Selecting features for re-orientation

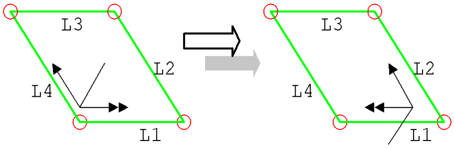
The following commands enable the local axes of Lines (where stated), Surfaces and Volumes (and Elements for a mesh-only model) to be re-oriented. First, the feature or element to be used as a basis for the re-orientation to be carried out must be selected (followed by the selection of any additional features to which the re-orientation of the first feature or element should apply) and then a menu item based upon **Geometry > (Feature) > (Command)** should be chosen.

- **Reverse** Lines, Combined Lines and Surfaces can have their direction reversed.

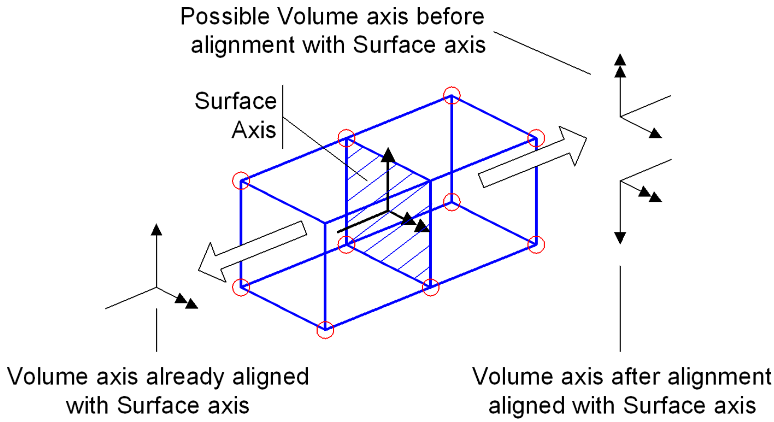
  This example shows the effect of a Line reversal. The local x axes of all elements on this Line will be reversed.



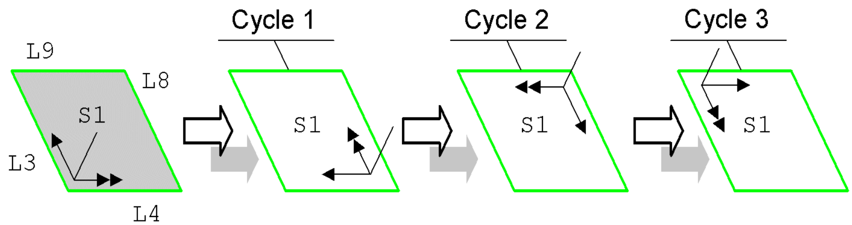
  In the Surface example the local x axis remains along the first Line in the Surface definition, but the Surface normal is reversed. Elements meshed on this Surface will be inverted.



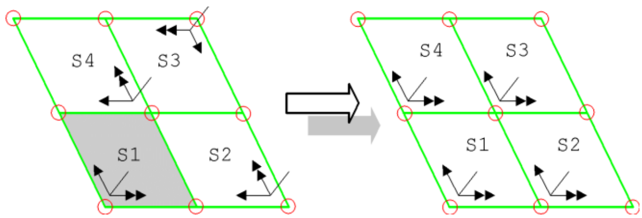
- **Axes to Surface** This command reorders the Surfaces that define a Volume such that the selected Surface becomes the first Surface in the volume definition. The z axis of the Volume is orientated normal to the first Surface, and pointing from that Surface towards the middle of the Volume. When the z axis of the Surface and the z axis of the selected Volume are the same their x, y and z axes will align. But if the z axis of the selected Surface and z axis of the selected Volume are in opposite directions (as may occur when Surfaces are shared between Volumes), the x axis of the Surface is used as the x axis of the Volume and the y axis of the Volume will be re-orientated accordingly. If required, the local x and y axes of the selected Volume(s) may be additionally changed using the Cycle Surface command.



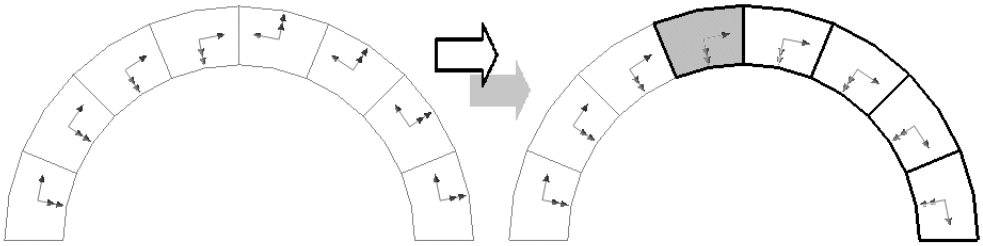
- **Cycle** Surfaces (or surface/solid elements in a mesh-only model) may have their definition order cycled, and Volumes may have the definition of the first Surface in the Volume definition cycled (and for mesh-only models, Elements may have the definition of the first Face in the Element definition cycled). In this example, the original Surface defined by Lines 1, 2, 3 and 4 is shown in grey. Cycling by changes Surface definition to Lines 2, 3, 4, 1. Cycling again defines the Surface as Lines 3, 4, 1, 2 . Surface normal directions remain consistent throughout.



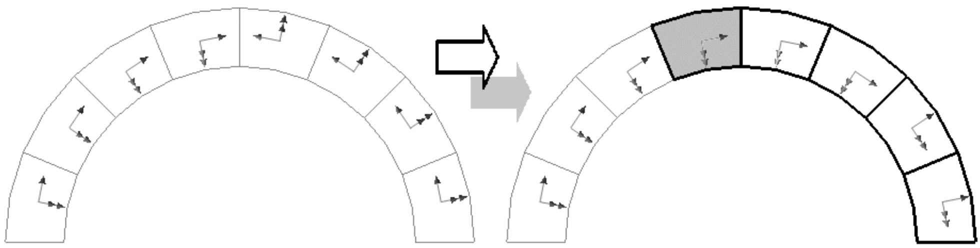
- **Cycle Relative** Cycles the definition order relative to the first feature of the same type in the current selection. Lines may have their axes aligned to a chosen Line, and Surfaces and Volumes (or surface/solid elements in a mesh-only model) may be reoriented in this way. In this example Surfaces 2, 3 and 4 are cycled using Surface 1 (shown greyed) as the reference orientation.



- **Cycle to Neighbours** (Mesh-only models) Cycles the element axes of all neighbouring and selected elements relative to the first element in the current selection. Whilst similar in effect to the Cycle Relative option for the example shown above (which would obtain the same result for that particular example), it caters for the case where, for certain orientations of elements, the cycle relative option would not be applicable. The example below shows one such example.



Cycle to Neighbours (correctly aligns element axes of the selected elements with initially selected neighbouring element)



Cycle Relative (only aligns element axes of the selected elements to best angle with respect to initially selected neighbouring element)

- ❑ **Cycle to Faces** (Mesh-only models) Solid elements may have their local z direction set from a selected Face. Element axes are defined by the direction of the axes on the first Face in their definition. This command reorders the Faces defining the Element such that the first Face in the definition is the selected Face. The local x and y axes of the element itself may then be changed using the Cycle command.

Notes

- When cycling Surfaces it is not the Lines that change, but the order of the Lines in the Surface definition. Since the Surface local x axis lies along the first Line in the Surface definition the orientation of the Surface changes.

- When cycling Volumes the cycle command only changes the order of Surfaces in the Volume definition. Since the Volume axes are determined by the orientation of the Lines that define the first Surface in the Volume, the orientation of the Volume changes.

Case Study. Changing Element Orientation For a Feature-based Model

When it is necessary for the local axes of Lines and Surfaces to be consistent this can be achieved by reversing and cycling the features. Consistent axes for underlying Lines, Surfaces and Volumes ensure that their elements also have consistent axes.

1. Draw the orientation axes for the features which are required to be consistent (Geometry Layer Properties). It may be useful to draw just Surface normals.
2. In the case of lines, reverse any lines whose orientation is to be changed.
3. In the case of Surfaces, reverse Surface normals (local z axis) and cycle Surfaces xy axes until all axes are consistent. Surfaces can be cycled relative to a reference Surface if required.
4. In the case of Volumes, set up the Volume local z axis and cycle xy axes by cycling the first Surface in the Volume definition until all axes are consistent. Volumes can be cycled relative to a reference Volume.

CAD Interfacing

CAD interfacing is the process of importing or exporting the geometry and other data from and to a CAD package.

When importing from CAD, only the relevant data should be exchanged. Annotation and construction lines should not be included as these will then be converted into LUSAS geometry. Control over the data imported into LUSAS is achieved using the **Advanced** button accessed from the **File> Import Geometry** menu item. Similarly, control over what is exported is achieved by options on the Export dialog accessed from the **File> Export** menu item. For more information see **Interface Files**

Chapter 5 : Model Attributes

Attributes

Attributes are used to describe the properties of the model. They are defined by using the **Attributes>** menu item. Once defined, attributes are assigned to geometry **features** (or to **mesh objects** in a mesh-only model) and are not lost if the geometry is edited, or if the model is re-meshed. Some attributes, such as material, geometric, and constraint attributes are always assigned on a per-analysis basis. Some attributes, such as activate, deactivate, support and loading attributes (usually) are assigned on a per-loadcase basis. Attribute assignments are inherited when geometry features are copied and are retained when geometry features are moved.

The attribute types are:

General Attributes

- **Mesh** describes the element type and discretisation on the geometry.
- **Geometric** specifies any relevant geometrical information that is not inherent in the feature geometry, for example section properties or thickness.
- **Material** defines the behaviour of the element material, including linear, plasticity, creep and damage effects.
- **Support** specifies how the structure is restrained. Applicable to structural, pore water and thermal analyses.
- **Loading** specifies how the structure is loaded.

Specific Attributes

- **Local Coordinate** provides a transformation for loads and supports, and an alternative to the global coordinate system.
- **Composite** defines the lay-up properties of composite materials in the model.
- **Slideline** slidelines control the interaction between disconnected meshes.
- **Constraint Equation** provides the ability to constrain the mesh to deform in certain pre-defined ways.

- **Thermal Surface** defines thermal surfaces, which are required for modelling thermal effects.
- **Retained Freedom** specifies the master nodes used in a Guyan reduction or superelement analysis.
- **Damping** defines the damping properties for use in dynamic analyses.
- **Birth and Death** allows elements to be added (birth) and removed (death) throughout an analysis, e.g. in a tunnelling process or a staged construction.
- **Equivalence** allows nodes which are close to each other but on different features to be merged into one according to defined tolerances.
- **Influence** attributes define the type of behaviour of the structure at and around an influence point
- **Age** defines the time between creation and activation of features in the model.
- **Search Area** restricts discrete (point and patch) loads to only apply over certain areas of the model.
- **Crack tip** define a crack tip attribute to allow a crack tip location to be defined at a point or line in a model.
- **Design strength** define strength data for use in conjunction with Design Factor Plots

Defining and Assigning Attributes

Attributes are defined from the **Attributes** menu. Defined attributes are shown in the Attributes Treeview and can be assigned to selected geometry features (or to **mesh objects** in a mesh-only model) by dragging and dropping them onto selected features of the model or assigning them from their context menu.

Attribute symbols explained

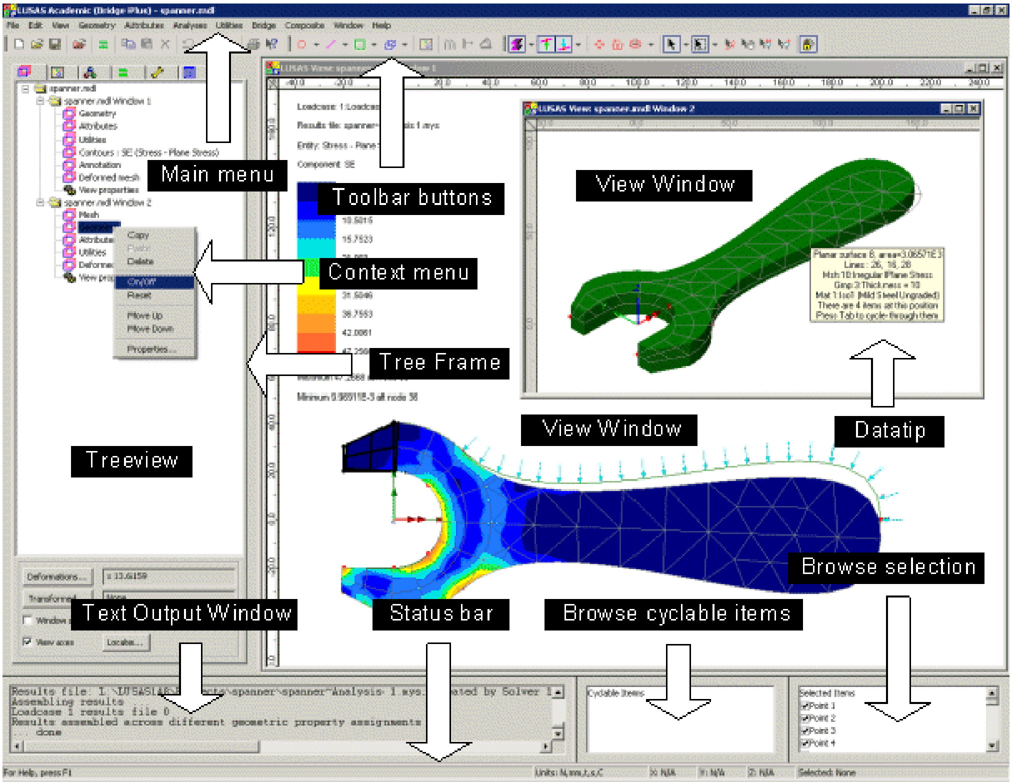
A symbol adjacent to each attribute name in the Attributes Treeview shows the status of each attribute present.

A coloured attribute icon shows an attribute has been assigned to the model or used in the definition of another attribute.

A greyed-out attribute icon shows an attribute has yet to be assigned to the model or used in the definition of another attribute.

A coloured or greyed-out attribute icon with a surrounding red box indicates an attribute that has been set as default, meaning it will automatically be assigned to features of the model as they are generated.

If any attributes that were previously created by a Prestress Wizard are edited outside of the prestress wizard they will be marked with a warning symbol to show that the Prestress Wizard must be re-run to recalculate new values.



Attribute manipulation

Attributes are manipulated using the context menu in the Attributes Treeview (accessed by clicking the right mouse button on the attribute), with the following commands:

- **Edit Definition** Allows the properties that were originally defined to be modified on the dialog that was used to create them.
- **Edit Attribute** Allows the properties of the attribute to be modified.
- **Copy** Enables the attribute to be copied and assigned to a selected group, or view window, in the current model using paste.
- **Rename** Attributes can be given meaningful names, for example 'Steel' to describe a material, or 'Beam - Four Divisions' to describe a Line mesh.
- **Delete** Deletes the selected attribute.
- **Visible** Makes visible all features to which the selected attribute is assigned.
- **Invisible** Makes invisible all features to which the selected attribute is assigned.
- **Set As Only Visible** Sets the whole model invisible and then makes visible only those features to which the selected attribute is assigned.

- **Advanced Visibility** Provides fine control over the visibility of features to which the selected attribute is assigned.
- **Results Plots** permits results for selected attributes to be selectively plotted.
- **Select Assignments** Selects the features that have the selected attribute assignment.
- **Deselect Assignments** Deselects the features that have the selected attribute assignment.
- **Visualise Assignments** Switches on and off the visualisation of features that have the selected attribute assignment. Note this is not an action but a change of state.
- **View Assignments** Shows loadcases and analyses to which attributes such as materials, supports and loadings etc. have been assigned.
- **Assign** Assigns the selected attribute to the selected features. The attribute will only be assigned to features where the assignment would be valid. Some attributes require further information in order to be assigned and in these cases a dialog is displayed. For attributes that can only be assigned once to a feature, assigning another attribute will overwrite any previous assignment of that attribute type.
- **Assign to all** Assigns the selected attribute to all currently defined geometric features (or mesh objects in a mesh-only model) and to all new geometric features as they are created. Some attributes require further information in order to be assigned and in these cases a dialog is displayed. A tick is displayed in the attribute context menu when Assign to all is switched on, selecting the Assign to all menu item again deassigns the attribute from all and removes the tick.
- **Deassign** Deassigns the selected attribute. Choose **From All** or **From Selection**.
- **Set Default** Automatically assigns the selected attribute to all new features as they are created.

Clarifying where attributes have been assigned

When Modeller is unable to automatically carry out an operation such as 'Set As Only Visible'' or 'Select Assignments' on a set of objects to which an attribute is assigned, clarification will be sought, requesting the user to identify an analysis or loadcase from those that include the chosen attribute.

Visualising Attribute Assignments

Attribute assignments can be visualised using:

- **Attributes layer** The Attributes layer is a window layer in the Layers Treeview that is normally added during the initial start-up of Modeller. The Attributes layer properties define the styles by which assigned attributes are visualised. The attributes layer properties may be edited directly by double clicking the layer name in the Layers Treeview. Assigned attributes can also be visualised individually by selecting an attribute name in the Attributes Treeview and clicking the right mouse button to choose **Select Assignments** from the context menu. This is the easiest way to interrogate the assignment of a single attribute. Note that the option **Visualise**

Assignments is not an action but a change of state. It sets the visualisation of assignments to 'on' or 'off', such that any assignments will be drawn or not drawn for that attribute.

- **Contour layer** (materials/geometry/loading attributes only) Allows the model to be contoured with a specified value obtained from the material, geometric or loading attribute assignments. This is especially useful when an attribute value changes across the model. e.g. when defined using a **variation**. To use this feature click the right mouse button in the view window (with nothing selected), choose Contours from the context menu, then select either **Loading (model)**, **Geometry (model)** or **Materials (model).**
- **Colour by attribute** (Accessed from the properties dialog of the Geometry layer) colours the geometry according to which attributes are assigned to which features. A key is generated to identify the colours.
- The options **Combine assignments using loadcase history** and **Show only assignments in the active loadcase** (accessed from the properties dialog of the Attribute layer for supports/activate attributes only) allows load dependent attributes to be visualised from the accumulated load history or from only those attributes assigned to the active loadcase.

In addition to the above, toolbar buttons for Supports and Loading can be used to turn-on and turn-off the display of any supports or loadings that are assigned to a particular loadcase.

- See **Geometric Properties** for the visualisation of beam cross sections and surface thickness (fleshing).
- See **Composites** for visualising composites materials.

Deassigning Attributes

Attributes may be deassigned from all or selected features by selecting the attribute in the Attributes Treeview with the right hand mouse button and picking the **Deassign** entry from the context menu. The menu item entries **From Selection** or **From All** may then be chosen to deassign from the items in the current selection or from all the features in the model. Unassigned attributes will be denoted with a greyed-out bitmap .

Attributes may also be deassigned by selecting **View Assignments** from the context menu for the attribute and unchecking the relevant loadcase.

Copying attributes between models

Attributes can be copied between models by using the Library Browser. See **Transferring Data Between Models** for details.

Deleting attributes from the Treeview

Attributes can be deleted from the Attributes Treeview individually by selecting the attribute name and pressing the Delete key, or choosing 'Delete attribute' from its context menu. Attributes can also be deleted en-mass by attribute type (e.g. Mesh, Geometric, Supports, Loading etc), by selecting an attribute folder name and selecting **Delete all** from its context menu. Deletion of selected attribute data can be carried out in one go using the Library Browser. See **Transferring Data Between Models** for details.

Drawing Attribute Labels

Labels are a layer in the Layers Treeview. To display attribute labels:

1. With nothing selected, click the right mouse button in the graphics area and choose **Labels** from the context menu.
2. Switch on labels for the chosen attribute type.

Set Default Assignment

Certain attributes, (mesh, geometric, material, composite), can be assigned automatically to all newly created features. Default attributes are set by right-clicking the attribute in the Attributes Treeview , then choosing the **Set Default** entry from the context menu. This is useful for models with constant materials or thickness throughout, or where the same element is to be applied to all features. Attributes that are set as default are displayed with a red box around them in the Attributes Treeview.

Attribute dependency across analyses / loadcases

See **Base Analyses** for a table that summarises how attributes assigned in a base analysis are inherited in other dependent analyses.

About Meshing

Feature-based models are defined in terms of geometry **features** which are sub-divided into finite elements for analysis. This process is called meshing. Mesh attributes contain information about:

- **Element Type** Specifies the element type to be used in a Line, Surface or Volume mesh attributes may be selected either by describing the generic element type, or naming the specific element. See **Element Selection**.
- **Element Discretisation** Controls the density of the mesh, by specifying the element length or the number of mesh divisions, spacing values and ratios.
- **Mesh Type** Controls the mesh type e.g. regular, transition or irregular.

For feature-based geometry models, mesh attributes are defined from the **Attributes> Mesh** menu item for a particular feature type i.e. Point, Line, Surface or Volume. The mesh attributes are then assigned to the required features. The orientation of the model Geometry is used to define the local axes of the elements. See **Changing Geometry / Element Orientation** if element axes need to be changed. Assigned mesh attributes apply to all analyses defined for that model.

When modelling 2D frame problems it is recommended to use the relevant family of 3D beam elements (rather than 2D elements) if, at a later date, it is intended to convert the 2D model into a full 3D model.

Mesh-only models

Mesh-only models are comprised of nodes and elements and do not contain any geometric feature types, or indeed any geometric data at all. The number and shapes of the elements of a mesh-only model are fixed. The type of element may be changed and this is done by use of the **Change Element Type** option on the context menu on the element group name. In doing so, the number of nodes defining the element topology may be reduced but not increased. For instance, an 8-noded brick elements may be defined for use on previously defined 20-noded brick elements. See **Changing Geometry / Element Orientation** if element axes need to be changed.

Mesh Types

Various mesh patterns can be generated:

- **Regular** Only used on **regular** Surfaces and Volumes. Any element shape may be selected for regular meshing. Options exist to automatically allow transition or irregular meshes to be generated when regular meshing is not possible.
- **Irregular** Used for Surfaces and Volumes. An irregular Surface mesh may consist of triangular or quadrilateral elements. A irregular Volume mesh must consist of tetrahedral elements. Irregular Volume meshes will only be generated if specified as acceptable in the mesh attribute.
- **Interface Meshes** Only applicable to joint and interface elements.

Mesh Visualisation

The Mesh layer properties control how the mesh is displayed in the current Window. The same controls are available for the undeformed mesh and the deformed mesh, but since they are different layers in the Treeview, different properties can be applied to each layer.

Mesh settings

- **Wireframe** Displays the element mesh as a wireframe using the pen specified. Only the visible mesh lines are drawn.
- **Solid** Displays the mesh as solid panels using the colour specified. Click on the coloured square to change the colour used.
- **Maximum shade** Controls (using a percentage value) the darkness of any specified solid mesh colour.

- **Outline only** Displays only the outline of the mesh for each assigned feature. This is also useful for spotting cracks or discontinuities in the mesh due to features not being merged or equivalenced correctly.
- **Show nodes** Draws the mesh nodes. Nodes define the vertices of elements.
- **Hidden parts** Displays all elements and not just the visible element faces. **Dotted** displays the hidden edges in a dotted line.
- **Show normals** Displays the element normal for Surface elements.
- **Show element axes** Displays the element axes as a local axis set.
- **Orientations only if selected** Displays the surface normal or element axes only if the element is selected.
- **% of elements remaining** Shrinks the elements to the percentage specified.
- **Colour by** Enables the mesh colour to be changed.
  - Mesh colour - colours elements in default mesh colour.
  - Group - colours elements by group.
  - Connectivity - colours element edges by number of neighbours.
  - Element Type - colours elements by element type.
  - Normals - colours elements by surface normal direction.

Note. The arrow sizes used for element axes and normals are defined on the **Default** tab of the **Model Properties**.

Visualise settings

- **Joint elements** Marks any joint elements in the model with a symbol.
- **Active mesh** Marks the active elements with a symbol.
- **Beam end releases** Draws the beam end release for elements that use end releases.

Meshing Points

Point mesh attributes are used to assign non-structural point mass elements, joint elements and mesh spacing parameters to the model. Point mass and spacing attributes are assigned to a single Point whereas joint attributes are assigned to pairs of Points. The first Point is referred to as the Master, the second is the Slave. Joint property assignments should be made to the Master Point.

Meshing Lines

Line meshing is carried out by defining a Line mesh attribute and assigning this to a selected Line. Line mesh attributes are defined from the **Attributes> Mesh> Line** menu item. The number of elements can be specified using either element length or number of divisions.

Note that when modelling varying cross sections with constant section beam elements care should be taken to ensure that sufficient elements have been assigned. Greater than 8 elements should be used for small variations in cross section along the length of the geometry to which the section has been assigned, and considerably more elements should be used for larger variations.

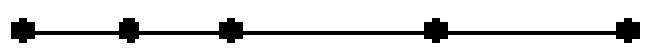
Line Mesh Spacing

By default elements are evenly spaced but this can be user defined. Non-uniform spacing is specified by clicking on the Spacing button from the Line mesh dialog then using one of the following methods:

- ❑ **Uniform Spacing**

- ❑ **Uniform Transition** Specify a ratio of the first to last mesh division length. The spacing ratios are assigned in the direction of the line to which they are applied. This example uses a ratio of 4 which is the ratio of the length of the first element to the last. To reverse the mesh spacing the ratio could be specified as 0.25 (or the Line could be reversed).

- ❑ **General Spacing** Enter a grid of numbers which defines the individual segment length ratios explicitly. They are specified in the form:



  - **Number** Defines the number of elements at this ratio, (from the start of the line as defined by the line direction). The numbers must add up to the number of divisions specified, e.g. for the example below 2+2 = 4 divisions.

  - **Ratio** Defines the spacing ratio of the elements in the 1st column to the total number of elements.

  This example uses general spacing 2@2, 2@4 (spacing ratios are applied in the direction of the Line).

When specifying spacing the Line direction is important as the spacing is defined from the start to the end of the line. If the spacing appears to be in the incorrect direction the line may be reversed by selecting the Line and using the **Geometry> Line> Reverse** menu item.

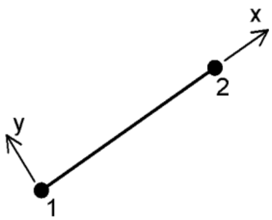
If desired, the element spacing can also be defined using a **background grid mesh**. The use of a background mesh is specified when the mesh attribute is assigned to the Line.

Line Element Axis Orientation

The element x axis always runs along the Line. Orientation of the local y and z axes of 3D beam elements may be defined using a beta angle or a local coordinate when the Line mesh attribute is assigned to the Line. By default the element z axis coincides with the global Z axis and the element y axis forms a right hand set. Elements may also be orientated using a **local coordinate** which is assigned to the geometry.

End Releases for Beam Elements

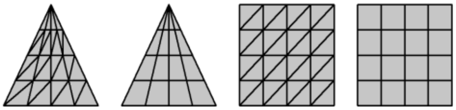
Freedoms at the ends of a Line can be freed to rotate or translate using an element with end releases. See the *Element Library* for more information on these element types. When defining a Line mesh attribute, with a valid element selected, click on the End Release button. Releasing beam element end freedoms can be used as an alternative to using a **joint** element, for example when defining a pin between two beams.



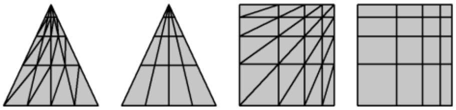
Meshing Surfaces

Regular Surface Meshing

Regular meshing is used to generate a set pattern of elements on Surfaces and Volumes. Only surfaces which are regular (defined by 3 or 4 lines) can be meshed using a regular mesh pattern.



- In order to generate a regular grid mesh pattern the number of mesh divisions on opposite sides of the Surface must match. If they do not match transition patterns will be used (if allowed in the mesh attribute definition). The examples shown here mesh triangular and quadrilateral Surfaces using both triangular and quadrilateral elements.



- The Surface mesh may be graded using mesh spacing parameters in 'None' element Line meshes assigned to the boundary Lines. In the examples shown here mesh spacing has been used to bias the elements into the apex of the triangle or one corner of the rectangle.

Irregular Surface Meshing

Irregular meshing is used to generate elements on any arbitrary Surface.

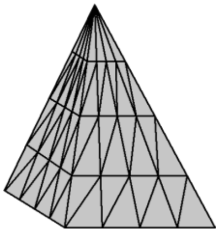
- **Element Size** specifying the required approximate element edge length.

Meshing Volumes

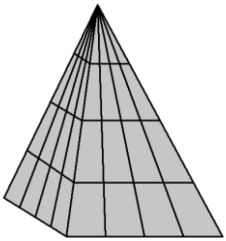
Volumes are meshed using regular mesh patterns, transition mesh patterns, or irregular tetrahedral meshing.

Tetrahedral Volumes

Tetrahedral Elements

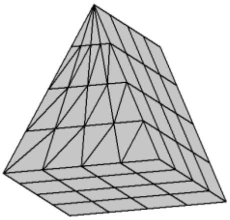


Pentahedral/Tetrahedral Elements

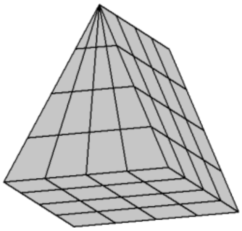


Pentahedral Volumes

Pentahedral Elements

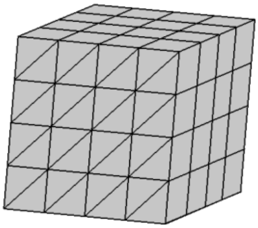


Hexahedral/Pentahedral Elements

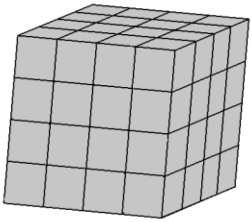


Hexahedral Volumes

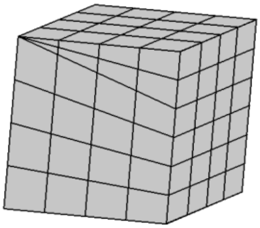
Pentahedral Elements



Hexahedral Elements



Hexahedral/Pentahedral Elements



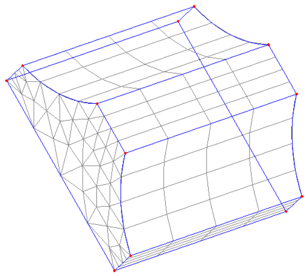
The mesh density for volumes is taken from the boundary surface mesh density.

Regular Volume Meshing

In order to generate a regular mesh pattern the number of mesh divisions on opposite faces of the volume must match. If they do not match then transition patterns will be used. Pentahedral/tetrahedral elements will automatically be inserted in the appropriate positions of a transition mesh.

Extruded Irregular Mesh

Volumes defined by sweeping an irregular Surface may be meshed with a regular Volume mesh attribute. The interconnecting Lines between the irregular end Surfaces must all be straight, or all minor or major arcs with a common axis of rotation. The side Surfaces must all be defined by four Lines so they can be meshed with a regular grid of quadrilateral faces. The irregular opposite Surfaces must not share any common boundary lines therefore wedge-shaped Volumes cannot be meshed as extruded irregular Volumes.

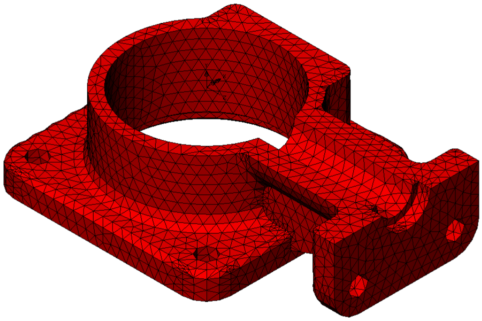


Case Study: Meshing Volumes by Extruding Irregular Surfaces

It is possible to mesh an irregular volume with hexahedral or pentahedral elements if the volume has been formed by sweeping an irregular surface.

1. Define an irregular Surface with more than 4 sides.
2. Define a Volume by sweeping the irregular surface.
3. Define a Volume mesh attribute with Hexahedral or Pentahedral elements, use a regular mesh with automatic divisions to ensure an equal number of divisions on the swept edges.
4. Assign the Volume mesh attribute to the Volume.

Irregular Tetrahedral Meshing



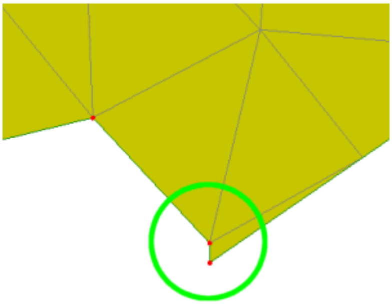
Arbitrary shaped irregular Volumes defined by any number of Surfaces may be meshed with tetrahedral elements. The element size may be specified on the mesh attribute, taken from the defining geometry or interpolated from a background grid. The mesh may be refined around small features and stress concentration using 'None' Surface and Line mesh attributes. By default the maximum angle around an arc subtended by a single element is 90 degrees. This may be adjusted on the **Meshing** tab of the **Model Properties** dialog.

Notes

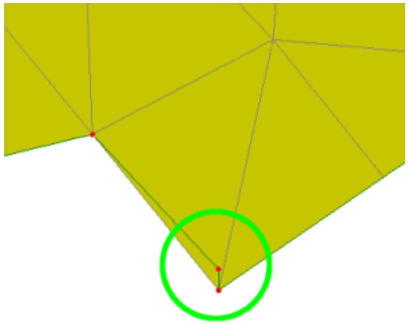
- A good initial mesh is usually obtained by specifying the element size as approximately1/50th of the diagonal model size. Specifying too small an element size will cause too many elements to be generated and may result in LUSAS using up all the available memory. Specifying too large an element size will cause the meshing algorithm to fail.
- The success of tetrahedral meshing is dependent on the quality of the Surface mesh. If the meshing algorithm fails, invoke edge collapsing or set the Volume mesh to "From defining geometry" and adjust the element size using Line and Surfaces mesh attributes of type None. If the meshing still fails try breaking the Volume into a number of smaller Volumes.

Edge Collapsing

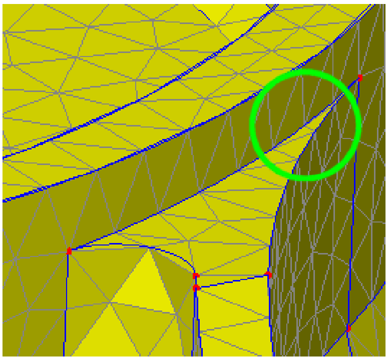
The quality of the mesh may be improved using edge collapsing. Edge collapsing removes elements with very short sides or acute angles by merging them with neighbouring elements. This is particularly useful when generating tetrahedral elements on imported CAD models where very short lines are present. Edge collapsing is invoked from the Advanced button on the **Meshing** tab of the **Model Properties** dialog.



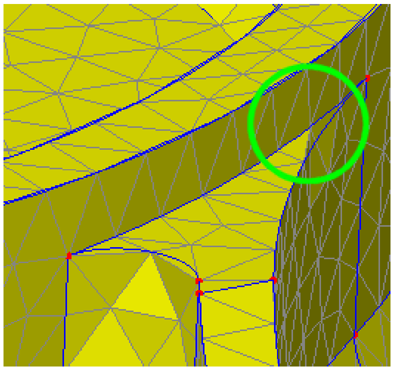
Mesh before edge collapsing



Mesh after edge collapsing
(short edge removed)



Mesh before edge collapsing



Mesh after edge collapsing
(Elements with small subtended angles removed)

Controlling the Mesh Density

The simplest way to define the mesh density is to define the number of divisions to be used in the mesh attribute. This method should only be used for simple models because changing the mesh density when multiple mesh attributes have been defined is both time consuming and prone to error. For most model the mesh density should be controlled using boundary discretisation.

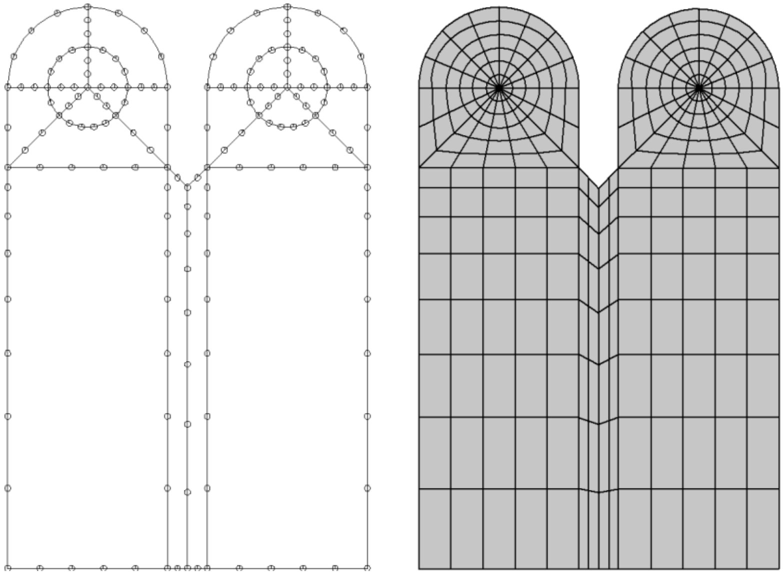
Boundary Discretisation

In the case of Surface or Volume meshing, the number of mesh divisions may either be specified directly in the Surface or Volume mesh attribute, or using Line or Surface mesh attributes of element type ‘None’. In many realistic problems, where several Surfaces or Volumes exist, using attributes with an elements of type 'None' is the most convenient way to define the mesh density. For Lines the spacing is specified using either element length or number of divisions and for Surfaces the mesh size is specified as the element edge length.

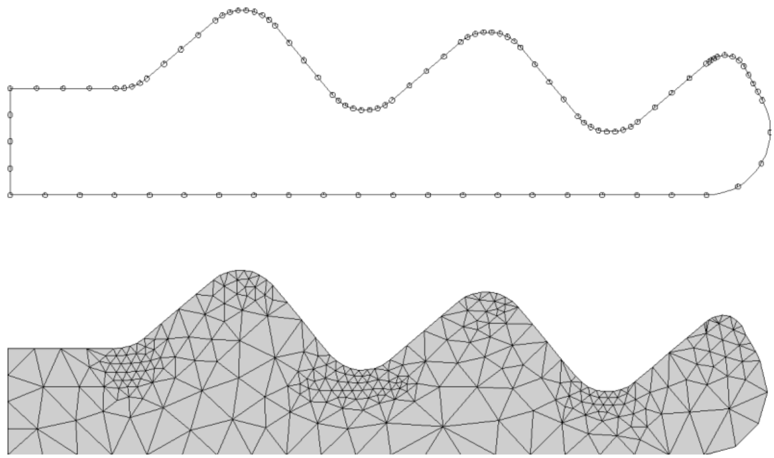
Notes

- If the element size is specified differently in the Line and Surface mesh attribute the Line element size will be used.
- If the element size has not been specified the **default number of mesh divisions** will be used.

❑ **Regular Surface Meshing** The applied boundary discretisation (left) produces the irregular mesh pattern on the Surface (right).



❑ **Irregular Surface Meshing** The applied boundary discretisation (top) produces the irregular mesh pattern on the Surface (bottom).



Default Number of Mesh Divisions

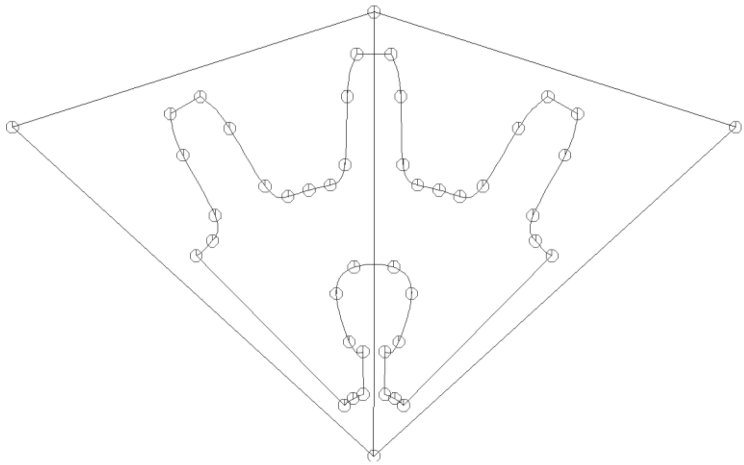
If the discretisation has not been specified in the mesh attribute, or by using a Line mesh of element type 'None' the Line will be sub-divided according to the default number of mesh divisions. This is specified on **Meshing** tab of the **File> Model Properties** dialog.

Background Grid Meshing

Background grid meshing is a method of controlling the size of elements generated during automatic meshing. It is generally only used when specification of spacing and stretching parameters at Points is required to grade the mesh pattern locally when irregular surface meshing.

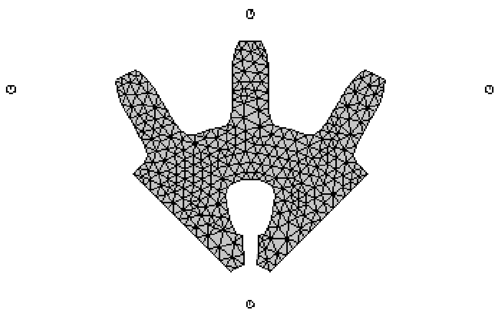
A background grid is a collection of triangular or tetrahedral shapes which completely encompasses the features to be meshed. A Line, Surface or Volume mesh is used to define the element type in the usual way and point meshes assigned to the points of the background grid are used to control the element size in the vicinity of each point. Finer control is achieved by using more Points in the background grid definition or by using Line mesh assignments to override the mesh size on specific edges.

The background grid may be specified explicitly from Points at each vertex or generated automatically. Any mesh distortion required may be entered using the point mesh stretching parameters. If generated automatically, tetrahedral shapes will always be used.



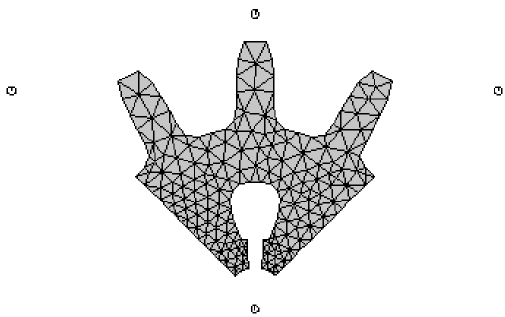
Constant Mesh Spacing

Same spacing parameters (Point meshes) are assigned to all Points in background grid.



Varied Mesh Spacing

Different spacing parameters (Point meshes) are assigned to the top Points (spacing=7) and the bottom Points (spacing=1) in the background grid.

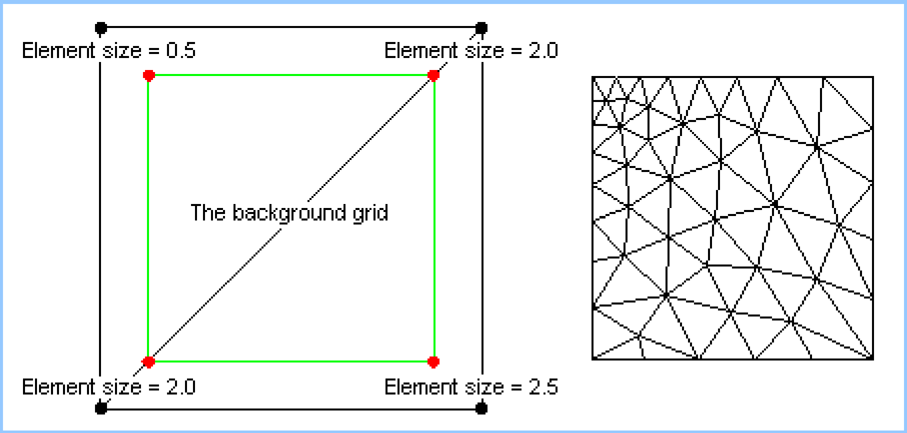


Notes

- Background grid meshing requires all Points defining the background grid to have a point mesh assignment.
- To remove a background grid delete the Background Grid from the  Treeview.

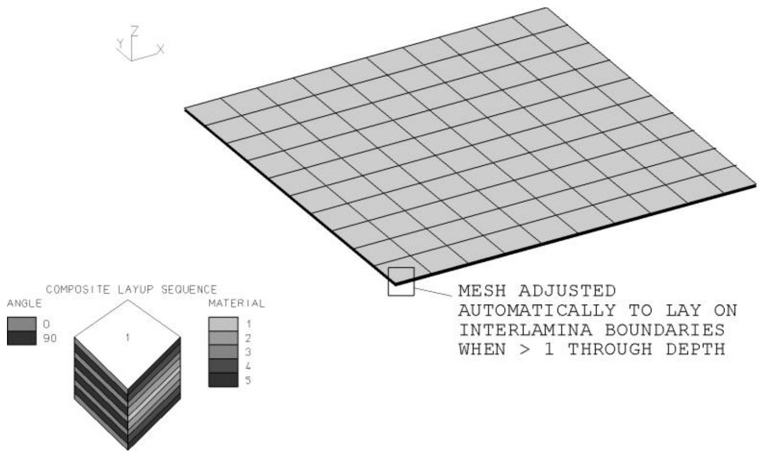
Case Study. Using Background Grid Meshing

1. With a Surface or Volume already drawn and selected, define a background grid by selecting the **Utilities> Background Grids** menu item and choosing **Enclose Selection**. Define one element in each direction and give the background grid a name.
2. Define Point mesh attributes from the **Attributes> Mesh> Point** menu item with the desired element sizes for use at assigned points on the background grid. For example Point mesh spacing attributes of 0.5 and 2.5 might be used for opposing corners of the grid, and a spacing of 2.0 might be used for the remaining points.
3. Assign the Point mesh attributes to the appropriate Points on the Background Grid. All points must be assigned a point mesh attribute.
4. Define a Surface or Volume mesh attribute with an **Irregular mesh** type, leaving the element size blank.
5. Select the Surface or Volume and assign the Surface or Volume mesh attribute to the model selecting the **From background grid** in the mesh assignment dialog. The mesh generated will be based upon the governing point mesh values defined for the background grid.



Composite Material Assignment

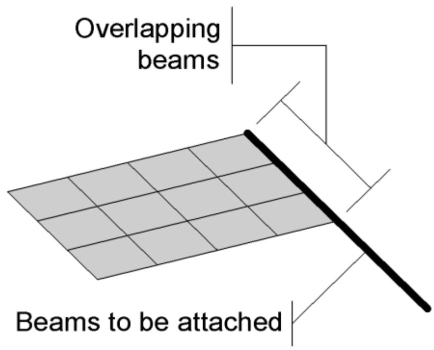
When a Volume feature with a composite material assignment is meshed the nodes are moved onto the composite layer boundaries. This ensures an exact number of layers in each element.



Connecting Beam and Shells

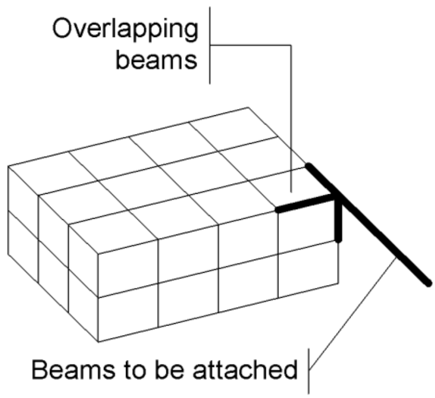
Beam Shell Connectivity

Extend the beams along the edge of the shell indicated by thick lines.



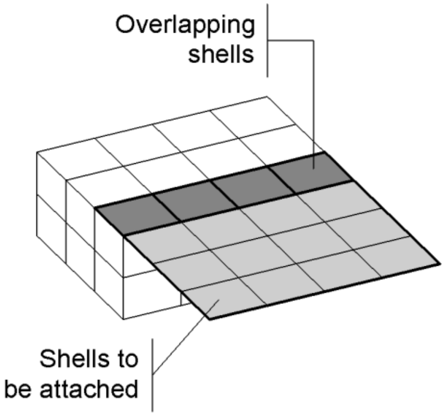
Beam Solid Connectivity

Extend the beams along the edge of the solid elements indicated by thick lines. Torsion is restrained using out of plane beams.



Shell Solid Connectivity

Extend the shells over a portion of the solids indicated by the dark shaded area.



Case Study: Connecting Shells and Solids

Solid and shell elements may be connected but the procedure is not as straightforward as it first may appear. Solids and shells have different sets of nodal freedoms and the rotational freedom present in the shells can only be passed through to the solid elements by extending the shell around the side of the solid, thus passing through the rotation via combined translation effects. This form of connection stops rotation relative to a solid which only has translational degrees of freedom.

The following procedure outlines the general method of fixing shells to solids:

1. Define the Surfaces and Volumes.
2. Define suitable mesh attributes, for example define linear hexahedral elements and linear quadrilateral shell elements and assign these to the Volume and Surface parts of the model.
3. Now assign the surface mesh attribute to a surface that forms part of the solid elements and which shares a common edge with the shell Surface that is being fixed to the solid part of the model. Do not forget to assign material and geometric attributes to the surface attached to the solid.

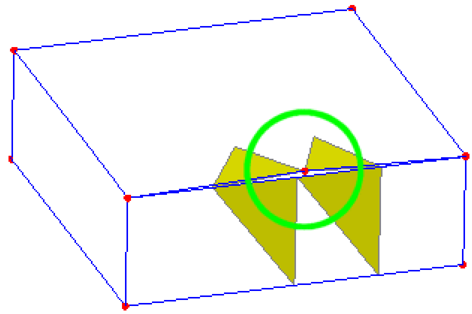
Note: It is advisable to make a connection such as this reasonably distant from the main area of interest as it may affect the quality of the results locally.

Fixing Mesh Problems

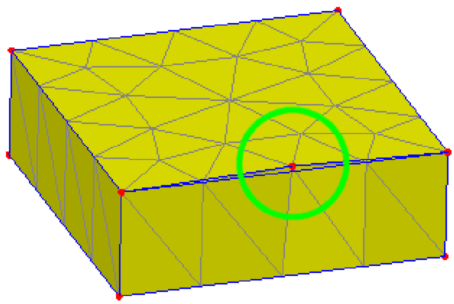
When meshing, any features which failed to mesh are noted in the text window. These can be identified using the Identify Object dialog invoked by double clicking on the error message in the **text window**. Alternatively, a group can be created containing all the features that failed to mesh. Only features that are not part of meshed higher order features will be added to the group. To activate this facility choose the check box **Create a group of features that failed to mesh** invoked from the Advanced button on the **Meshing** tab of the **Model Properties** dialog.

When importing CAD models meshing errors can sometimes occur due to very short lines and small surface slivers in the model. These problems can be alleviated in the meshing process by using **edge collapsing**.

When meshing open **hollow volumes** with tetrahedral elements the edges with nodes which have failed to merge are displayed automatically.



Unmerged nodes on volumes which failed to mesh.



Mesh with nodes merged by adjusting node merge tolerance

Notes

- Unmerged nodes are most easily seen when meshed using solid fill.
- Nodes which have failed to merge may be forced to merge by adjusting the **node merge tolerance** on the volume properties dialog or by assigning a suitable line mesh of type None to the lines.

Mesh Utilities

Mesh utilities provide the means to query distances between nodes; to control the meshing and re-meshing of all or parts of a model and to use a defromed mesh as a starting point for a further analysis. Mesh utilities are accessed from the **Utility** > **Mesh** menu item

- ❑ **Distance Between Nodes** - displays in the Text Output window the relative distance between any selected nodes.

- **Show Closest Nodes** - displays in the Text Output window the distance between the closest nodes of any of those selected.

To control changes being made to a mesh the following menu items can be used in various inter-related ways.

- **Mesh Lock** - Disables automatic remeshing of a model and prevents any changes to a mesh being made. LUSAS automatically locks the mesh on a model when a results file is loaded because any subsequent mesh changes may lead to the assembly of results in a misleading fashion. Mesh changes involve renumbering or reorientation of elements and results are associated with node and element numbers.. Therefore results requested after a remesh (without the mesh being locked) may appear in the wrong location and in the wrong order in the structure.You can unlock the mesh using the menu item Utilities > Mesh > (uncheck) Mesh Lock and Utilities > Mesh > Mesh reset
- **Mesh Reset** - Deletes the current mesh and forces a complete re-mesh of the whole model using the assigned mesh attributes. If Mesh Lock is 'on' this will not occur.
- **Mesh Now** - meshes the whole model regardless of any Mesh Lock being set.
- **Mesh Selected Items** - meshes only those features selected.
- **Use Deformations** - uses the deformed mesh caused by one analysis to be used as the starting point for a further analysis. The mesh may be tabulated with node coordinates computed from the deformations in the active loadcase multiplied by a specified factor. To do so, a model file and its results file must be loaded with the required results loadcase set active.

Checking mesh refinement in a model

Checking the mesh refinement in a model is an essential part of any finite element analysis. By nature, all finite element analysis is an approximation, a "model" of a real (or potentially real) object. The results will only be accurate if the mesh is defined in such a way as to simulate the change in load effects across the structure effectively.

The appropriate degree of mesh refinement can only really be discovered by experimentation: increasing the refinement and observing changes in the results. When refinement produces a negligible change in key results, the level of refinement required has been exceeded. A "negligible" change in results might be where the inaccuracy derived from the FE analysis may be considered small by comparison to other assumptions inherent in the design calculations. There is no hard rule because the load effects vary across "real" structures according to the structural form and loading conditions.

It is normal to find that a higher level of refinement is needed in certain parts of the structure compared to other areas. Changing between linear and quadratic order elements can help identify the behaviour of the structure and the refinement considerations.

Joint and Interface Elements

- **Joint elements** are used to connect two or more nodes with springs having translational and rotational stiffness. They may have initial gaps, contact properties, an associated mass and damping, and other nonlinear behaviour.

- **Interface elements** are used for modelling interface delamination in composite materials.

Both joint and interface elements may be inserted between pairs of corresponding nodes and features. Joint elements can be inserted between points, lines and surfaces. Interface elements can only be inserted between lines and surfaces.

Defining and Assigning Joints / Interface Elements

A Joint element is defined as a Point, Line or Surface mesh attribute using the **Attributes > Mesh** menu and specifying the structural element type to be used. Once defined, it is assigned to the model in one of two ways:

- **To a single pair of features** This requires two points (joints only) or two lines or surfaces to be selected. The first selected feature becomes the master. The second selected feature becomes the slave.
- **To multiple pairs of features** This uses **selection memory** to define a set of slave features prior to selecting a set of master features.

Defining and Assigning a Joint / Interface Element Mesh to a Single Pair of Features

To model a single joint element between a pair of features (two points, two lines or two surfaces, when applicable):

1. Define a Joint mesh attribute with the chosen joint element.
2. Select the first (master) feature.
3. Add the second (slave) feature to the selection.
4. Assign the Joint mesh to the two features. Options exist to allow definition of the local axes of the joint element.

Defining and Assigning a Joint / Interface Element Mesh to Multiple Pairs of Features

To define joint elements between multiple pairs of features (two or more sets of points, lines or surfaces, when applicable):

1. Define a Point, Line or Surface mesh attribute with the chosen joint element.
2. Select the slave features and add them to **selection memory**.
3. Select the master features.
4. **Assign** the joint mesh.

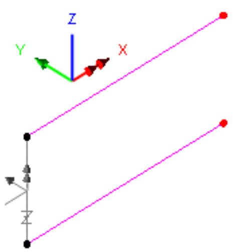
In making multiple selections of points, lines or surfaces it is important to select the separate slave and master features in the same order to ensure that the joints / interface element mesh created map to the correct features.

As the same joint mesh attribute is assigned to both master and slave features, a mesh pattern is created between the two features, with the mesh definition for the feature determining the number of joints generated in the joint interface mesh (see Examples of Assigned Joints that follow). When using interface meshing the joint elements are automatically created, joining all nodes on the master and slave features, and each joint stiffness is automatically computed from the representative length or area of the elements on the master/slave features.

Notes.

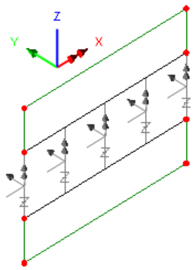
- Joint meshes require Joint material properties to be assigned to them.
- For joints with rotational degrees of freedom an eccentricity must be specified. An eccentricity of zero may be specified.
- The Joint symbol is drawn at the quarter-point along the joint nearest to the first master feature defining the joint element.
- Joint properties should be defined per unit length when assigned to Lines and per unit area when assigned to Surfaces.
- Master features hold the mesh assignment data. A point can only hold one joint assignment so if multiple joints are to be assigned to a single point that point must be designated the Slave by ensuring it is the second point in the selection.
- Joints defining spring supports act relative to the initial, unstressed configuration rather than that on any previous loadcase. This means that to introduce a stiffness to a support in a loadcase during a staged nonlinear analysis, relative to the stressed state on the previous analysis, you will need to use joints and activate them on this loadcase.

Examples of Assigned Joints/Interface Element



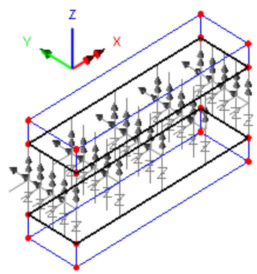
Joint between two points

(Lower point is the master)



Joint / Interface mesh between two lines

(Lower line is the master, four line mesh divisions are assigned to line)



Joint / Interface mesh between two surfaces

(Lower surface is the master, four line mesh divisions are assigned to each line)

Joint Local Axis Direction

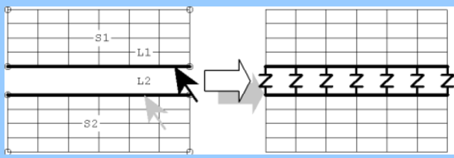
The joint local axis direction is defined when the mesh is assigned. Three options are available:

- **Follow point axes** (Default selection) Adopt the axes assigned from the Local Coordinate (if any) assigned to the point. Any Local Coordinate that has been assigned to a feature can, additionally, be chosen to be ignored as a separate option.
- **By point in selection memory** A Point previously added to **selection memory** is used together with the points defining the master and slave features to define the xy-plane of the local axis direction.
- **By specified local coordinates** The element axes are defined using a previously defined Local Coordinate.

For Joint element assignments only, the order of the features selected determines the Master and Slave. The master assignment of a joint is always the first of the two selected features that the joint is assigned to. For assignment to multiple pairs of features selected, those features in selection memory will become the slave assignments. Joint elements are orientated from the Master to the Slave by default. To reverse the joint orientation to be from Slave to Master, deselect the Mesh from Master to Slave option.

Case Study. Joint/Interface Mesh (2D)

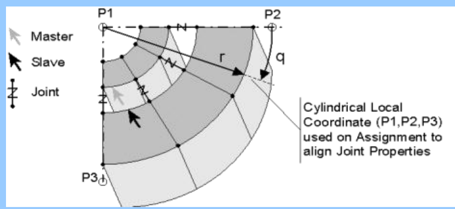
In this example Line 2 is placed in selection memory (Slave) and a Line joint mesh attribute with 6 divisions is assigned to Line 1 (Master). Joints are created automatically to tie the Lines together with an interface joint mesh.



Note. The unmerge facility allows coincident features to be created from a single feature and also allows a feature to be set as Unmergable, so it will not be accidentally merged back with another coincident feature. See **Merging and Unmerging** for more details.

Case Study. Cylindrical Joint/Interface Mesh (3D)

In this example, a Surface joint mesh is assigned to Surfaces between two concentric cylinders. Cylindrical axes are defined for the joint properties using a local coordinate. Joint local x axes will then coincide with the cylinder radial direction.



Joint Material and Geometric Properties

Joint material and geometric properties are assigned to the master feature.

- **Joint Material Properties** Joint meshes require joint properties to be assigned to them. These are defined from the **Attributes**> **Material**> **Joint** menu item.
- **Joint Geometric Properties** For joints with rotational degrees of freedom an eccentricity must be specified using the **Attributes**> **Geometric**> **Joint** menu item.

Non-Structural Mass Elements

Non-structural mass elements are used to define a lumped mass at a Point, or a distributed mass along a Line or over a Surface. Variations may be used to vary the mass along the Line or over the Surface.

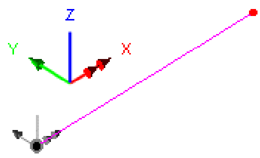
Defining and Assigning Non-Structural Mass Elements

A non-structural mass element is defined as a mesh attribute using the **Attributes**> **Mesh** menu item and specifying the structural element type to be used. Once defined it is assigned to selected features using the standard drag and drop technique.

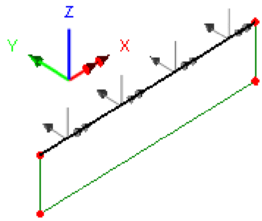
Non-structural mass elements also require material properties to be assigned to them. Use the **Attributes**> **Material** menu item to create the required mass for assignment.

Note. Mass properties should be defined per unit length when assigned to Lines and per unit area when assigned to Surfaces.

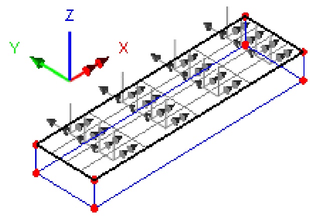
Examples of Assigned Non-Structural Mass Elements



Lumped mass at a point



Distributed mass along a line



Distributed mass over a surface

Non-Structural Mass Local Axis Direction

Since the mass can be used to model hydrodynamic effects it is defined in local directions. In the case of a line, the direction may need to be normal to the line or in the case of a surface, normal to that surface. When carrying out large deformation analyses these directions are continually updated as the solution progresses. The element axis direction can be defined when the mesh is assigned. Three axis orientation options are available:

- **Follow point axes** (Default selection) Adopt the axes assigned from the Local Coordinate (if any) assigned to the point. Any Local Coordinate that has been assigned to a feature can, additionally, be chosen to be ignored as a separate option.
- **By point in selection memory** - A Point previously added to **selection memory** is used to define the xy-plane.
- **By specified local coordinates** The element axes are defined using a previously defined Local Coordinate.

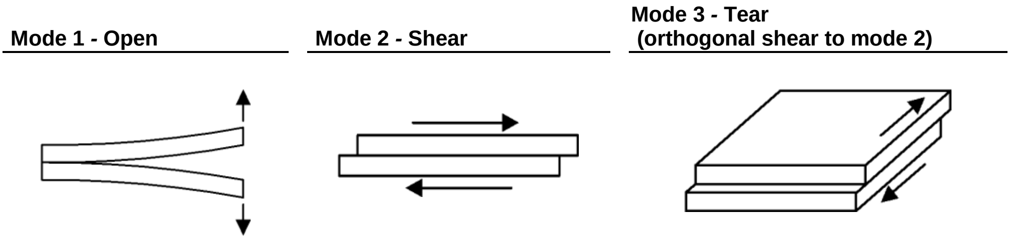
Delamination Interface Elements

Interface elements may be used at planes of potential delamination to model inter laminar failure, and crack initiation and propagation.

If the strength exceeds the strength threshold value in the opening or shearing directions the material properties of the interface element are reduced linearly as defined by the material parameters and complete failure is assumed to have occurred when the fracture energy is exceeded. No initial crack is inserted so the interface elements can be placed in the model at potential delamination areas where they lie dormant until failure occurs.

Fracture Modes

Three fracture modes exist: **open**, **shear**, and **tear** (orthogonal shear for 3D models). The number of fracture modes corresponds to the number of dimensions of the model. The diagram below illustrates the three modes.

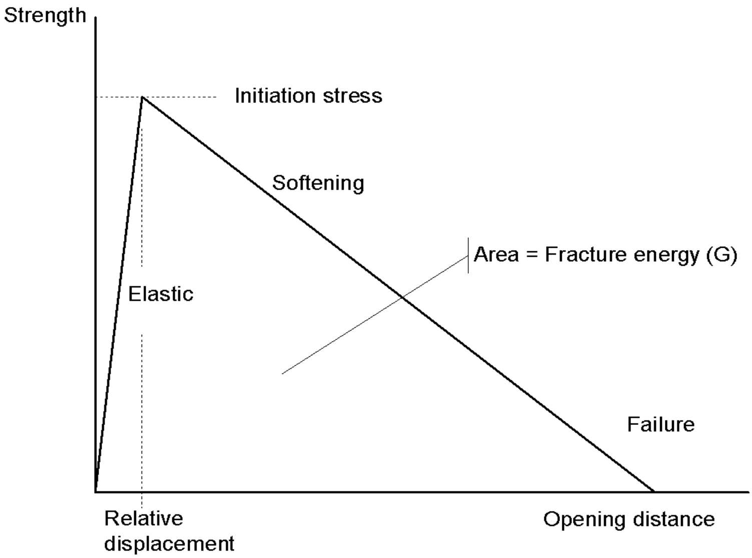


The interface elements are used to model delamination in an incremental nonlinear analysis. These elements have no geometric properties and are assumed to have no thickness.

Interface elements are defined as Line or Surface mesh attributes using the **Attribute> Mesh** menu item.

Interface Material Properties

The interface material properties are defined from the **Attribute> Material> Specialised** menu item and then assigned to the same geometry as the interface mesh.



Material Parameters

- **Fracture energy** Measured values for each fracture mode depending on the material being used, e.g. carbon fibre, glass fibre.
- **Initiation Stress** The tension threshold /interface strength is the stress at which delamination is initiated. This should be a good estimate of the actual delamination tensile strength but, for many problems, the precise value has little effect on the

computed response. If convergence difficulties arise it may be necessary to reduce the threshold values to obtain a solution.

- ❑ **Relative displacement** The maximum relative displacement is used to define the stiffness of the interface before failure. Provided it is sufficiently small to simulate an initially very stiff interface it will have little effect.

Coupling Model

- ❑ **Coupled/mixed interface damage -** Recommended method.
- ❑ **Uncoupled /reversible** - Unloading is reversible along the loading path.
- ❑ **Uncoupled /origin** - Unloading is directly towards the origin ignoring the loading path.

Notes

- It is recommended that automatic nonlinear incrementation is used with the arc length procedure option set to **root with the lowest residual norm**, when defining loadcase control.
- It is recommended that fine integration is selected for the parent elements from the Solution tab of the **Model Properties** dialog.
- The nonlinear convergence criteria should be set to converge on the residual norm.
- Choose **Continue solution if more than one negative pivot occurs** from the Model properties, Solution tab, Nonlinear options dialog and set option 252 to suppress pivot warning messages from the solution process.
- The non symmetric solver is run automatically when mixed mode delamination is specified.
- Although the solution is largely independent of the mesh discretisation, to avoid convergence difficulties it is recommended that a least two elements are placed in the process zone.

Element Selection

About LUSAS Elements

Elements are classified into groups according to their function. The main element groups are listed below.

- See also **Joint Element Meshes**, and **Interface Elements (for composite delamination)**.
- For full details of all elements refer to the ***Element Reference Manual***.

- For full details of the element formulations refer to the *LUSAS Theory Manual*.

Point Element Selection

Non structural mass and Joint elements are defined at or between points.

Elements in plain text are available Standard product versions of LUSAS software. Elements in **bold** text are in Plus versions.

Generic Element Types	2D	3D
Non structural mass	**PM2**	**PM3**
Joint (no rotational stiffness)	JNT3	JNT4
Joint (for beams)	JPH3	JSH4
Joint (for grillages)	JF3	
Joint (for axisymmetric solids)	JAX3	
Joint (for axisymmetric shells)	JXS3	

Line Element Selection

The following table lists the elements available for Line meshing by type and by name. The first column matches the option list in the Line mesh dialog box.

Elements marked with (LT) are in Lite versions of LUSAS software. Elements in plain text are available in Standard product versions. Elements in **bold** text are in Plus versions.

Generic Element Types	2D 2 noded	2D 3 noded	3D 2 noded	3D 3 noded
'None'	-	-	-	-
Bar	BAR2 (LT)	BAR3	BRS2 (LT)	BRS3
Thin beam	-	BM3	-	**BS3, BS4**
Thick beam	BEAM (LT)	-	BMS3 (LT), BTS3	BMI21 (LT), **BMI22, BMX22, BMI31, BMX31, BMI33, BMX33**
Thick beam (nonlinear)	-	-	-	-
Engineering grillage	GRIL (LT)		-	-
Cross-section beam	-	BMX3	-	**BSX4**
Semiloof beam	-	-	-	**BSL4**
Axisymmetric membrane	BXM2	BXM3	JNT4	-

Generic Element Types	2D 2 noded	2D 3 noded	3D 2 noded	3D 3 noded
Joint (no rotational stiffness)	JNT3	-	JSH4	-
Joint (for beams)	JPH3	-	-	-
Joint (for grillages)	JF3	-	-	-
Joint (for axisymmetric solids)	JAX3	-	-	-
Joint (for axisymmetric shells)	JXS3	-	-	-
Thermal bar	BFD2	BFD3	BFS2	BFS3
Axisymmetric thermal membrane	BFX2	BFX3	-	-
Thermal link	LFD2	-	LFS2	-
Interface	-	**IPN6**	-	-
Non structural mass	**LM2**	**LM3**	**LMS3**	**LMS4**

Notes

- Quadratic elements are curved with a mid-side node.
- For some beam elements **rotational freedoms** at the ends of a Line can be made free to rotate by using an element with moment release end conditions.
- When modelling 2D frame problems it is recommended to use the relevant family of 3D beam elements (rather than 2D elements) if, at a later date, it is intended to convert the 2D model into a full 3D model.
- No check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, however LUSAS Solver will stop the analysis if the element is unsuitable.
- This list is a guide as to which elements to use. Not all elements are listed here. See the ***Element Reference Manual*** for full details of all elements.

Surface Element Selection

The following table lists the elements available for surface meshing by type and by name. The first column matches the option list in the Surface mesh dialog box.

Elements in plain text are available in Standard product versions of LUSAS software. Elements in **bold** text are in Plus versions.

Generic Element Types	Triangle 3 noded	Quadrilateral 4 noded	Triangle 6 noded	Quadrilateral 8 noded
Plane stress	TPM3	QPM4M	TPM6	QPM8
Plane strain	TPN3	QPN4M	TPN6	QPN8
Axisymmetric solid	TAX3	QAX4M	TAX6, **TAX6P**	QAX8, **QAX8P**
Thin plate	TF3	QF4	-	-
Thick plate	-	QSC4	TTF6	QTF8
Thin shell	TS3	QSI4	**TSL6**	**QSL8**
Thick shell	TTS3	QTS4	**TTS6**	**QTS8**
Membrane	TSM3	SMI4	-	-
Fourier	**TAX3F**	**QAX4F**	**TAX6F**	**QAX8**
Plane field (thermal)	TFD3	QFD4	TFD6	QFD8
Axisymmetric solid field	TXF3	QXF4	TXF3	QXF8
Explicit dynamic - plane stress	**TPM3E**	**QPM4E**	-	-
Explicit dynamic - plane strain	**TPN3E**	**QPN4E**	-	-
Explicit dynamic - axisymmetric	**TAX3E**	**QAX4E**	-	-
Interface	-	-	-	**IS16**
Non structural mass	**TM3**	**QM4**	**TM6**	**QM8**

Notes

- No check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, however LUSAS Solver will stop the analysis if the element is unsuitable.
- This list is a guide as to which elements to use. Not all elements are listed here. See the ***Element Reference Manual*** for full details of all elements.

Volume Element Selection

The following table lists the elements available for volume meshing by type and by name. The first column matches the option list in the Volume mesh dialog box.

Elements in plain text are available in Standard product versions of LUSAS software. Elements in **bold** text are in Plus versions.

Generic Element Types	Tetrahedral 4 noded	10 noded	Pentahedral 6 noded	12 noded	15 noded
Stress	TH4	**TH10**	PN6	-	**PN15**
Thermal	TF4	**TF10**	PF6	-	**PF15**
Explicit dynamic	**TH4E**	-	**PN6E**	-	-
Stress composite	-	-	**PN6L**	**PN12L**	**PN15L**
Thermal composite	-	-	**PF6C**	**PF12C**	**PF15C**

Generic Element Types	Hexahedral 8 noded	16 noded	20 noded
Stress	HX8M		**HX20**
Thermal	HF8		**HF20**
Explicit Dynamic	**HX8E**		
Stress Composite	**HX8L**	**HX16L**	**HX20L**
Thermal Composite	**HF8C**	**HF16C**	**HF20C**

Notes

- No check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, however LUSAS Solver will stop the analysis if the element is unsuitable.
- This list is a guide as to which elements to use. Not all elements are listed here.

Geometric Properties

Any geometric properties that have not already been inherently defined by the feature geometry need to be specified using geometric property attributes.

Geometric properties are element dependent and are defined for an element family such as bars, beams, shells, joints etc. Note that geometric attributes are not required for plane strain, axisymmetric solid or 3D solid elements. For more details on the properties required for a specific elements refer to the *Element Reference Manual*

Geometric attributes are defined for each feature type using the **Attributes> Geometric** menu item and then **assigned** to the required geometric feature (or to an appropriate **mesh object** in a mesh-only model). Geometric properties can be defined for:

- **Lines**
- **Surfaces**
- **Joints**

Geometric Line Properties

Geometric line section properties such as cross-sectional area, second moments of area etc., for bar/link, grillage, axisymmetric membrane etc., and thin/thick beam elements can be defined either by:

- Using the **Attributes> Geometric> Section Library** menu item to access supplied and user-created sections in the **section library.**
- Using the **Attributes> Geometric> Line** menu item to enter section properties directly on the Geometric Line dialog. If this option is chosen all beam cross-section information will need to be defined if the section was not selected from a standard section library or created by using the arbitrary section property calculator. The arbitrary section property calculator should generally be used to create sections in preference to manually defining geometric line property data since cross-sectional values (used by the fleshing facility) are automatically stored as part of the section property calculation.

Varying sections can be defined by:

- Using the **Attributes> Geometric> Tapered Section** menu item to define a tapering section where the section geometry varies linearly for the length of the assigned line. See Tapering Sections for details
- Using the **Attributes> Geometric> Multiple Varying Section** menu item to create a geometric line attribute that defines completely varying section built from any number of cross-sections that are to be assigned to a line or a whole series or path of lines at specified distances. See **Multiple Varying Sections** for details.

Reinforcement attributes (used in the calculation of particular crack width calculations) can be defined by:

- Using the **Attributes> Geometric> Bar Reinforcement** menu item to access the reinforcement attribute dialog. See **Reinforcement Attributes** for details.

The following options are available when defining geometric line properties:

- **Visualise** Cross-sectional shapes for standard library items and for user sections (stored in local and server libraries that were created by the standard and arbitrary section property calculators) will be automatically visualised. General beam sections defined by using the Attributes > Geometric > Line menu item, will require cross-section properties to be defined manually in order for geometric visualisation (and fleshing) to take place. The orientation of the visualised section is based upon the vertical axis defined for the model. In LUSAS, 2D models are assumed to be drawn in the XY plane with the Y axis vertical, and 3D models with have the Z axis set to be vertical. The local axes of the section are located at its centroid. A circle, of the same pen colour that is defined for Lines, represents the location of the nodal line about which the section will be positioned when the section is assigned to a line on the model. Setting an eccentricity (according to element type) moves the location of the nodal line with respect to the centroid. See **Eccentricities** for more details.

- **Tapering... / Non-Tapering...** Tapering beam sections can be defined by specifying section properties for each end of the beam. For complex sections this would normally be done by drawing selected cross sections for key locations along a model and using the arbitrary section property calculator in Modeller to calculate and save the properties to a library prior to using this dialog. Where both ends of a beam have been defined using either a LUSAS supplied standard library item or one of the LUSAS standard section calculators an 'exact' calculation can be made to arrive at intermediate section properties based upon the known shapes at either end of the beam. As a result, no interpolation options can be specified for this situation. In cases where one or both ends of a beam section have been defined using the arbitrary section calculator (including properties calculated from the precast section range) a choice of interpolation method is provided. When the tapering option is chosen, the vertical and horizontal alignment of one end of the beam section from the other can be specified. Tapered beams would normally use the same section shape at either end, but differing sections can be accommodated. Eccentricities of beam ends from nodal positions can also be defined. When modelling varying cross sections with constant section beam elements care should be taken to ensure that sufficient elements have been assigned. More complex tapering can be achieved using the **Multiple Varying Section** facility.
- **Section details** of section properties, fibre locations and coordinates defining the Cross-sectional geometry of the section can be viewed in tabular format.

  - **Properties** Provides listings of Standard, Additional (used primarily for design checking) and Plastic properties. Plastic geometric properties are required for beams when using the stress resultant material model (model 29) and comprise:

Standard properties

A - Cross-sectional area

Ixx - Second moment of area about x axis

Iyy- Second moment of area about y axis

Izz - Second moment of area about z axis

Ixy, Iyz, Izx - Product moments of area

J - Torsional constant

Asx - Effective shear in x direction

Asy - Effective shear in y direction

Asz - Effective shear in z direction

Additional properties for design checking

kx - Radius of gyration about x axis

ky - Radius of gyration about x axis

kz - Radius of gyration about x axis

Cw - Warping torsional constant

xo - Shear centre, distance from centroid along x axis

yo - Shear centre, distance from centroid along y axis

zo - Shear centre, distance from centroid along z axis

Plastic properties

Ap - Plastic cross-sectional area (= elastic area)

Zpy - Plastic section modulus about y axis

Zpz - Plastic section modulus about z axis

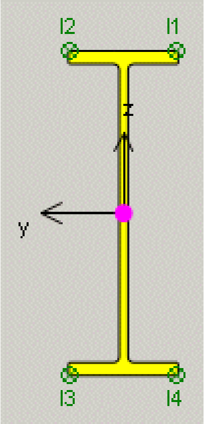
Zpt - Plastic torsional section modulus

Yp - Plastic neutral axis, distance from centroid along y/z axis

Zp - Plastic neutral axis, distance from centroid along y/z axis

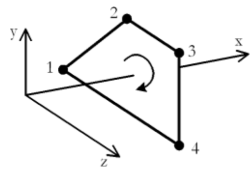
The actual parameters required for an analysis depend on the chosen beam element. See the *Element Reference Manual* for further details.

Eccentricity values can be used to position the top or bottom of a section with respect to a nodal line. The use of fibre location values can assist here too.



- **Fibre locations** drawn as green symbols with an identifying reference define positions on the perimeter of the beam cross-section at which stresses can be plotted when visualising results. Standard sections, precast beam sections, and box sections added to a library will have their cross-sectional geometry pre-defined. They also have default fibre locations stored for each section. Sections drawn by users and added to the user and server libraries using the arbitrary section property calculator have extreme fibre locations calculated automatically - these are denoted by 'A1', 'A2', etc. Additional fibre locations can also be added manually for points on the outer boundary of a section. Fibre location values can also be used for copying and pasting into the eccentricity cell of the main dialog if the top or bottom of a section is to lie on a nodal line, or for use in calculating an eccentricity value to position the section in any other chosen position with respect to a nodal line.

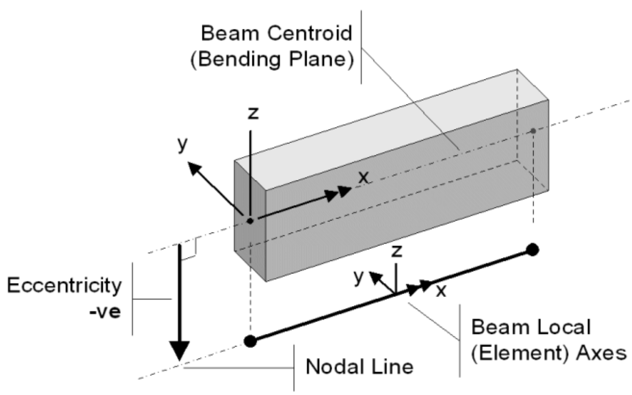
- **Cross-section** information is used to define a series of quadrilateral shapes that represent the cross-sectional shape of the section. These only need to be defined if a beam's properties have been defined manually and it is required that the beam's shape is visualised when using the fleshing option, otherwise cross-sectional information is automatically for sections provided or created by LUSAS section property calculators.



In defining the cross-section shape the coordinate of each quadrilateral must be defined in local zy cross-section coordinate pairs at each node, z1, y1, z2, y2, z3, y3, z4, y4. When defining a cross-section by this method the centroid of the section must reside at 0,0. For Cross Section Beam elements (for advanced use only) the number of integration points (also known as gauss points) can also be set.

Once defined, the geometric section properties are added to the Treeview using the **OK** or **Apply** button. The section is then available for assigning to the appropriate Lines on the model. Assigned beam section properties may be fleshed using the fleshing button or from the Attributes Treeview.

Eccentricities



- Beam elements accommodate Eccentricities which are measured **<u>from</u>** the bending plane **<u>to</u>** the nodal line in the local element direction. Beam section properties are always input relative to the beam axis.
- When sections are defined at either ends of a tapering beam the eccentricity of one section to the other to achieve the vertical and horizontal alignment setting specified is automatically calculated. Subsequent entering of an eccentricity value for the 'master' end in the Value field will automatically position and update the value for the other 'follower' end by an equivalent amount to ensure the beam ends are moved equally from their nodal positions. If a vertical or horizontal alignment

eccentricity is stated in the Alignment panel of the dialog that value will only affect the 'follower' end.

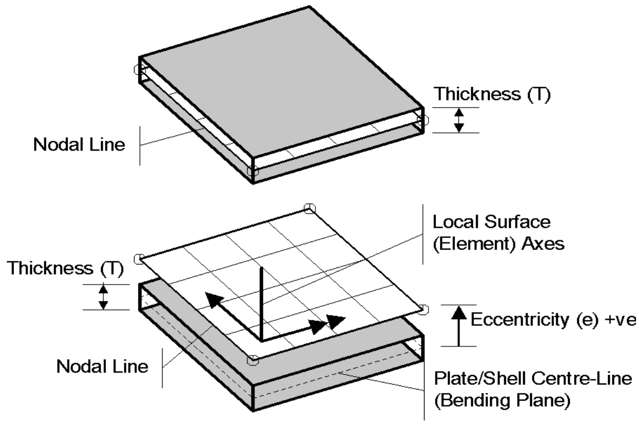
Notes

- The orientation of the visualised section is based upon the **vertical axis** that is defined for the model. In LUSAS, 2D models are assumed to be drawn in the XY plane with the Y axis vertical. 3D models normally have the Z axis set to be vertical.
- For tapering sections defined using the same standard section (rectangular, rectangular hollow section, circular, circular hollow section etc) at each beam end the standard LUSAS section property calculator is used to accurately calculate all section property values.
- For tapering sections defined using arbitrary sections of the same section shape at each beam end the standard LUSAS section property calculator is used to calculate the values of A, Iyy, Izz and Iyz. By default the enhanced interpolation method is used to calculate the values of J, Asy and Asz, but the linear interpolation method is also available. The Enhanced interpolation method has been proven to generally produce more accurate section property values than the Linear method.
- For tapering sections defined using arbitrary sections with different section shapes at each beam end, by default the Enhanced method is used to calculate all values but the linear interpolation method is also available.
- The visualisation of tapering sections on the Geometric Line dialog is for information only. Only by selecting the Visualise button will a correct representation of the relative arrangement of both sections be seen, incorporating any alignment options specified on the main dialog.
- When modelling varying cross sections with constant section beam elements care should be taken to ensure that sufficient elements have been assigned. Greater than 8 elements should be used for small variations in cross section along the length of the geometry to which the section has been assigned, and considerably more elements should be used for larger variations.
- Models created prior to version 14.2 will not have any fibre locations data stored for each beam. However, the relevant fibre location data can be added automatically for these models by double-clicking on each Geometric line entry in the Treeview and re-selecting the same section size from the appropriate sections library.
- Double-clicking on a geometric line attribute name in the Treeview allows editing of beam section information.

Geometric Surface Properties

Geometric attributes are defined for surfaces using the **Attributes> Geometric> Surface** menu item.

- Structures modelled using plate, membrane or shell elements require a **Thickness** to be defined for each surface.



Eccentricity

- Optionally an **Eccentricity** can be specified for certain element types. Eccentricity is measured **from** the bending plane **to** the nodal line in the local element z direction.

Varying surface properties

Geometric properties can be varied over a surface by using a variation. See *Variations* **for more details.**

Geometric Joint Properties

Geometric attributes are defined for joints using the **Attributes> Geometric> Joint** menu item. For certain joint elements eccentricity (in teh local z direction) is an optional geometric property that can be defined. See the *Element Reference Manual* for details

Setting Geometric Attributes for Default Use

A geometric attribute may be designated as the default assignment using **Set Default** on the context menu. When activated, default attributes are automatically assigned to new geometry as it is defined.

Visualising Geometric Assignments

Geometric property visualisation of beams of standard or arbitrary cross-section (that are held in section libraries), and surface thicknesses can be toggled on and off using fleshing button.

Geometric property attributes can be visualised individually by selecting an attribute name in the Attributes Treeview and clicking the right mouse button to choose **Select Assignments** or **Visualise Assignments** from the context menu. This is the easiest way to interrogate the assignment of a single attribute.

Geometric property dependency across analyses / loadcases

Geometric properties defined and assigned to features for the first loadcase within a **base analysis** can be optionally inherited by all other analyses. Assigning a different geometric property to selected features in a following analysis supercedes any inherited geometric properties but for only those assigned features.

Styles

The Attributes layer properties define the styles by which geometric properties are visualised and also permits visualisation of loadings on an individual basis. The attributes layer properties may be edited directly by double clicking the layer name in the Layers Treeview. Loading can be visualised in three ways:

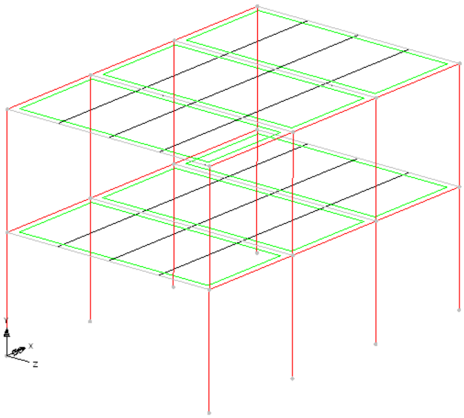
- **By colouring geometry** Colours the geometry according to which attributes are assigned to which features.
- **Visualise cross section** displays any beam section shape exaggerated by a specified %.
- **Show variation of offsets smoothed** displays a series of offset beams as a smoothed line rather than series of steps.
- **Use solid fill for volumes in model** colour fills volumes for visualisation purposes.
- **Cross-section end shrinkage** options of **Automatic** or **Specified distance** permit the beam sections to be shrunk away from the line ends for clarity at connections. Selecting **None** visualises the geometry from point to point with no shrinkage.

See **About Model Attributes** for more details.

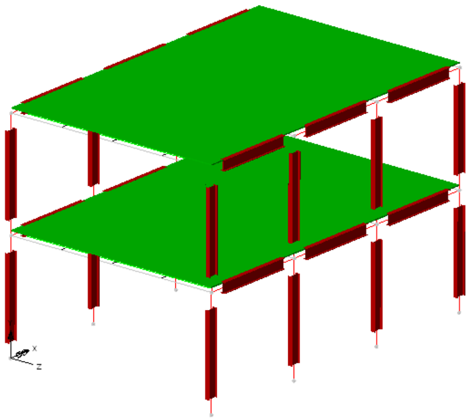
Fleshing

Beams of standard or arbitrary cross-section (that are held in section libraries), and surface thicknesses can be visualised on the geometry model using the fleshing button, or from the Attributes Layer properties Geometric tab.

From the geometric settings dialog or from the context menu of the fleshing button the cross-section may be exaggerated in size and shrunk in from the ends of the assigned Line to aid visualisation at the connections. When processing results the deformed cross-section shape may also be visualised by selecting the **Deform** option on the geometric properties dialog.



Visualisation of Attributes Without Fleshing



Visualisation of Attributes With Fleshing
(and end shrinkage applied)

Reinforcement Attributes

Reinforcement bar attributes are used to model the steel reinforcement in reinforced concrete. When used with the Smoothed Multi Crack Concrete Model (Model 102) and the **Crack Widths calculation utility,** contours and values of crack widths can be plotted in accordance with EN 1992-1-1:2004 Eurocode 2. Crack width contours are plotted along the bar elements (actually on the surface of the fleshed bar section) corresponding to the steel strains used in the calculation,

Reinforcement attributes are similar to geometric line attributes but need to be assigned to lines meshed with bar elements for a chosen analysis to enable a crack width calculation to be carried out.

Bar attributes may be defined for a single reinforcing bar (Discrete bar method), as would be used in a 3D model, or be defined with suitable properties to represent a bundle or simplified arrangement of bars (Equivalent bar method) as would be used typically in a 2D plane strain analysis.

- **Bar area** The cross sectional area of an individual reinforcing bar if discrete bars are being defined, or the equivalent area of a number of reinforcing bars if a bundle or simplified arrangement of bars is being considered.

Parameters for nonlinear concrete crack checking

This check box must be checked to enable crack width checking to take place.

- **Reinforcement cover** - the nominal cover to the reinforcing bar from the concrete face in tension.
- **Bar diameter** - the diameter of a single bar if individual (discrete) reinforcing bars are being considered, or the equivalent diameter if a bundle or simplified arrangement of bars is being considered.

- **Reinforcement type** - may be defined as **Discrete bars** or **Equivalent bars** Choosing Discrete bars requires input of properties for an individual bar. Choosing Equivalent bars requires input of a bar area that represents the bundle or simplified arrangement of bars being considered. For this, the bar diameter specified should always be that of a single bar.
- **Breadth of section/Bar spacing** - If Equivalent bars are being defined the breadth of concrete section (the width of the effective concrete tension area) that the bars are occupying would be stated. If Discrete bars are being defined the centre to centre distance of the bar from its nearest neighbour would be stated.
- **Ratio of bond strength (ξi)** - An adjusted ratio of bond strength that will take into account the different diameters of prestressing and reinforcing steel if both are used. This has a (default) value of 1 when only reinforcement bars and no prestressing tendons are present. Refer to EN 1992-1-1:2004 Table 6.2 for suggested values otherwise.

Notes

- Any number of reinforcement attributes can be created and assigned to lines in the model for different analyses.
- The cross-sectional shape is automatically created from the bar diameter.
- If changing a bar diameter for a Discrete bar the Bar area will also need changing, and vice-versa.
- If changing the bar diameter for Equivalent bars the Bar area will also need changing, and vice-versa.
- If previously defined reinforcement attribute values are changed the model must be re-solved prior to running the Crack Width calculation facility.

Section Library

The section library is available from the **Attributes> Geometric> Section Library** menu item. Standard steel section libraries are currently available for the following:

- **Australia steel sections**
- **Canada steel sections**
- **China steel sections**
- **EU steel sections**
- **KS steel sections (including Korean Rail Sections)**
- **UK steel sections**
- **US steel sections**

The library of precast concrete sections supplied currently includes:

- **UK beams: Y, YE, TY, TYE, SY, M, UM and U beam types**
- **US beams: AASHTO Types II to VI, Florida Bulb T72 and T78 beams, NU Girders and Texas DoT 'I' Beams, North-East 'T' Bulbs.**
- **Australia and New Zealand beams: Super-T beams T1 to T5 (open and closed)**
- **Canada 'I' beams**

In addition, user-created section properties can be saved in the following libraries:

- **User (local)** - for use inside the current project folder only
- **User (server)** - for use across all projects

See **Library Locations** for details about library filenames and locations.

Sections selected from a library are added to the Attributes Treeview. From there they can be assigned to selected line features on a model (or to appropriate elements in a mesh-only model). For more details on the use of section library items see the **Geometric Properties** section.

Adding Additional Sections to the Section Library

In addition to sections provided in the geometric beam section library, other sections can be added to the library by using Section Property Calculator facilities. These are accessed from the **Utilities > Section Property Calculator** menu. Facilities exist to calculate the properties of standard sections, precast beam sections with a top slab, simple and complex box sections, and user-defined arbitrary sections that are created in LUSAS Modeller.

Multiple Varying Sections

The multiple varying section dialog is accessed using the **Attributes > Geometric > Multiple Varying Section** menu item. It enables pre-defined cross-sections to be specified at distances that can be interpolated into a single varying section for subsequent assignment to a single line, or to a series of lines with reference to a pre-defined reference path. A table is built up specifying the section shapes which define the varying section, the interpolation method to be used in order to describe the change of section shape between sections, and the alignment method to be used to set-out each section with respect to another.

Basic usage and section selection

A multiple varying section is defined for use with a particular element type, selected in the Usage part of the dialog. In the section selection table the sections that will be used to generate a varying cross section along a line or path of lines must each be added, and an interpolation method and a distance from a starting point must be specified for each. Note that user-defined sections need to be saved to the local or server libraries prior to using this facility.

- Clicking on the launch dialog button in the **Section** cell allows a pre-defined section to be chosen from a section library.

- Clicking on the drop-list button ▾ in the **Shape Interpolation** cell permits the definition of an interpolation setting which defines the shape between adjacent pairs of defined sections.
- Specifying a **Distance interpretation** dictates how the section spacing values entered are to be interpreted.

More detail is provided in the following sections.

Shape Interpolation

A shape interpolation setting defines how the shape between adjacent pairs of defined sections is calculated. Note that it does not define or calculate the engineering properties for those shapes (for that see **Section Interpolation** below). Shape interpolation options are only available for second and subsequent entries in the table. The following interpolation options are available:

- **Smoothed** is available for use with any number of sections. In practice, smoothed is linear for two, quadratic for three, and a cubic spline fit for more than three. Thus, when the interpolation technique is manually chosen, sections are entered in groups.
- **Linear** can be used with any number of defined sections (if there are more than two, a piecewise linear approach is assumed).
- **Quadratic** can be used for sets of sections, that is for 3 sections, 5 sections, 7 sections etc., where three values can be interpolated (for the first, middle, and last section in a set).
- **Function** is a function based method where the order (n) for the function that is shown on the dialog defines the shape between adjacent pairs of sections.

Shape interpolation settings are only available for second and subsequent entries in the table. If 'Specify interpolation order' is unchecked (off), the ‘shape interpolation’ column in the grid is disabled, and a smoothed (spline) interpolation technique is used to fit all the given sections at their given distances.

Distance interpretation

- ❑ **Scaled to fit each line individually** For this option values entered in the Distance column of the table represent the locations along each line that the sections will be assigned. Values are entered as proportional distances along a line (for example, entering 0, 0.5 and 1 would specify a section at either end and at a mid-point of any line that was selected and assigned this geometric line attribute). Distance values are normalised to the actual line length and are offset by the start distance. For example, a section that varies from defined geometry at start, middle and end can be input with distances as 0, 0.5 and 1.0, or 0, 5, and 10, or 10; 15, and 20. This flexibility can be used to entire a series of definitions along a common parallel chainage.

- **Along reference path** For this option values must be entered in the Distance cells of the table that specify the actual distance at which each section will be positioned along a reference path. Values are entered as absolute and not relative distances. Note that a section does not necessarily have to be defined to start at a distance of 0. A particular reference path can be specified prior to assignment or on assignment.

Examples are provided of distance types. See **Distance Types and Methods of Assignment.**

Table related buttons

- The **Symmetric section** check box copies any rows above the last defined entry and reverses them to create a symmetric odd-numbered arrangement.
- **Edit**, **Insert** and **Delete** buttons provide the means to select sections from the library, create new rows above a selected row, and to delete rows from the table.
- **Flip** mirrors the contents of the table (and the visualisation of the varying section shape).

Alignment

- **Vertical and horizontal alignments** - these govern how tops, centres, bottoms and sides of adjacent beam sections are set out relative to the 'master' section specified in the adjacent 'Alignment' drop-down list. An option to enter user-defined individual eccentricities is also provided and enables more advanced alignment to be achieved.
- **Align all sections to section (number)** - aligns all sections with respect to the specified 'master' section number. See Section eccentricities explained.

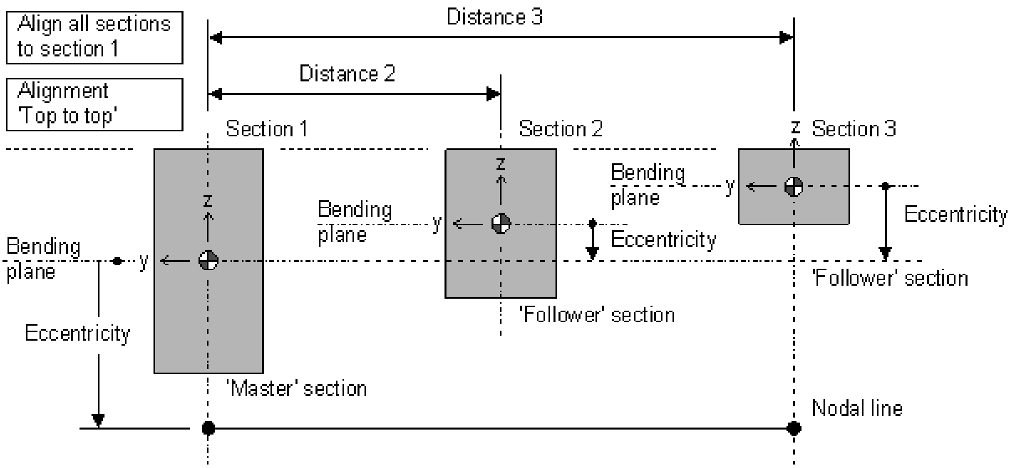
Section eccentricities explained

When sections are specified to define a multiple varying section beam it is important to remember that the 'master' section is the one to which all 'follower' sections are aligned to. The eccentricity of each 'follower' section to achieve the desired vertical and horizontal alignment with respect to the 'master' is automatically calculated for each section. The eccentricity values for 'follower' sections are made up of a value corresponding to the automatically calculated eccentricity required to achieve correct alignment to the master section plus any additional user-defined individual eccentricity that may have been defined for the 'master' section. These values can be seen by clicking the launch dialog button in the Section cell of the table for the section.

Maintaining a consistent 'master' section will make it easier to marry multiple varying section attribute assignments. Subsequent updating of an eccentricity value for a 'master' section will automatically update the eccentricity values for all of the 'follower' sections by an equivalent amount to ensure the sections are moved equally to re-align with the master section. Updating of eccentricity values is only possible by selecting the 'Individual offsets' option on the Vertical or Horizontal alignment drop-down lists, and clicking the launch dialog button in the Section cell of the table for a section, and entering an eccentricity value on the 'Enter section' dialog).

Note that for beam elements an eccentricity is measured **from** the bending plane of the section **to** the nodal line in the local element direction. This can result in both positive and negative eccentricity values depending upon the size of the 'master' and 'follower' sections.

To move a multiple varying section beam up or down from its nodal line position only the eccentricity for the 'master' section need be modified because all other sections will have their eccentricity values updated automatically to be moved by the same amount.



Whilst not commonly used, user-defined individual eccentricity can be entered for selected sections by selecting the **Individual eccentricity** menu item from either the Vertical or Horizontal Alignment drop-down menu, then accessing a section's properties by clicking the launch dialog button in this cell and entering an eccentricity value on the Enter section dialog. When an individual eccentricity is specified on a 'follower' section any connection with the 'master' section is broken and any eccentricity specified for a 'master' section will no longer update the offsets on a 'follower' section.

Interpolation of section properties

Where all sections have been defined using either a LUSAS supplied standard library item or one of the LUSAS standard section generators an 'exact' calculation is made to arrive at intermediate section properties based upon the defined shape (see **Shape Interpolation** above) between the sections.

If the shape of the cross-section cannot be interpolated (because one of the sections has been defined using the arbitrary section calculator, the precast library, or if sections are of completely different shapes) the engineering properties at locations along the multiple varying section can be calculated in the following ways:

- **Enhanced** interpolation uses proprietary LUSAS equations to calculate best-estimate cross-section properties for locations along a beam from the cross-sectional area (A) and Moment of Inertia (I) values of the sections defined at each beam end. See the *Theory Manual* for more details.

- **Linear** interpolation calculates cross-section properties for locations along the beam by linearly interpolating the cross-sectional area (A) and Moment of Inertia (I) values of the sections defined at each beam end. This method has generally known **limitations** for particular section types.

When modelling varying sections with constant beam elements (such as BMS elements) care should be taken to ensure that sufficient elements have been assigned to the line(s) representing the tapering beam to achieve sufficient accuracy of results. If varying section beam elements are used (the BMI group of elements) only one element per line may be required.

Attribute name

The full name of the geometric line attribute added to the Treeview will include the Attribute name followed by an automatically created name based upon the number of section library items used. The automatically created part of the name is uneditable.

Once defined, the geometric section properties are added to the Treeview using the **OK** or **Apply** button. The section is then available for assigning to the appropriate lines on the model (or elements in a mesh-only model). Assigned beam section properties may be fleshed using the fleshing button or from the Attributes Treeview.

Section visualisation

As the multiple varying section is built-up in the table, a visualisation of the longitudinal and vertical alignment and of the cross-section shapes used is displayed on the dialog. Note that longitudinal section visualisation only takes place once all required data for a row has been entered, and then only for sections that are compatible. The visualisation can be inspected by zooming and panning in the display panel. A changing cursor image indicates whether the facility is enabled or not. If necessary click in the panel to activate this facility. Use the mouse wheel to zoom in and out. Click and hold-down the left mouse button, or click and hold-down the mouse wheel to pan.

Multiple varying sections would normally be defined to use the same section shape having the same set of fibre definitions throughout, but differing sections can be accommodated. In situations where the varying sections are too different to be connected together section visualisation on the dialog and fleshing of any assigned attributes on the model is not possible.

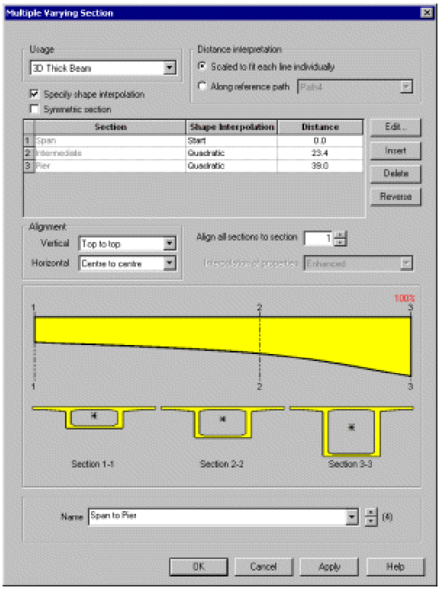
Incompatible section types

Checking for incompatible section types can only be carried-out when the OK button is pressed.

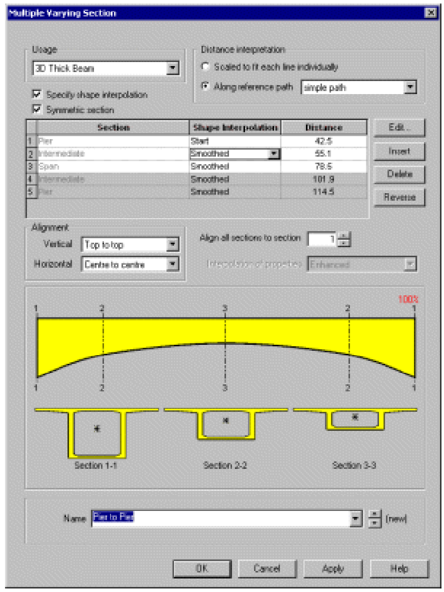
Multiple Varying Section Distance Types and Methods of Assignment

A multiple varying section geometric line attribute can be defined for assignment to either:

- **A single line** - where section spacing distances are scaled to fit each line individually
- **A series of lines** - where section spacing distances are defined for use with a pre-defined reference path



Varying section distances defined for assignment of the line attribute to a single line.

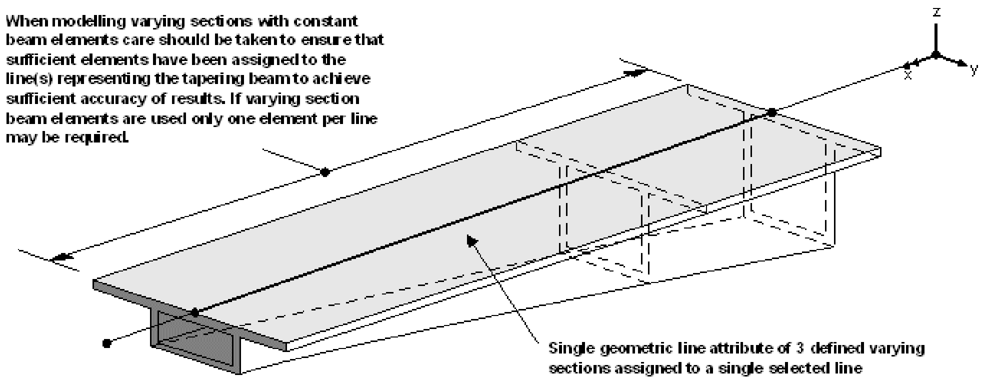


Varying section distances defined for assignment to multiple lines with reference to a path.

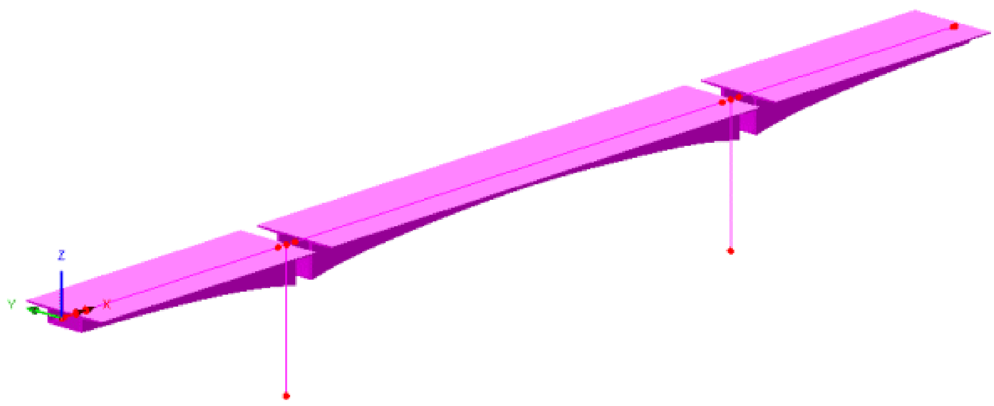
The values entered in the distance cell of the multiple varying section dialog depend upon the intended assignment. Examples of each type follow.

Scaled to fit each line individually

Use of the Scaled to fit each line individually option enables a simple prototyping and assessment model for a bridge formed of tapered box sections to be created in a very straightforward manner. In their simplest form, multiple varying sections can be defined for assigning to single selected lines on a model (or appropriate elements on a mesh-only model). Distance values are normalised to the actual line length and are offset by the start distance. For example, a section that varies from defined geometry at start, middle and end can be input with distances as 0, 0.5 and 1.0, or 0, 5, and 10, or 10; 15, and 20. This flexibility can be used to entire a series of definitions along a common parallel chainage.



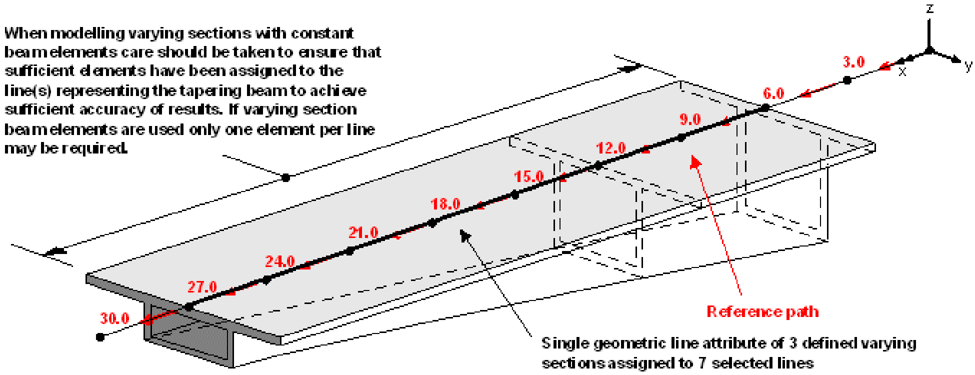
Single line beam assigned a single multiple varying section line attribute (for clarity beam line has been visualised at top of section)



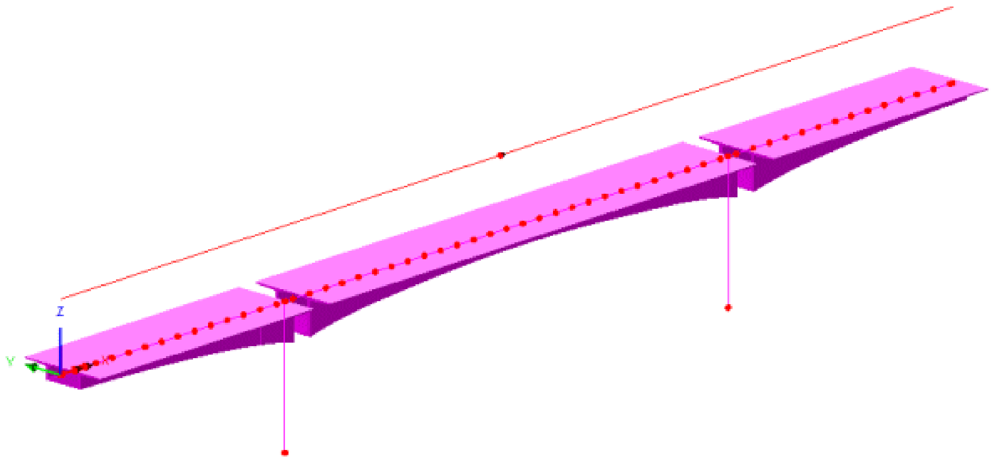
Three varying section line attributes assigned to three separate lines of a model

Along reference path

Use of the multiple varying section with the reference path facility enables beam models of bridges formed of tapered box sections to be created for use in a staged construction analysis. With this, multiple varying sections are assigned to multiple lines. Distance values are entered that define the distances at which a defined section is to be positioned along a set of selected lines with reference to a pre-defined **reference path**. Note that a section does not necessarily have to be defined to start at a distance of 0. A path can be specified to start at a particular distance. This method has uses in staged construction analyses where, as lines are activated, the appropriate elements and geometric properties for those activated features are added to the model.



Multiple line beams assigned a single multiple varying section line attribute with reference to a path (for clarity beam line has been visualised at top of section)



Three varying section line attributes assigned to multiple sets of lines of a model with the use of a reference path

If necessary, one geometric multiple varying section line attribute could be defined for a series of multiple varying sections at specified distances that define the complete end-to-end run of cross-sections for a bridge. Subsequent assignment of solid or diaphragm sections at supports or mid span would then override any previously assigned 'temporary' assignments of voided sections that may have been previously made.

Line mesh density

When modelling varying sections with constant beam elements (such as BMS elements) care should be taken to ensure that sufficient elements have been assigned to the line(s) representing the tapering beam to achieve sufficient accuracy of results. If varying section beam elements are used (the BMI group of elements) only one element per line may be required.

When defined section distances do not map exactly to lines on a model

In defining distances at which sections will apply and then assigning geometric line attributes containing those sections to lines on a model (with reference to a path) there may be occasions where the set of defined sections are too short for the assigned line (or lines) or too long.

- If a set of defined sections are too short for the assigned line (or lines) the geometric line attribute will stay assigned to all line (or lines) selected but no visualisation (fleshing) will take place on any line in a set of lines that does not have a complete line attribute assigned. Any attempt to solve a model containing such assignments will also produce tabulation errors.
- If a set of defined sections are too long for the assigned line (or lines) fleshing of the section shape will take place but only for the length of line (or lines) that was selected and no tabulation errors will occur as a result when solving the model.

Material Properties

Material attributes are defined from the **Attributes> Materials** menu item and then **assigned** to the required geometric feature (or to **mesh objects** in a mesh-only model). Every part of a finite element model must be assigned a material property attribute. Note that not all elements accept all material property types. Refer to the *Element Reference Manual* for full details of valid element/material combinations.

Linear and Nonlinear Material Properties

- **Isotropic/Orthotropic** Defines **linear** elastic or **nonlinear** material properties with options for plasticity, hardening, creep, damage (continuum or composite, viscosity and two-phase materials.
- **Anisotropic** Different material properties are specified in arbitrary (non-orthotropic) directions by direct specification of the modulus matrix. See *LUSAS Theory Manual 2* for details.
- **Joint** Linear and nonlinear joint material models for contact and impact analyses using joint elements.

- **Rigidities** Allows direct specification of the material rigidity matrix. In-plane and bending rigidities are defined from prior explicit integration through the element thickness.
- **Mass** material models for specifying mass in structure using non-structural mass elements.

Specialised Material Properties

- **Delamination** models for use with the composite delamination interface elements.
- **Rubber** Defines materials with hyper-elastic or rubber-like mechanical behaviour.
- **Crushing** A volumetric crushing model such as would be used for crushable foam-filled composite structures.
- **Modified Cam-Clay** A variant of the classic Cam-clay model used for the modelling of clays. It is capable of modelling the strength and deformation trends of clay realistically.
- **Elasto Plastic Interface** Defines a material to represent the friction-contact relationship within planes of weakness between two discrete bodies.
- **Concrete Creep Models** Define material to predict the mean behaviour of a concrete section due to creep effects. Models include CEB-FIP (1990), Chinese creep code for dams, and Eurocode 2
- **Concrete Creep CEB-FIP (1990)** Defines a material to predict the mean behaviour of a concrete section due to creep effects.
- **Concrete Creep Chinese Code** Defines a material to predict the mean behaviour of a concrete section due to creep effects.
- **Concrete Creep Eurocode 2** Defines a material to predict the mean behaviour of a concrete section due to creep effects
- **Generic Polymer** Defines a model that consists of a set of Maxwell dampers which are used to model visco-elasticity, an Eyring dashpot which is used to model viscoplasticity and a linear spring.
- **Generic Polymer with Damage** Accounts for strain rate behaviour and irrecoverable damage in the modelling of polymers and other polymer-like materials.
- **Multi Linear Bar** for no-tension (compression-only) or no-compression (tension only) material modelling for bar elements.
- **Resultant User** A user defined constitutive model defining the stress resultants from strain.
- **Nonlinear User** A user defined constitutive model defining the stress from strain.
- **Thermal User** Applicable to thermal (field) elements only. Whenever thermal elements have been used in a model thermal material properties should be defined and assigned to the relevant parts of the model.

Editing of Material Properties

Editing of pre-defined material data (such as that provided in the material library) allows users to view both the original material definition input data, as well as modify the values used. Editing of user-defined material properties only permits viewing and editing of the values used.

Material properties added to the Attributes Treeview have context menu entries named Edit Definition and Edit Attribute.

- Selecting the **Edit Definition...** menu entry or double clicking the attribute displays the original definition dialog with all the original input data intact.
- Selecting the **Edit Attribute...**menu entry displays values that can be modified. For materials added from the material library, these values may be changed but this breaks the link to the original definition dialog and a warning message will be displayed.

Notes

- Material property attributes can be formed into a composite lay-up using the **composite** attribute facility.
- Once assigned to geometry, material directions can be **visualised** using the Attributes layer in the Layers Treeview
- Rubber, crushing, and plastic material attributes cannot be combined.

Material dependency across analyses / loadcases

Materials defined and assigned to features for the first loadcase within a **base analysis** can be optionally inherited by all other analyses. Assigning a different material to selected features in a following analysis supercedes any inherited material but for only those assigned features.

Material Library

The more commonly used structural material properties are defined in the material library which is located under the **Attributes> Material> Material Library** menu item.

The units will default to those chosen on model startup but may be changed if desired. Pick the material required from the drop down list and click **OK** or **Apply** to add the material properties to the Treeview. The material properties may then be assigned to the model.

Composite Library

The more commonly used structural composite material properties are defined in the composite library which is located under the **Attributes> Material> Composite Library** menu item.

The units will default to those chosen on model startup but may be changed if desired. Pick the composite material required from the drop down list and click **OK** or **Apply** to add the properties to the Treeview. The composite material properties may then be used to define a composite stack or be assigned to the model in the usual way.

Isotropic/Orthotropic Material

Isotropic and **orthotropic** material attributes can be defined using the **Attributes > Material** menu item. The following material properties can be specified.

- **Elastic** is used to specify linear elastic material properties including Young's modulus, Poisson's ratio, mass density. Orthotropic material orientation is specified as global, relative to a local coordinate system or relative to the feature local axis system. Optional thermal expansion and dynamic constants can be specified. Note that not all elements accept all the orthotropic models. Refer to the *Element Reference Manual* for full details of valid element/material combinations. Orthotropic models are **Plane stress**, **Plane strain**, **Thick**, **Axisymmetric**, **Solid**.
- **Thermal** is used to specify properties for general thermal and heat of hydration analysis. For general thermal analysis phase change state, thermal conductivity, specific heat coefficient, and enthalpy values can be set. For **concrete heat of hydration** analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, additional thermal options such as exotherm type and cement type can be specified. See **Concrete Heat of Hydration** for details.
- **Plastic** Used to model ductile yielding of nonlinear elasto-plastic materials such as metals, concrete, soils/rocks/sand.
- **Hardening** Used to model a nonlinear hardening curve data. Hardening is defined as part of the plastic properties. Isotropic, Kinematic and Granular sub-types are available. Isotropic hardening can be input in three ways.
- **Creep** Used to model the inelastic behaviour that occurs when the relationship between stress and strain is time dependent.
- **Damage** Used to model the initiation and growth of cavities and micro-cracks.
- **Shrinkage** Used to define the shrinkage properties of a material as a piecewise linear curve.
- **Viscosity** Used to model viscoelastic behaviour. Coupling of the viscoelastic with nonlinear elasto-plastic materials enables hysteresis effects to be modelled.
- **Two-phase** Required when performing an analysis in which two-phase elements are used to define the drained and undrained state for soil.

Plastic Material Models *- Isotropic*

The following are Isotropic models available from the **Attributes> Material> Isotropic** menu item by choosing the **Plastic** check box on the material attribute dialog.

- **Stress Resultant** (model 29) Applicable to certain beam and shell elements
- **Tresca** (model 61) Represents ductile behaviour of materials which exhibit little volumetric strain (for example, metals and undrained clays). Incorporates isotropic hardening.

- **Stress Potential** general nonlinear properties based on von-Mises or modified von-Mises criterion including hardening.
- **Optimised implicit von Mises** (model 75) Represents ductile behaviour of materials which exhibit little volumetric strain (for example, metals). Especially for explicit dynamics.
- **Modified Mohr Coulomb** This model applies a tensile and/or a compressive cut-off to the standard Mohr-Coulomb model. In tension this prevents unrealistic tensile stresses developing in materials such as soil or rock. In compression the cut-off results in irreversible deformations once the maximum compressive stress is exceeded. The cut-off can be applied to either a single principal stress component or to the mean pressure.
- **Mohr-Coulomb** (model 65) with non-associative flow. Represents materials which exhibit volumetric plastic strain but no volumetric strain during shear. (for example, granular materials such as rock and soils). Incorporates isotropic hardening.
- **Drucker-Prager** (model 64) Represents ductile behaviour of materials which exhibit volumetric plastic strain (for example, granular materials such as concrete and soils). Incorporates isotropic hardening.
- **Concrete** (Multi Crack (model 94) and Smoothed Multi Crack (model 102) A pair of two and three-d imensional concrete material models that model both cracking and crushing behaviour.

Plastic Material Models *- Orthotropic*

- **Stress Potential** **(Hill and Hoffman models)** These models are available from the **Attributes> Material> Orthotropic** menu item by choosing the **Plastic** check box on the material attribute dialog.

The stress potential model defines nonlinear material properties applicable to a general multi-axial stress state requiring the specification of yield stresses in each direction of the stress space. Incorporates hardening, yield stress and Heat fraction. **Hoffman** is a pressure dependent material model allowing for different properties in tension and compression.

Concrete Heat of Hydration

Concrete heat of hydration properties can only be defined for a Thermal Analysis. Thermal material properties can be entered by using the **Attributes> Material> Isotropic** or **Attributes> Material> Orthotropic** menu items.

See **Isotropic/Orthotropic Material** for general details.

The following heat of hydration parameters can be defined:

- **Cement type** Parameters defining the chemical composition of cement types I, II, III and V may be chosen, or the chemical composition of any cement can be defined explicitly from the *User* option in the *Cement type* drop down list. Further information on this load type can be found in the*Theory Manual.*

Modelling units to be used

Although any set of units can be used for weights and volumes, the temperatures used in the analysis are assumed to be, and must be defined in, degrees Celsius.

For all cement types the following data, using the model units, must be defined:

- ❑ **Weight of cement per unit volume**
- ❑ **Water/Cementitious ratio** =(weight per unit volume of cement and any slag and any fly ash)/(weight per unit volume of water)
- ❑ **Weight of slag per unit volume** (Ground granulated blast furnace slag)
- ❑ **Weight of fly ash per unit volume** (Also known as flue ash)
- ❑ **CaO content of fly ash (%)** This must be specified if fly ash is to be included. Some typical values are Class C fly ash (24%) and Class F fly ash (11%).
- ❑ **Assumed cure temperature** Temperature at which the concrete is assumed to cure (must be in degrees Celcius)

If the **User** cement type option is selected then the following also need to be defined:

❑ **C3S content**	C3S content (%) of cement
❑ **C2S content**	C2S content (%) of cement
❑ **C3A content**	C3A content (%) of cement
❑ **C4AF content**	C4AF content (%) of cement
❑ **Free CaO content**	Free CaO content (%) of cement
❑ **SO3 content**	SO3 content (%) of cement
❑ **MgO content**	MgO content (%) of cement
❑ **Blaine value**	Sometimes referred to as fineness of cement (surface area/unit weight)

Timescale units used for heat of hydration analysis are those specified for the model when first defined, or as specified on the Model properties dialog.

See the worked example Staged Construction of a Concrete Dam in the *Application Examples Manual (Bridge, Civil&Structural* for a typical use of this facility.

Rigidity

The linear rigidity model is used to define the in-plane and bending rigidities from prior explicit integration through the element thickness.

Notes

- Angle of orthotropy is relative to the reference axis (degrees).

- The element reference axes may be local or global (see *Local Axes* in the *Element Reference Manual* for the proposed element type). If the angle of orthotropy is set to zero, the anisotropy coincides with the reference axes.

See the *Solver Reference Manual* for further details.

Mass

Mass material models are used in conjunction with non-structural mass elements to define mass in a structure. Mass Properties are input for element nodes in the element local translational (x, y or z) directions or relative to the local coordinate system assigned to the feature.

See the *Element Reference Manual* for further information.

Thermal Material Properties

Thermal material properties are used to define the thermal behaviour of a material when using Thermal (Field) elements. The thermal properties describe the way in which heat flows. Heat may be transferred through conduction, convection or radiation. For linear steady state heat transfer problems only the conductivity needs to be specified.

For materials in which the conductivity is constant in all directions isotropic material input should be used. When the conductivity varies in different directions orthotropic material input should be used. The direction of orthotropy is defined relative to any local coordinate systems.

For transient thermal analysis the specific heat capacity is also required. It should be noted that within LUSAS a specific heat coefficient is used. The specific heat coefficient is computed by multiplying the specific heat capacity by the density.

If phase change is to be modelled the enthalpy must be specified. Two phase changes models are available . When carrying out a phase change it is recommended that lumped specific heat (OPTION 105) is used. This is specified in the **Model Properties> Solution> Element Options** dialog by choosing the lumped mass option.

For heat of hydration analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, additional thermal options such as exotherm type, cement type and timescale units can be set.

LUSAS Solver can model temperature dependent properties but this needs to be defined in the Solver datafile. See the *Solver Reference Manual* for further details.

Stress Resultant (Model 29)

The Stress Resultant model is accessed using the **Attributes> Material Isotropic**menu item.

The model is formulated directly with the beam or shell stress resultants plus geometric properties, therefore it is computationally cheaper. Consult the Element Reference Manual the check which elements are valid for this material model.

Material Parameters

- **Yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.
- **Section shape** Match the section type to the element being used.

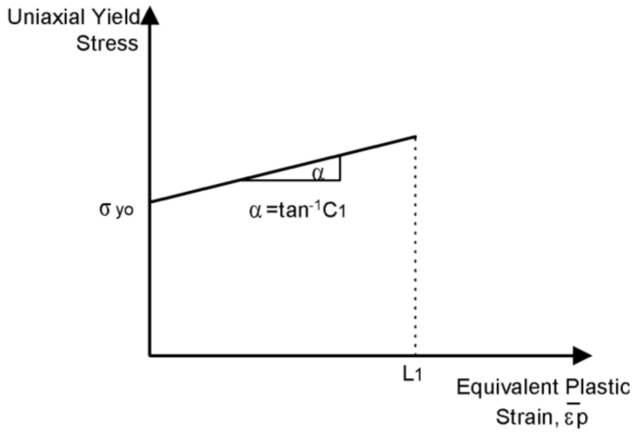
See the *Solver Reference Manual* for further details.

Tresca (Model 61)

The Tresca model is accessed using the **Attributes> Material> Isotropic**menu item.

Material Parameters

- **Yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.
- **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.
- **Slope of Yield Stress** The slope of the uniaxial yield stress against equivalent plastic strain.
- **Plastic strain** The limit of equivalent plastic strain up to which the hardening curve is valid.



Hardening Curve Definition for the Tresca Yield Model

Note

The Modified Mohr-Coulomb model can also model the Tresca yield surface as a special case and provides an improved formulation with better convergence characteristics. It is applicable to solid, axisymmetric and plane strain applications.

Stress Potential (von-Mises, Hill, Hoffman)

The Stress Potential model is accessed using the **Attributes> Material> Isotropic**menu item.

The use of nonlinear material properties applicable to a general multi-axial stress state requires the specification of yield stresses in each direction of the stress space when defining the yield surface (see the *LUSAS Theory Manual*).

Notes

- The yield surface must be defined in full, irrespective of the type of analysis undertaken. This means that none of the stresses defining the yield surface can be set to zero. For example, in a plane stress analysis, the out of plane direct stress, σ**zz**, must be given a value which physically represents the model to be analysed.
- The stresses defining the yield surface in both tension and compression for the Hoffman potential must be positive.

Material Properties

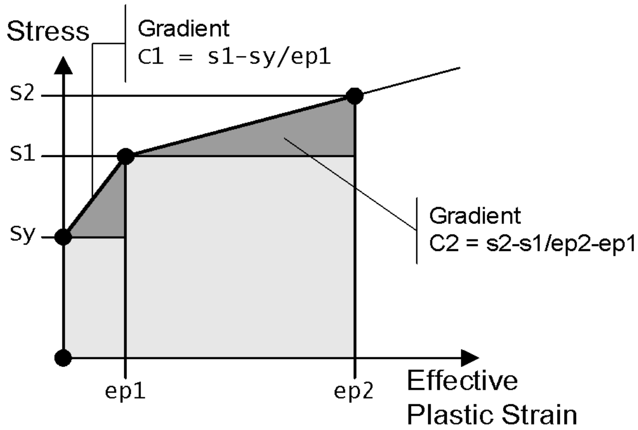
- **Yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.
- **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.

Hardening Properties

There are three methods for defining nonlinear hardening. Hardening curves can be defined in terms of either the hardening gradient, the plastic strain or the total strain as follows:

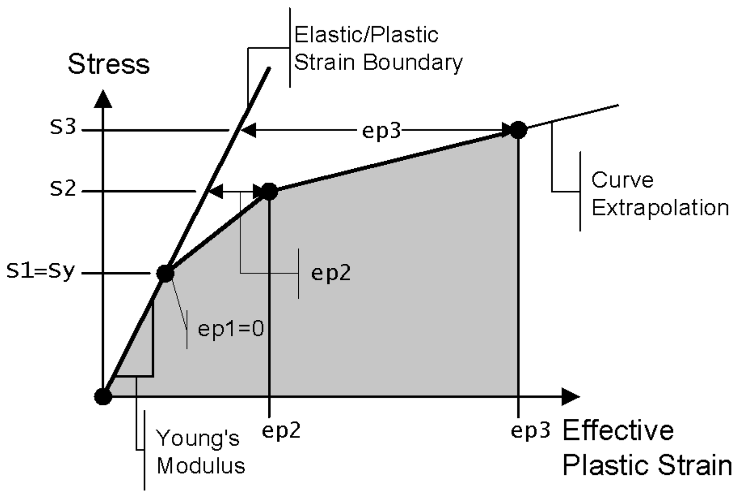
- **Hardening gradient vs. Effective plastic strain** Requires specification of gradient and limiting strain values for successive straight line approximations to the stress vs. effective plastic strain curve.

  In this case hardening gradient data will be input as (C1, ep1), (C2, ep2) for each straight line segment. LUSAS extrapolates the curve past the last specified point.



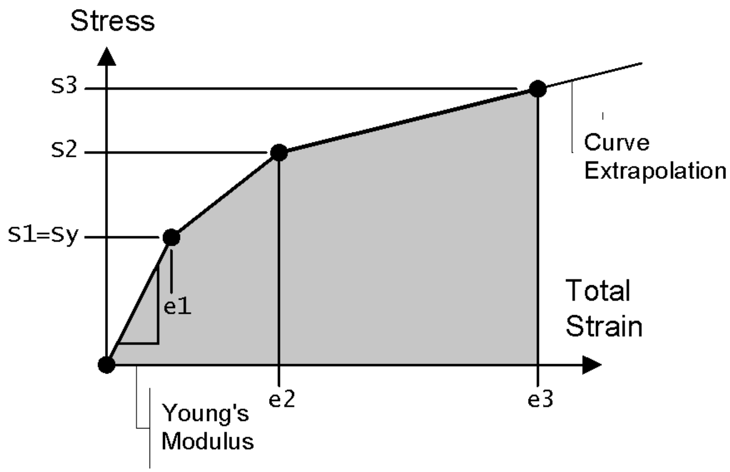
- **Uniaxial yield stress vs. Effective plastic strain** Requires input of coordinates at the ends of straight line approximations to the uniaxial yield stress vs. effective plastic strain curve.

  For the curve shown here the plastic properties will contain the yield stress (sy) and the hardening data will be input as (s1, ep1), (s2, ep2), etc. LUSAS extrapolates the curve past the last specified point.



- **Uniaxial yield stress vs. Total Strain** Requires input of coordinates at the ends of straight line approximations to the stress strain curve.

  Linear properties specify the slope of the stress strain curve up to yield in terms of a Young's modulus. Plastic properties specify the yield stress (sy) and the hardening data is input as a series of coordinates, for example (s1, e1), (s2, e2), etc. LUSAS extrapolates the curve past the last specified point.



Optimised von Mises (Model 75)

The Optimised von Mises model is accessed using the **Attributes> Material> Isotropic**menu item.

This model represents ductile behaviour of materials that exhibit little volumetric strain (for example, metals). It is especially suitable for explicit dynamics.

Material Parameters

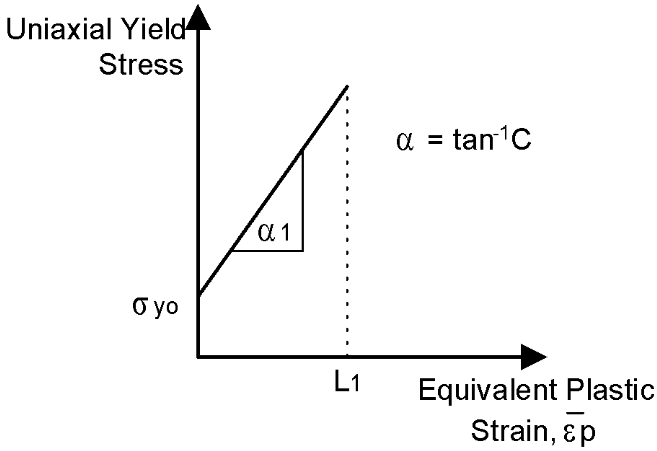
- **Yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.
- **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.

Hardening (von Mises)

- **Kinematic hardening** Plasticity hardening formulation associated with translation, as opposed to expansion, of the yield surface.

In the optimised implicit model the direction of plastic flow is evaluated from the stress return path. The implicit method allows the proper definition of a tangent stiffness matrix which maintains the quadratic convergence of the Newton-Raphson iteration scheme otherwise lost with the explicit method. This allows larger load steps to be taken with faster convergence. For most applications, the implicit method should be preferred to the explicit method.

The model incorporates linear isotropic and kinematic hardening.



Nonlinear Hardening Curve for the von Mises Yield Model (Model 75)

See the *Solver Reference Manual* and the *Theory Manual* for further details.

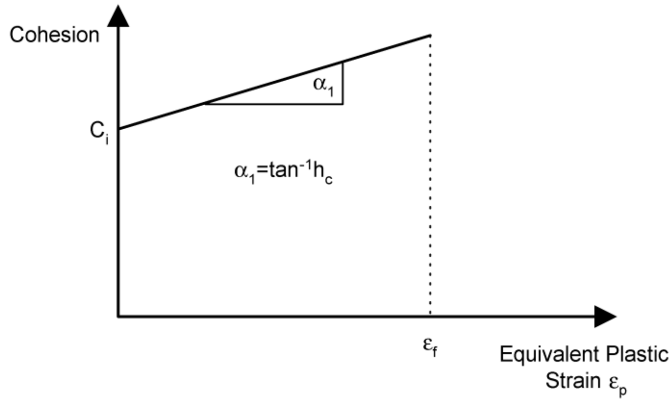
Non Associated Mohr-Coulomb (Model 65)

The Non Associated Mohr-Coulomb elasto-plastic model is accessed using the **Attributes> Material> Isotropic**menu item.

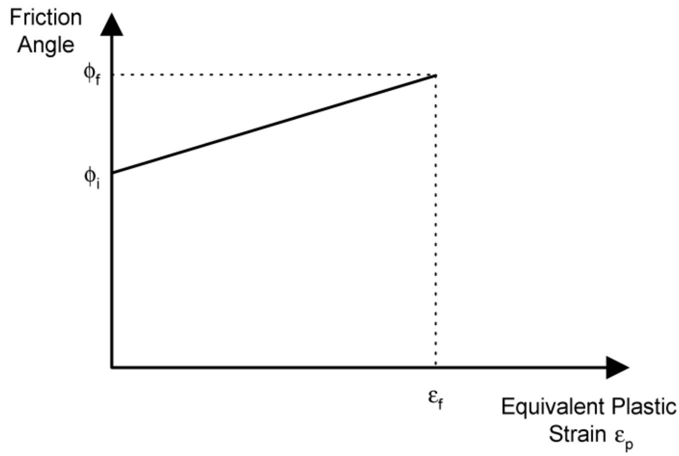
This model may be used to represent dilatant frictional materials that exhibit increasing shear strength with increasing confining stress (for example, granular soils or rocks). The model incorporates isotropic hardening and dilatancy.

Material Properties

- **Initial Cohesion** defining the degree of granular bond and a measure of the shear strength.
- **Friction Angle** defining angle of shearing resistance
- **Dilation Angle** defining magnitude of plastic volume strains.



Cohesion Definition for the Non-Associated Mohr-Coulomb Model (Model 65)



Friction Angle Definition for the Non-Associated Mohr-Coulomb Model (Model 65)

See the *Solver Reference Manual* for further details.

Modified Mohr Coulomb Model

The Modified Mohr-Coulomb model is accessed using the **Attributes> Material> Isotropic** menu item.

This model applies a tensile and/or a compressive cut-off to the Non Associated Mohr-Coulomb model. When the standard Mohr-Coulomb model would potentially predict tensile stresses beyond the uniaxial tensile limit of the material, the tension cap may be used to prevent such stresses occurring.. In compression the cut-off results in irreversible deformations once the maximum compressive stress is exceeded. The cut-off can be applied to either a single principal stress component or to the mean pressure.

The Tresca model is recovered as a special case of the Modified Mohr-Coulomb model by setting both the angles of friction and dilation to 0. The cohesion value C is equal to half the uniaxial yield stress.

Notes

- The model may be used with 3D continuum, plane strain and axi-symmetric elements as well as the corresponding two-phase elements.
- If neither of the Rankine or Pressure options is selected the model will behave as the standard Mohr-Coulomb model.
- The cohesion, tensile and compressive stresses may be specified to be functions of the effective plastic strain. Their variations with effective plastic strain are input as a series of piecewise linear segments.
- The dilation angle must be set to the angle of friction to model associative flow.
- The use of non-symmetric solver is recommended if the dilation angle is not equal to the angle of friction, or if strain hardening is defined.
- The effects of strain hardening on the yield surfaces using a pressure cut-off are shown diagramatically in the Solver Reference Manual.

See the *Solver Reference Manual* and the*Theory Manual* for further details.

Drucker-Prager (Model 64)

The Drucker - Prager model is accessed using the **Attributes> Material> Isotropic**menu item.

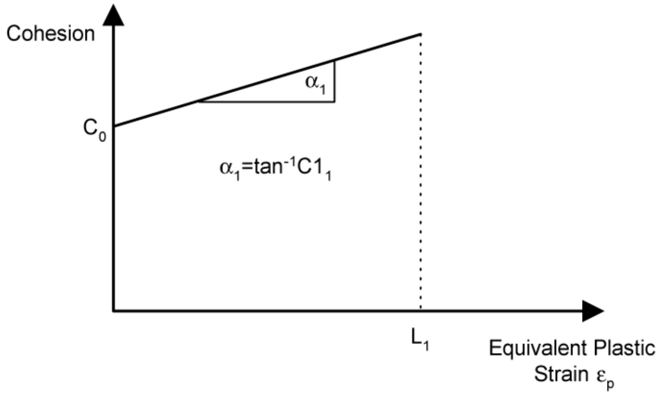
This model may be used to represent the ductile behaviour of materials which exhibit volumetric plastic strain (for example, granular materials such as concrete, rock and soils). The model incorporates isotropic hardening.

Material Properties

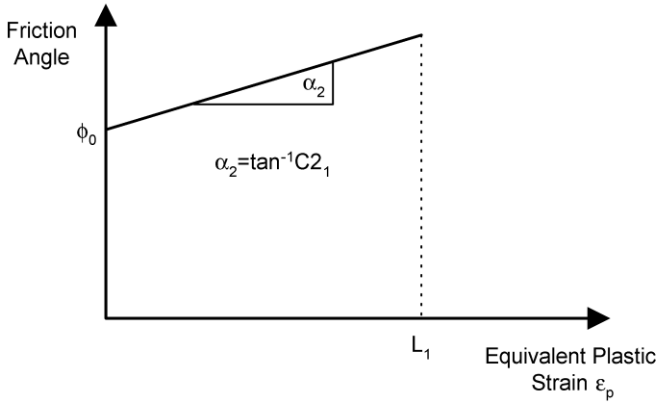
- ❑ **Initial Cohesion** A material property of granular materials, such as soils or rocks, describing the degree of granular bond and a measure of the shear strength. Setting the initial cohesion to zero is not recommended as this could cause numerical instability under certain loading conditions.
- ❑ **Initial Friction angle** A material property of granular materials, such as cohesive soils and rocks.
- ❑ **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat

produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.

- **Slope of Yield Stress** The slope of the uniaxial yield stress against equivalent plastic strain.
- **Plastic strain** The limit of equivalent plastic strain up to which the hardening curve is valid.



Cohesion Definition for the Drucker-Prager Yield Model (Model 64)



Friction Angle Definition for the Drucker-Prager Yield Model (Model 64)

See the *Solver Reference Manual* for further details.

LUSAS Multi Crack Concrete Models

When a structural analysis is being carried out LUSAS multi crack concrete models can be accessed by using the **Attributes> Material> Isotropic** menu item, clicking the **Plastic** check-box and selecting **Concrete**.

Two concrete models are available: the Multi Crack Concrete Model (Model 94), and the Smoothed Multi Crack Concrete Model (Model 102). Both can be used with 2D and 3D continuum elements, 2D and 3D explicit dynamics elements, solid composite elements and semiloof or thick shell elements. The capabilities, uses and limitations of both concrete material models are described in this topic, with notes as necessary when they are only relevant to one or the other.

LUSAS also supports three concrete creep model codes. See **Concrete Creep Models**.

Multi Crack Concrete Model (Model 94)

The Multi Crack Concrete model is a plastic-damage-contact model in which damage planes form according to a principal stress criterion and then develop as embedded rough contact planes. The basic softening curve used in the model may be controlled via a fixed softening curve or by a fracture-energy controlled softening curve that depends on the element size. The former, a distributed fracture model, is applicable to reinforced concrete applications, while the latter localised fracture model is applicable to un-reinforced cases.

- For this concrete model, crack/crush results can be plotted for the Values layer as component 'Crack/Crush' under the Stress entity. Planes representing the cracks are displayed in two and three-dimensions. Points of crushing are displayed as stars. Calculation and plotting of contours of crack widths is not possible using this concrete model.

Smoothed Multi Crack Concrete Model (Model 102)

The Smoothed Multi Crack Concrete model is similar to the Multi Crack Concrete model, converges more rapidly and is more robust, but requires a number of different input parameters. In contrast to the Multi Crack Concrete Model the Smoothed Multi Crack Concrete Model does not have distinct crack statuses (open or closed/interlocked).

- For this concrete model crack/crush results can be plotted for the Values layer as component 'Crack/Crush' under the Stress entity. Planes representing the cracks are displayed in two and three-dimensions. Points of crushing are displayed as stars.
- General crack width calculation is possible with this model and can be plotted for the Contour and Values layers as components of CWMax and EFSMax under the Plastic strain - Plane stress entity. Note that these components, will only be applicable for certain elements.
- When this concrete model is used in conjunction with **reinforcement attributes**, a linear steel material model, and with the **Crack Widths calculation utility**, crack

width calculations in accordance with EN 1992-1-1:2004 Eurocode 2 can be carried out. See *Application Manual (Bridge, Civil & Structural) Crack Width Calculation to EN 1992-1-1* for more information. Crack width information can be plotted for the Contour and Values layers as components of Maximum Crack Width under the Crack Widths EN1991-1 entity.

Note that a better estimation of crack widths is likely to be achieved when linear elements are used; the use of quadratic elements is likely to result in less accurate crack widths. A fine mesh may be required to capture the stress localisation band that results in the formation of a crack.

See the worked example 'Nonlinear analysis of a concrete beam' in the *Application Examples Manual (Bridge, Civil & Structural).*

Elastic Material Properties

- **Young's modulus** It is important that the Young's modulus, E, is consistent with the strain at peak compressive stress, ε_c. A reasonable check is to ensure that $E > 1.2\ f_c/\varepsilon_c$
- **Mass density** The mass density of concrete (ρ) is generally taken to be of the order of 2400kg/m³ for mass concrete and 2500kg/m³ for reinforced or prestressed concrete. Where more accurate determination is required information specific to the concrete in question should be obtained or a value determined experimentally.

Plastic Material Properties

These primary material properties apply to both unreinforced (mass concrete) and reinforced concrete except where stated.

- **Uniaxial compressive strength** (f_c) e.g. 40 N/mm^2. The generally available "characteristic compressive strength" may not be appropriate for use in a finite element analysis with a concrete material model. This is because cube strengths do not represent a uniaxial compressive strength due to the influence of significant boundary restraints caused by friction on the platens of any testing machine, so it may be necessary to convert a cube strength to a cylinder strength. Note also that, commonly, "characteristic" (5% lower percentile) compressive strength is used for engineering purposes, and consideration should also be given to whether a mean, characteristic, or other value is most appropriate to the analysis in question. The CEB-FIP Model Code 1990 equation 2.1-1 suggests that a mean cylinder strength (f_{cm}) may be estimated from a characteristic cylinder strength (f_{ck}).
- **Uniaxial tensile strength** (f_t) e.g. 3 N/mm^2. This is likely to be a principal parameter in any analysis where concrete cracking is of significance. Ideally, the tensile strength would be available from test results. In the absence of test results, a value must be assessed using available data and relationships found in literature. Many design codes offer estimates of the value and the CEB-FIP Model Code 1990 gives minimum, maximum and mean strengths based on an available characteristic cylinder strength.

- **Fracture energy per unit area** (G_f) e.g. 0.1 or 0 if e_0>0. For concrete, G_f is usually in the range 60 to 150J/m², varying with aggregate size but not so much with concrete strength. Where no data is available then a value of G_f = 0.13N/mm is suggested.

Notes relating to material properties

- The 'Uniaxial compressive strength' parameter in a concrete model is the ultimate strength of the material in compression. Since concrete behaves linearly only up to around 40% of the ultimate strength, crushing is expected to occur before that strength has been reached. As a result, concrete crush symbols will be plotted on the model (if requested to do so) when the plastic work-hardening parameter (which is calculated internally) is greater than zero. The presence of crushing symbols therefore indicates that the point at which the material begins to damage (i.e. crush) and behave nonlinearly, has been passed.

Advanced properties

These more advanced material properties are related to measurable characteristics. Additional information relating to some of the properties and suggested values are stated in the notes that follow.

- **Strain at peak uniaxial compression** (ε_c) e.g. 0.0022
- **Strain at end of softening curve** for distributed fracture (ε_o) e.g. 0.035 or 0 if G_f>0.
- **Biaxial to uniaxial stress ratio** (β_r) e.g. 1.15
- **Initial relative position of yield surface** (Z_o) e.g. 0.6
- **Dilatancy factor** (giving plastic potential slope relative to that of yield surface) (ψ). e.g. -0.1
- **Constant in interlock state function** (m_g) e.g. 0.425
- **Contact multiplier on $\boldsymbol{\varepsilon_o}$** for the 1st opening stage (m_{hi}) e.g. 0.5
- **Final contact multiplier on $\boldsymbol{\varepsilon_o}$** ($m_{ful}$) e.g. 5.0
- **Shear intercept to tensile strength ratio for local damage surface** ($r\sigma$) e.g. 1.25
- **Slope of friction asymptote for damage** (μ) e.g. 1.0
- **Angular limit between crack planes** (Radians) e.g.1.0 (for Multi Crack Concrete Model only)
- **Crack fixity strain** e.g. 0.05 (for Smoothed Multi Crack Concrete Model only)
- **Fracture process zone width** e.g. 0.06 (for Smoothed Multi Crack Concrete Model only)
- **Number of iterations** e.g. 2 (for Smoothed Multi Crack Concrete Model only)

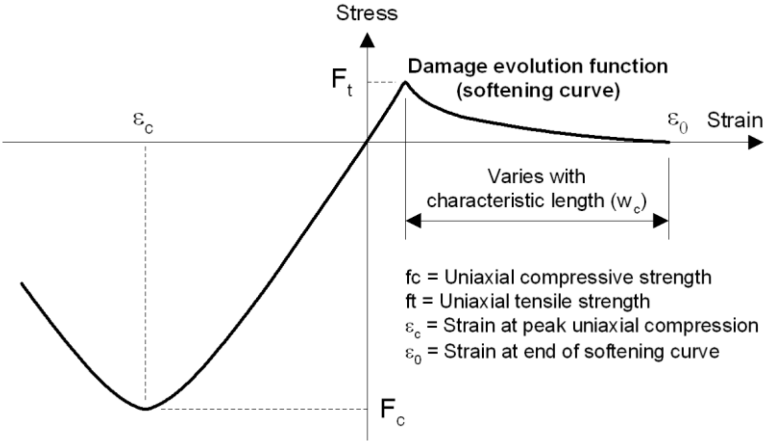
Notes relating to advanced properties

- **Strain at peak uniaxial compression** If no data for the strain at peak uniaxial compression is available it can be estimated see *Solver Reference Manual* for details. As a guide, a reasonable value for most types of concrete is 0.0022. It is important that the Young's modulus, E, (defined on the Elastic properties page) is consistent with the strain at peak uniaxial compression, ε_c. A reasonable check is to ensure that $E > 1.2\ f_c/\varepsilon_c$

- **Strain at end of softening curve** For reinforced concrete, distributed fracture will be the dominant fracture state. In this case a value for the strain at the end of the tensile softening curve, ε_0, should be entered and G_f set to zero. If no data is available then a value for ε_0 of 0.0035 is reasonable to use for most concretes.

- The value for ε_0 must be set to zero when the fracture energy per unit area, G_f, is given a positive value. G_f varies with aggregate size but not so much with concrete strength. Typical values for various maximum coarse aggregate sizes are:

  16 mm aggregate: $G_f = 0.1$ N/mm;
  20 mm aggregate: $G_f = 0.13$ N/mm;
  32 mm aggregate: $G_f = 0.16$ N/mm;



Stress / strain curve for multi crack concrete model (Model 94)

- If the effective end of the softening curve parameter, ε_0 , is set to zero, it will be calculated from $\varepsilon_0 \sim 5G_f / W_c\ f_t$ where W_c is a characteristic length for the element; if a finite value is given for e_0 , G_f will be ignored.

- **Biaxial to uniaxial stress ratio** A range of 1.05 to 1.3 is suggested and a default of 1.15 is provided.

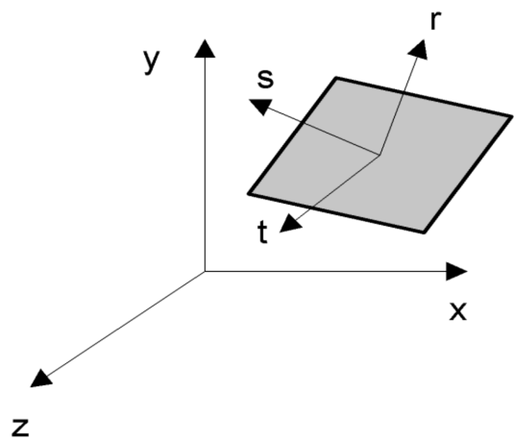
- **The initial relative position of yield surface** is governed by the value of Z_0. For most situations in which the degree of triaxial confinement is relatively low, a value of between 0.5 and 0.6 is considered appropriate for Z_0 however, for higher confinements a lower value of 0.25 is better.
- **Dilatancy factor** The parameter ψ is used to control the degree of dilatancy. Associated plastic flow is achieved if ψ=1, but it has been found that ψ values in the range -0.1 to –0.3 are required to match experimental results. Generally ψ is set to –0.1, but for high degrees of triaxial confinement –0.3 provides a better match to experimental data.
- **Constant in interlock state function** [m_g] can be obtained from experimental data from tests in which shear is applied to an open crack. The default value for m_g is taken as 0.425 but it is considered that a reasonable range for m_g for normal strength concrete is between 0.3 and 0.6. However, it was found that a low value of 0.3 could lead to second cracks forming at shallow angles to the first, due to the development of relatively large shear forces.
- **Contact multiplier on ε_0** [m_{hi}] is used to govern the amount of contact from micro-cracks. Experimental evidence suggests that the shear contact potential drops off more quickly for some concretes than for others, m_{hi} governs the early (micro-crack) loss of shear contact potential and m_{ful} controls the final amount (associated with coarse aggregate interlock). To truly calibrate these parameters, tests, which open cracks to different degrees and then apply shear, are required; however for a wide range of standard concrete m_{hi} = 0.5 and m_{ful} = 5.0 give reasonable results. A range of 0.25 to 2.0 is suggested and a default of 0.5 is provided
- **Final contact multiplier on ε_0** It is assumed that there is a crack opening strain beyond which no further contact can take place in shear, e_{ful}, where e_{ful} is a multiple of ε_0, i.e. $e_{ful}=m_{ful}\ \varepsilon_0$. Trials suggest that when concrete contains relatively large coarse aggregate, i.e. 20 to 30mm, a value of m_{ful} in the range 10-20 is appropriate, whereas for concrete with relatively small coarse aggregate, i.e. 5 to 8mm, a lower value is appropriate, in the range 3 to 5. This variation is necessary because the relative displacement at the end of a tension-softening curve (related via the characteristic dimension to ε_0.) is not in direct proportion to the coarse aggregate size, whereas the clearance displacement is roughly in proportion to the coarse aggregate size. Thus e_{ful} is not in a fixed ratio to ε_0.
- **Shear intercept to tensile strength** A range of 0.5 to 2.5 is suggested and a default value of 1.25 is provided.
- **Slope of friction asymptote for damage** A range of 0.5 to 1.5 is suggested and a default value of 1.0 is provided.
- **Number of iterations** This is the number of iterations beyond which the crack directions are fixed. Increasing the number of iterations from the default value of 2 usually makes convergence more difficult but can lead to greater accuracy in some

cases. If the number of iterations is increased , then the Maximum number of iterations on the Nonlinear and Transient Control dialog should also be increased.

Planes of degradation

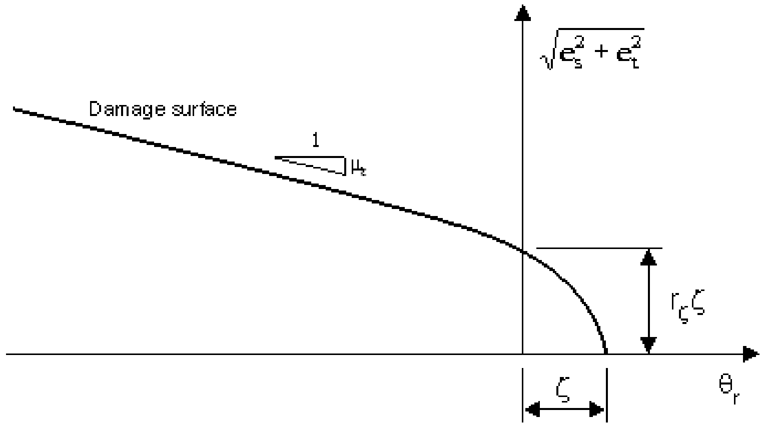
A Plane of Degradation (POD) is formed when the principal stress reaches the tensile strength (ft); the POD is formed normal to the major principal stress axis. Thereafter, further damage can develop due to the combined action of shear and direct (normal) strains:

- Further normal tensile stress would increase the damage, while both the stiffness of the material along the POD normal (i.e. the normal stiffness) and the shear stiffness, would decrease
- Further shear stress does not increase the damage; it causes aggregate interlock (i.e. contact) and an increase in both shear and normal stiffness
- Further normal compressive stress does not increase the damage; normal and shear stiffness would increase due to contact.



POD Local and Global Coordinate Systems

Local damage surface



Local Damage Surface

The constants r_ξ and μ_ε are the strain equivalents of the material input parameters r_σ and μ. The relative shear stress intercept to tensile strength ratio r_σ □= c / f_t where c is the shear stress intercept. Typical plots of the damage surface in stress space (the damage surface represents debonding or failure of the cement – aggregate interface) have shown the ratio between the shear strength and tensile strength of the interface bond to be approximately 1.25.

General notes

- If the dilatency factor (ψ) and the constant in interlock state function (m_g) are set to 1.0 a symmetric solution will be carried out, otherwise the non-symmetric solver will be invoked.
- It is recommended that fine integration be used with these material models as this helps prevent the occurrence of mechanisms when cracking occurs. However the use of fine integration is more expensive computationally and can degrade the performance of higher order element types.
- Line searches should not be invoked when using the Smoothed Multi Crack Concrete Model (Model 102), in NONLINEAR CONTROL, ITERATIONS, nalps should be set to zero.
- It is recommended that the following LUSAS options are used with these concrete models:

  252 Suppress pivot warnings.

  62 Allow negative pivots.

See the *Solver Reference Manual* and *Theory Manual* for further information relating to concrete material modelling.

Creep

Creep is the inelastic behaviour that occurs when the relationship between stress and strain is time dependent. The creep response is usually a function of the stress, strain, time and temperature history. Unlike time independent plasticity where a limited set of yield criteria may be applied to many materials, the creep response differs greatly for different materials.

Creep Properties

Power, exponential and eight parameter uniaxial creep laws are available and a time hardening form is available for each. The power creep law is also available in a strain hardening form. Fully 3D creep strains are computed using the differential of the von Mises or Hill stress potential. A user-definable creep interface is also available which allows a programmable uniaxial creep law.

Stress Potential

The definition of creep properties requires that the shape of the yield surface is defined. The stresses defining the yield surface are specified using the Stress Potential material model.

If a Stress Potential model is used in the Plastic definition then this will override the Creep stress potential and will apply to both the plastic properties and the creep properties. The Creep stress potential is only required when defining linear materials. If a stress potential type is not specified then von Mises is set as default.

None of the stresses defining the stress potential may be set to zero. For example, in a plane stress analysis, the out of plane direct stress must be given a value which physically represents the model to be analysed.

User Supplied Creep Properties

The user creep facility allows user supplied creep laws to be used. This facility provides completely general access to the LUSAS Solver property data input and provides controlled access to the pre- and post-solution constitutive processing and nonlinear state variable output.

Notes

- The user-supplied routine must return the increment in creep strain. Further, if implicit integration is to be used then the variation of the creep strain increment with respect to the equivalent stress, and also with respect to the creep strain increment, must be defined.
- If the function involves time dependent state variables they must be integrated in the user-supplied routine.

- If both plasticity and creep are defined for a material, the creep strains will be processed during the plastic strain update. Stresses in the user routine may therefore exceed the yield stress.
- User-supplied creep laws may be used as part of a composite element material assembly.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Damage

Damage is assumed to occur in a material by the initiation and growth of cavities and micro-cracks. The damage models allow parameters to be defined which control the initiation of damage and post-damage behaviour. Damage models are available for continuum and composite elements.

Continuum Damage Models

In LUSAS a scalar damage variable is used in the degradation of the elastic modulus matrix. This means that the effect of damage is considered to be non-directional or isotropic. Two LUSAS damage models are available (Simo and Oliver) together with a facility for a user-supplied model.

A damage analysis can be carried out using any of the elastic material models and the following nonlinear models:

- **von Mises**
- **Hill**
- **Hoffman**

Note. **Creep material properties** may be included in a damage analysis. See the *Solver Reference Manual* for further details.

Composite Matrix Failure Model

The composite matrix failure model simulates matrix failure using the Hashin damage criteria. The model can only be used with composite solid elements. It is defined under the **Damage** tab on the Material Orthotropic attribute dialog.

Material Properties

- Ply tensile strength in fibre direction
- Ply compressive strength in fibre direction
- Ply shear strength measured from a cross ply laminate
- Ply transverse tensile strength (normal to fibre direction)
- Ply transverse compressive strength

See the *Solver Reference Manual* for further details.

Viscoelastic

Viscoelasticity can be coupled with the linear elastic and non-linear plasticity, (isotropic or orthotropic), creep and damage models available in LUSAS. The model restricts the viscoelastic effects to the deviatoric component of the material response. This enables the viscoelastic material behaviour to be represented by a shear modulus G_v and a decay constant β. Viscoelasticity imposed in this way acts like a spring-damper in parallel with the elastic-plastic, damage and creep response. Coupling of the viscoelastic and the existing nonlinear material behaviour enables hysteresis effects to be modelled.

User Supplied Visco Elastic Properties

The user supplied viscoelastic properties facility enables routines for implementing a user supplied viscoelastic model to be invoked. This facility provides completely general access to the LUSAS Solver property data input via this data section and provides controlled access to the pre- and post-solution constitutive processing and nonlinear state variable output via these user supplied routines.

Notes

- When viscoelastic properties are coupled with a nonlinear material model it is assumed that the resulting viscoelastic stresses play no part in causing the material to yield and no part in any damage or creep calculations. Consequently the viscoelastic stresses are stored separately and deducted from the total stress vector at each iteration prior to any plasticity, creep or damage computations. Note that this applies to both implicit and explicit integration of the creep equations.
- Nonlinear Control must always be specified when viscoelastic properties are assigned. In addition Dynamic or Viscous Control must also be specified to provide a time step increment for use in the viscoelastic constitutive equations. If no time control is used the viscoelastic properties will be ignored.

See *Solver Reference Manual* for further details.

Shrinkage Properties

The cure of concrete and thermoset resins is accompanied by isotropic shrinkage which in the case of concrete depends on time, temperature and other environmental factors whilst for thermoset resins the shrinkage is normally described with respect to the degree of cure.

The shrinkage implementation in LUSAS allows an irreversible reduction in volume with time to be modelled. The shrinkage of concrete is accommodated using the equations defined in the design code CEB-FIP90 and also using a more general routine in which shrinkage is defined using a piecewise linear curve. In the general case, shrinkage can be defined as a function of time or degree of cure. A user facility is also available if required.

See the *Solver Reference Manual* for further details.

Two-Phase Material

Two-Phase material properties (defining a drained and undrained state) can be added to material definitions for Isotropic, Orthotropic and Modified Cam-Clay materials that are defined using the **Attributes> Material>** menu item. A **Two-phase** check box along the top of the material dialog adds a new property page to the current material definition, and allows Two-phase material data to be specified

Usage

Two-phase material properties are required when performing an analysis in which two- phase elements are used to define a drained and undrained state for soil. LUSAS can model the deformation of undrained/fully saturated and fully-drained/unsaturated porous media, and slow consolidation process, under isothermal conditions and small strain assumptions. For both 2D and 3D cases, nonlinear material models and large deformations are supported.

Two-phase material properties can be defined for the following soil types:

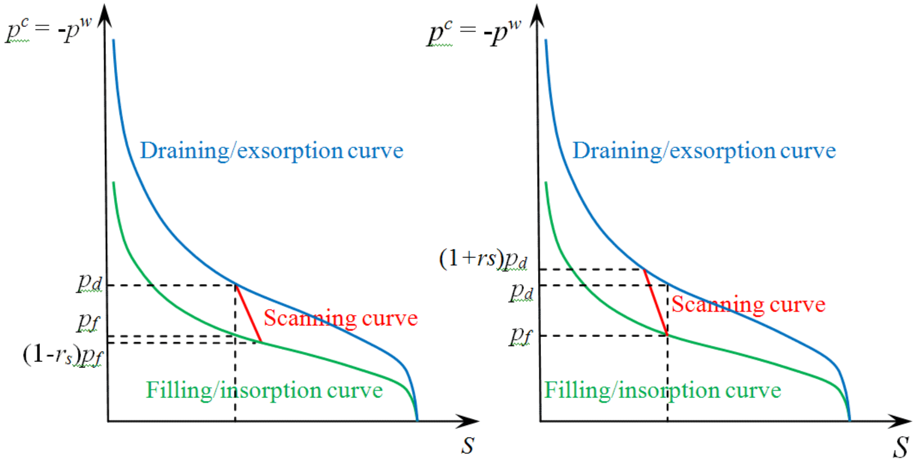
- **Saturated** - is the existing material with the addition of the fluid density. This requires general soil properties to be defined.
- **Partially saturated** - requires additional saturation properties, namely 'Irreducible saturation' and 'Degree of saturation to be considered fully saturated' to be specified.
- **Piecewise linear partially saturated** - requires the additional saturation values and a 'Curve tolerance' to be stated.

The **Define curves** button is only enabled for the new materials and allows the draining and filling curves to be defined.

The Bulk modulus of solid phase entry can be optionally excluded from the data input table by selection of the **Incompressible solids** option.

Draining and filling curves

Draining and filling curves can be specified for partially drained materials by defining the rate of water extraction, weight factor, air entry and permeability values. Manually-defined (piecewise linear) draining and filling curves can be defined for partially drained materials by specifying pore pressure, relative permeability and effective saturation values. For both methods, if a draining and filling curve is defined a **scanning curve factor** is used to specify how a jump should be made between the curves if jumping draining/exsorption to filling/insorption or vice-versa.



Typical draining and filling curve showing jump from draining/exsorption to filling/insorption (left) and jump from filling/insorption to draining/exsorption (right)

Notes

- Usually, the value of Bulk modulus of solid phase is quite large compared to Bulk modulus of fluid phase and not readily available to the user. If Bulk modulus of solid phase is input as 0, LUSAS assumes an incompressible solid phase. The bulk modulus of fluid phase is more obtainable, and for water is 2200 MPa.
- Two-phase material properties can only be assigned to two phase elements.
- When performing a linear consolidation analysis transient control settings must be specified specifying Consolidation in the Time domain section of the Nonlinear and Transient control dialog.
- Two-phase material properties may be combined with any other material properties and include creep, damage and viscoelastic properties if required.
- For some flow problems, a final steady state solution may only be required. For saturated soils a linear analysis should be used and this will give an exact solution. For unsaturated soils a nonlinear transient should be used to get accurate results. A nonlinear static analysis may be used but may give less accurate results. For example, the total discharge extracted from the reactions may be incorrect.
- For nonlinear solutions ill-conditioning may occur and a non-symmetric solution may be required. This can be specified by accessing the Nonlinear analysis options object. A Fast Parallel Direct Solver is recommended.

See *Solver Reference Manual* for further details.

Usage

Use of two-phase material and coupled pore fluid diffusion/stress analysis capabilities in LUSAS permits:

- Establishing the initial equilibrium state via a **geostatic analysis control** step.
- Modelling partially saturated fluid flow through porous medium, e.g. seepage of water through an earth dam, where the position of the phreatic surface (the boundary between fully saturated and partially saturated soil) is of interest.
- Including the influence of the pore fluid weight on the solid skeleton only (excess pore pressure solution) or on both the solid skeleton and fluid (total pore pressure solution).
- Using a default analytical capillary pressure relationship or defining a piece-wise relationship in a tabular form, as the accuracy of simulation depends on the use of an accurate relationship.
- Specifying different filling (or absorption) and draining (or exsorption) capillary (or pore water) pressure – effective saturation curves, which can exhibit hysteresis behaviour, as well as a scanning curve for transition between absorption and exsorption.
- In addition to prescribed head (pressure) and impervious (closed) boundary conditions, the inflow/outflow over a boundary can be considered. It is also possible to control the boundary condition automatically when a phreatic surface meets a boundary surface using lift-off supports.

Delamination Interface (Model 25)

The Delamination interface model appears under the **Attributes> Material> Specialised** menu item.

Delamination properties are assigned to interface elements to model delamination between elements. The model behaves linearly until the threshold strength is exceeded. Linear strain softening behaviour then occurs until the fracture energy is exceeded. Once the fracture energy is exceeded further straining occurs without resistance.

Fracture Modes

A 2D model has two fracture modes, a 3D model has three. The fracture modes are:

- **Mode 1** Opening
- **Mode 2** Shearing
- **Mode 3** Tearing (Shear orthogonal to mode 2)

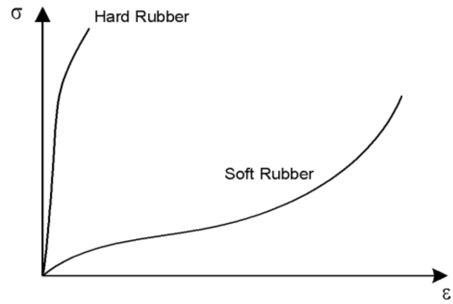
Material Parameters

- **Fracture energy** Measured values for each fracture mode depending on the material being used, i.e. carbon fibre, glass fibre.

- **Initiation Stress** The tension threshold /interface strength is the stress at which delamination is initiated. This should be a good estimate of the actual delamination tensile strength but, for many problems, the precise value has little effect on the computed response. If convergence difficulties arise it may be necessary to reduce the threshold values to obtain a solution.
- **Relative displacement** The maximum relative displacement is used to define the stiffness of the interface before failure. Provided it is sufficiently small to simulate an initially very stiff interface it will have little effect.
- **Coupled** The model used for coupling the failure modes (Coupled, Uncoupled Reversible, Uncoupled Origin).

See the *Theory Manual* for more details.

Rubber



Rubber materials maintain a linear relationship between stress and strain up to very large strains (typically 0.1 - 0.2). The behaviour after the proportional limit is exceeded depends on the type of rubber (see diagram below). Some kinds of soft rubber continue to stretch enormously without failure. The material eventually offers increasing resistance to the load, however, and the stress-strain curve turns markedly upward prior to failure. Rubber is, therefore, an exceptional material in that it remains elastic far beyond the proportional limit.

Rubber materials are also practically incompressible, that is, they retain their original volume under deformation. This is equivalent to specifying a Poisson's ratio approaching 0.5.

The strain measure used in LUSAS to model rubber deformation is termed a **stretch** and is measured in general terms as:

$$\lambda = \text{dnew}/\text{dold}$$

where:

dnew is the current length of a fibre.

dold is the original length of a fibre.

Several representations of the mechanical behaviour for hyper-elastic or rubber-like materials can be used for practical applications. Within LUSAS, the usual way of defining hyper-elasticity, i.e. to associate the hyper-elastic material to the existence of a strain energy function that represents this material, is employed. There are currently four rubber material models available:

- **Ogden**
- **Mooney-Rivlin**
- **Neo-Hookean**
- **Hencky**

The rubber constants (used for Ogden, Mooney-Rivlin and Neo-Hookean) are obtained from experimental testing or may be estimated from a stress-strain curve for the material together with a subsequent curve fitting exercise.

The Neo-Hookean and Mooney-Rivlin material models can be regarded as special cases of the more general Ogden material model. In LUSAS these models can be reformulated in terms of the Ogden model.

The strain energy functions used in these models includes both the deviatoric and volumetric parts and are, therefore, suitable to analyse rubber materials where some degree of compressibility is allowed. To enforce strict incompressibility (where the volumetric ratio equals unity), the bulk modulus tends to infinity and the resulting strain energy function only represents the deviatoric portion. This is particularly useful when the material is applied in plane stress problems where full incompressibility is assumed. However, such an assumption cannot be used in plane strain or 3D analyses because numerical difficulties can occur if a very high bulk modulus is used. In these cases, a small compressibility is mandatory but this should not cause concern since only near incompressibility needs to be ensured for most of the rubber like materials.

Using Rubber Material

Rubber is applicable for use with the following element types currently:

- ❑ **2D Continuum** QPM4M, QPN4M
- ❑ **3D Continuum** HX8M
- ❑ **2D Membrane** BXM2

Notes

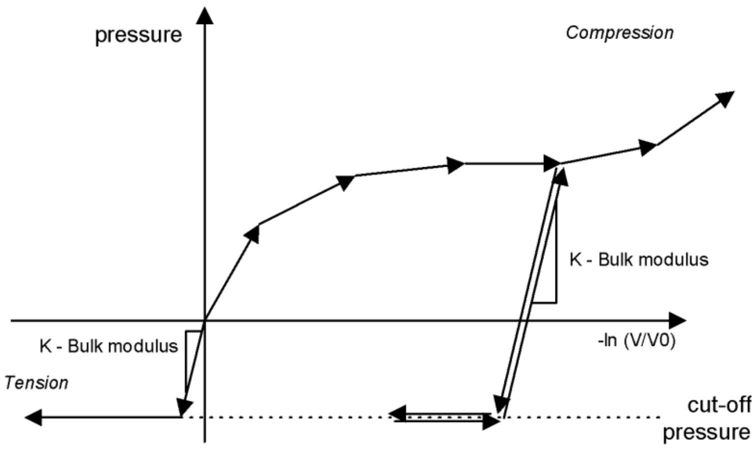
- For membrane and plane stress analyses, the bulk modulus is ignored because the formulation assumes full incompressibility. The bulk modulus has to be specified if any other 2D or 3D continuum element is used.
- Ogden, Mooney-Rivlin and Neo-Hookean material models must be run with geometric nonlinearity using either the total Lagrangian formulation (for membrane elements) or the co-rotational formulation (for continuum elements). The Hencky material model is only available for continuum elements and must be run using the co-rotational formulation. The large strain formulation is required in order to include the incompressibility constraints into the material definition.
- Option 39 can be specified for smoothing of stresses. This is particularly useful when the rubber model is used to analyse highly compressed plane strain or 3D continuum problems where oscillatory stresses may result in a "patchwork quilt" stress pattern. This option averages the Gauss point stresses to obtain a mean value for the element.
- When rubber materials are utilised, the value of det F or J (the volume ratio) is output at each Gauss point. The closeness of this value to 1.0 indicates the degree of incompressibility of the rubber model used. For totally incompressible materials

J=1.0. However, this is difficult to obtain due to numerical problems when a very high bulk modulus is introduced for plane strain and 3D analyses.

- Subsequent selection of state variables for displaying will include the variable PL1 which corresponds to the volume ratio.
- Rubber material models are not applicable for use with the axisymmetric solid element QAX4M since this element does not support the co-rotational geometric nonlinear formulation. The use of total Lagrangian would not be advised as an alternative.
- There are no associated triangular, tetrahedral or pentahedral elements for use with the rubber material models.
- The rubber material models are inherently nonlinear and, hence, must be used in conjunction with nonlinear control command.
- The rubber material models may be used in conjunction with any of the other LUSAS material models. However, it is not possible to combine rubber with any other nonlinear material model within the same material attribute.

Volumetric Crushing (Model 81)

Material behaviour can generally be described in terms of deviatoric and volumetric behaviour which combine to give the overall material response. The crushable foam material model accounts for both of these responses. The model defines the volumetric behaviour of the material by means of a piece-wise linear curve of pressure versus the logarithm of relative volume. An example of such a curve is shown in the diagram below, where relative volume is denoted by V/V_0.



Pressure - Logarithm of Relative Volume Curve

From this figure, it can also be seen that the material model permits two different unloading characteristics volumetrically.

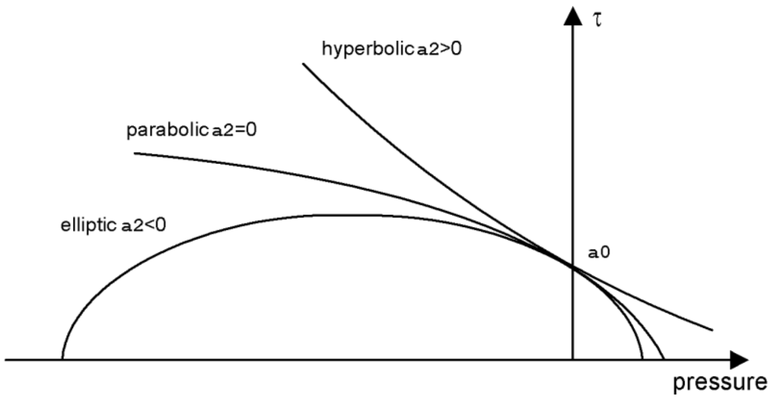
- **Unloading** may be in a nonlinear elastic manner in which loading and unloading take place along the same nonlinear curve.
- **Volumetric crushing** may be included (by clicking in the volumetric crushing check box) in which case unloading takes place along a straight line defined by the unloading/tensile bulk modulus `K` which is, in general, different from the initial compressive bulk modulus defined by the initial slope of the curve.

In both cases, however, there is a maximum (or cut-off) tensile stress, (cut-off pressure), that is employed to limit the amount of stress the material may sustain in tension.

The deviatoric behaviour of the material is assumed to be elastic-perfectly plastic. The plasticity is governed by a yield criterion that is dependent upon the volumetric pressure (compared with the classical von Mises yield stress dependency on equivalent plastic strain) and is defined as:

$$\tau^2 = a_0 - a_1 p + a_2 p^2$$

where p is the volumetric pressure, τ is the deviatoric stress and `a0`, `a1`, `a2` are pressure dependent yield stress constants. Note that, if `a1` = `a2` = 0 and `a0` = (`syld`2)/3, then classical von Mises yield criterion is obtained.



Yield Surface Representation For Different Pressure Dependent Yield Stress Values

Notes

- **Bulk modulus** used in tension and unloading (see 1st figure). The relationship between the elastic bulk (or volumetric) modulus, K, and tensile modulus, E, is given by:

  $$K = \frac{E}{3(1-2\nu)}$$

- **Shear modulus** The relationship between the elastic modulus values in shear, G, and tension, E, assuming small strain conditions, is given by:

  $$G = \frac{E}{2(1+\nu)}$$

- **Heat fraction coefficient** Represents the fraction of plastic work which is converted to heat and takes a value between 0 and 1.
- **Cut-off pressure** Should be negative (i.e. a tensile value).
- **Pressure dependent yield stresses** (`a0`, `a1`, `a2`) (Should be positive). The yield surface defined is quadratic with respect to the pressure variable. Therefore it can take on different conical forms (see 2nd figure), either elliptic (`a2`<0), parabolic (`a2`=0) or hyperbolic (`a2`>0). The parabolic form is comparable to the modified von Mises material model while the elliptic form can be considered to be a simplification of critical state soil and clay material behaviour.
- The **volumetric crushing** indicator effectively defines the unloading behaviour of the material. If there is no volumetric crushing, the same pressure-logarithm of relative volume curve is used in loading and unloading and if volumetric crushing takes place the alternative unloading/reloading curve is used (see 1st figure).
- **Log relative volume** Natural logarithm (loge, not log10) of relative volume coordinate for ith point on the pressure-logarithm of relative volume curve (see 1st figure)
- The **pressure-logarithm of relative volume** curve is defined in the compression regime hence logarithms of relative volume must all be zero or negative and the pressure coordinates must all be zero or positive.

Modified Cam-clay Model

The Modified Cam-clay model is defined using the **Attributes> Material> Specialised** menu.

The modified Cam-clay model is a variant of the classic Cam-clay model used for the modelling of clays, and is capable of modelling the strength and deformation trends of clay realistically. The model can be used with standard continuum elements as well as the two-phase elements that include the effects of pore water pressure. The modelling parameters required can be evaluated in standard laboratory tests.

Material Properties

- ❑ **Initial void ratio** - the void ratio of the soil at the start of the analysis
- ❑ **Swelling index** - relates to the change in specific volume to a decrease in mean pressure

- **Compression index** - relates the change in specific volume to an increase in mean pressure
- **Gradient critical state line** - relates the deviatoric stress to the mean pressure
- **Pre-consolidation pressure** - the maximum pressure the soil was subjected to prior to analysis
- **Poisson's ratio**
- **Mass density**

Notes

- The material modulus matrix is non-symmetric. Convergence may be improved by using a **non-symmetric** solution. This can be specified by accessing the Nonlinear analysis options control for a loadcase.
- The initial stress state can be input by using residual stress load types, or by using an initial stress with gravity.

Dynamic and Thermal Properties

Dynamic and Thermal expansion properties are optional and are usually only required when dynamic or thermal analyses are to be carried out.

Two-Phase Properties

Two-phase material properties are required when performing a consolidation analysis, in which two-phase elements are used to define a drained and undrained state for soil.

Varying Attributes

Some properties of the material typically vary with depth, for example, the initial void ratio; initial pre-consolidation pressure, and so on. Any such varying properties may be specified by selecting a previously defined variation. To do this press the variation button when you enter a value in any field.

See Solver Reference Manual : Modified Cam-Clay Model

Results processing

Once a model containing this material has been solved, Mean pressure, Effective stress, Shear modulus, Consolidation pressure, Bulk modulus and Percent change in specific volume components are all available within the results entity Cam-Clay Results.

Elasto-Plastic Interface (Model 27, 26)

Elasto-plastic interface models may be used to represent the friction-contact relationship within planes of weakness between two discrete bodies.

- ❑ The 2D Elasto-Plastic Interface Model (Model 27) is available in plane stress and plane strain elements. See the *Element Reference Manual* for more details.
- ❑ The 3D Elasto-Plastic Interface Model (Model 26) is available in solid continuum elements. See the *Element Reference Manual* for more details.

For both models, a line of appropriate 2D or 3D elements must lie between the two bodies in the finite element discretisation.

Elastic material properties are defined in the local basis, permitting differing values to be specified normal and tangential to the plane of the interface. In-plane and out-of-plane directions are relative to the surface axes, and refer to the contact plane rather than the plane of the surface features.

The nonlinear behaviour is governed by an elasto-plastic constitutive law, which is formulated with a limited tension criterion normal to the interface plane, and a Mohr-Coulomb criterion tangential to the interface plane. See the *Theory Manual* for further details.

Notes

- For the 2D model the tangential (in-plane) direction is in the element x direction and the normal (out of plane) is in the element h direction. The x direction is defined from the vector between nodes 1 and 3 of the element. See the *Solver Manual* for more details.
- For the 3D model the tangential (in-plane) direction lies in the element x-h plane and the normal (out of plane) in the element z direction. See the *Solver Manual* for more details.
- The models cannot be used within a geometrically nonlinear analysis

Concrete Creep Models

LUSAS accommodates three concrete creep model codes:

- ❑ **CEB-FIP Model Code 1990** - Uses a simplified linear approach to represent creep. This assumes that the service stresses in the concrete will not be exceeded and hence may not predict the effects of unloading or load cycling accurately. The model is valid for ordinary structural concrete (12-80 MPa) that has been loaded in compression to less than 40% of its compressive strength at the time of loading.
- ❑ **Chinese Creep Code (Draft 2007)**- Based upon equations taken from the Chinese Creep Code for Dams. Allows creep of concrete to be computed as concrete ages while subjected to external forces, including thermal stresses that may be due to the hydration process.
- ❑ **EN1992-1-1:2004 Eurocode 2** - Uses a simplified linear approach to represent creep. This assumption assumes that the service stresses in the concrete will not be exceeded and hence may not predict the effects of unloading or load cycling accurately. The model is appropriate for cases where the compressive stress in concrete is less than 45% of its compressive strength at the time of loading.

Note

Although CEB-FIP Model Code 1990 and EN1992-1-1:2004 Eurocode 2 are only applicable to beams, they have been extended in LUSAS to cover multi-axial stress states. The assumptions made in the derivation of this extension can be found in the LUSAS Theory Manual.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Concrete Creep and Shrinkage to CEB-FIP Model Code 1990 (LUSAS Model 86)

Concrete material properties to CEB-FIP Model Code 1990 are defined from the **Attributes> Material> Specialised**menu.

The CEB-FIP Model Code 1990 model uses a simplified linear approach to represent creep. This assumption assumes that the service stresses in the concrete will not be exceeded and hence may not predict the effects of unloading or load cycling accurately. The model is valid for ordinary structural concrete (12-80 MPa) that has been loaded in compression to less than 40% of its compressive strength at the time of loading. Relative humidities in the range 40-100% and temperatures in the range 5-30 degrees Celsius are assumed.

Material Properties

The following material parameters are required, citing clauses from CEB-FIP Model Code 1990 where appropriate.

- **Tangent modulus of elasticity** - the tangent modulus of elasticity of concrete at 28 days, E_{ci}. This value is typically determined from equation 2.1-15, adjusted for aggregate type according to the commentary to clause 2.1.4.2. Table 2.1.6 gives values in the range $27\times10^3 \leq E_{ci} \leq 44\times10^3$ MPa. Taking into account the adjustment for aggregate, the expected range of values is therefore $18\times10^3 \leq E_{ci} \leq 53\times10^3$ MPa. The default value of $E_{ci} = 36\times10^3$ MPa corresponds to Grade C40 concrete and quartzite aggregate.
- **Poisson's Ratio** - will lie in the range $0.1 \leq \nu_c \leq 0.2$ according to clause 2.1.4.3. A value of $\nu_c = 0.2$, corresponds to an uncracked section, acknowledging the primary use of the creep material in prestressed concrete where cracking is minimised.
- **Mass density** - is in the range $2400 \leq \rho \leq 2500$kg/m³ for normal weight concrete according to clause 2.1.2. Typically it is taken as ρ= 2500kg/m³.
- **Coefficient of thermal expansion** - may be taken as 10E-6 per degree K according to clause 2.1.8.3.
- **Mean compressive strength at 28 days** - refers to the mean value of compressive strength, f_{cm}, (the cylinder strength) associated with the characteristic compressive strength, f_{ck}. This value is typically determined from equation 2.1-1, that is, $f_{cm} = F_{ck} +$ 8MPa. For grades C12 to C80, as per clause 2.1.1.2, this would give a likely range 20 $\leq f_{cm} \leq$ 88MPa. The default value of f_{cm} = 48MPa, corresponds to Grade C40.

- **Cement type** - refers to clauses 2.1.6.1 and 2.1.6.4.3(c), and may be **Slow Hardening** , **Normal or Rapid Hardening** (the default option) or **Rapid Hardening High Strength**.
- **Relative humidity** - is used in the calculation of both creep coefficient and, where required, shrinkage. Typical percentage values according to Table 2.1.10. would be 50 for indoor, dry atmospheric conditions, or 80 for outdoor, humid atmospheric conditions.
- **Internal perimeter factor** or **Nominal size** appear depending upon selections made on dialog. See Calculation of nominal section size for further details.

Creep calculations

Creep calculations are carried out to CEB-FIP Model Code 1990 clause 2.1.6.4.3 with the following limitations:

Substantial deviations from a mean concrete temperature of 20°C are not considered. According to clause 2.1.6.1, the age of the concrete should be adjusted according to equation 2.1-87. This affects the modulus of elasticity (equation 2.1-58) and the notional creep coefficient (equation 2.1-65). This adjustment for temperature is not implemented in the LUSAS model code, therefore the calculations are only strictly valid for mean temperatures from 5°C to 30°C (see clause 2.1.6.4.2).

Nonlinear effects of high stresses are not considered. According to clause 2.1.6.4.3(d), a nonlinear notional creep coefficient (equation 2.1-73a, 2.1-73b) should be used for areas of high stress. This is not implemented in the LUSAS model code, therefore the calculations are only strictly applicable for a stress level where $\sigma_c \leq 0.4f_{cm}(t_0)$.

Shrinkage

The effects of shrinkage can be included in accordance with clause 2.1.6.4.4.

Calculation of nominal section size

Nominal section size can be calculated automatically from the section dimensions using either a full perimeter (outer and inner, if voids are present) or by specifying an internal perimeter factor. Alternatively a nominal section size may be specified directly. For beams the nominal size of member is computed automatically by LUSAS for each element, and for tapered sections will vary along the beam.

- **Automatically calculate the nominal size from section dimensions** and **Use full perimeter** when selected together cause the nominal size of section to be calculated as $2A_c/u$ where A_c is the area of the cross section and u is the length of the perimeter of the cross section that is in contact with the atmosphere - both externally and internally if voids are present.
- **Automatically calculate the nominal size from section dimensions** and **Specify internal perimeter factor** when selected together cause the internal perimeter factor, termed k_s here, to be in the calculation of, u, the total perimeter of the member in

contact with the atmosphere. The calculation of u takes the form: $u = u_o + k_s \times u_i$ - where, for sections calculated using LUSAS Section Property Calculators, u_o is the outer perimeter and u_i is the internal perimeter of any voids (e.g. the cell of a box girder section). The value of u determined in this way will be used in the calculation of nominal size, h, following CEB-FIP Model Code 1990 clause 2.1.6.4.3(b). For voided cross-sections a default internal perimeter factor of 0.5 is entered into the dialog (and can be modified to suit the situation). For solid sections the value of internal perimeter factor is irrelevant.

- When **Specify nominal size** is selected the Nominal size entered should be compatible with the modelling units in use.

Notes

- The use of an internal perimeter factor to calculate a nominal section size is not set out in CEB-FIP Model Code 1990, however it is adopted by other similar codes, including AASHTO, which in clause 5.4.2.3.2 states: 'The surface area used in determining the volume-to-surface ratio should include only the area that is exposed to atmospheric drying. For poorly ventilated enclosed cells, only 50 percent of the interior perimeter should be used in calculating the surface area.' To use the approach as described in AASHTO therefore, k_s=0.5 would be used.
- CEB-FIP Model Code 1990 can be used with beam elements, 2/3D contimuum elements, semi-loof and thick shells, and composite solid elements. CEB-FIP Model Code 90 is only strictly applicable to beams, however, in LUSAS the creep equations have been extended to 2D and 3D stress states. The assumptions made in the derivation of this extension can be found in the LUSAS Theory Manual. It must be noted that it may be difficult to establish an appropriate value of "nominal size" for anything other than beam elements.
- The CEB-FIP Model Code 1990 creep and shrinkage model must be run in a transient nonlinear analysis in which the time step and analysis termination times must be specified in units that represent days. The meaning of the time step and total response time values is specified on the Model Properties dialog. An option exists in the advanced time step options to use an exponent to increase the time step as the analysis progresses.
- The age of concrete at the time an element is introduced to the analysis may be defined using the **Age** attribute.
- Nonlinearity of creep due to high compressive stress (eqn 2.1-73 in CEB-FIP Model Code 1990); the effect of type of cement and curing temperature (eqn 2.1-72 in CEB-FIP Model Code 1990); and the effect of temperature on the maturity of concrete (eqn 2.1-87 in CEB-FIP Model Code 1990) are not currently considered and should be assessed using code requirements.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Concrete Creep and Shrinkage to EN1992-1-1:2004

Concrete material properties to EN1992-1-1:2004 are defined from the **Attributes> Material> Specialised** menu.

The EN1992-1-1:2004 creep and shrinkage model uses a simplified linear approach to represent creep. This assumes that the service stresses in the concrete will not be exceeded and hence may not predict the effects of unloading or load cycling accurately. The model is appropriate for cases where the compressive stress in concrete is less than 45% of its compressive strength at the time of loading.

Material Properties

The following material parameters are required:

- **Tangent modulus of elasticity** - the tangent modulus of elasticity of concrete at 28 days, E_c This value is typically determined from E_{cm} in Table 3.1, adjusted for aggregate type according to the commentary to clause 3.1.3(2) and using E_c = 1.05Ecm according to clause 3.1.4(2). Taking into account the adjustment for aggregate and 1.05 factor, the expected range of values is therefore $19\times10^3 \leq E_c \leq 56\times10^3$ MPa. The default value of $E_c = 36.75\times10^3$ MPa, corresponds to concrete with f_{ck} = 40MPa and quartzite aggregate.
- **Poisson's Ratio** - may be taken as 0.2 for uncracked concrete and 0 for cracked concrete according to clause 3.1.3(4). A value of $\nu_c = 0.2$, corresponds to an uncracked section, acknowledging the primary use of the creep material in prestressed concrete where cracking is minimised.
- **Mass density** - may be taken as ρ= 25 kN/m³ = 2548.4kg/m³ for normal weight concrete according to Table A.1,
- **Coefficient of thermal expansion** - may be taken as 10E-6 per K according to clause 3.1.3(5).
- **Mean compressive strength at 28 days** - refers to the mean value of compressive strength, f_{cm}, (the cylinder strength) associated with the characteristic compressive strength. This value is typically determined from Table 3.1., i.e. $f_{cm} = F_{ck}$ + 8MPa. This would give a likely range $20 \leq f_{cm} \leq 98$MPa. The default value of f_{cm} = 48MPa, corresponds to F_{ck} =40Mpa.
- **Cement type** - refers to cement type clauses 3.1.2(6) and Annex B, clause B.1(2) and may be: **Class S** (slowly hardening cements), **Class N** (Normal or rapidly hardening cements) or **Class R** Rapid hardening high strength cements. **Class N** is the default option.
- **Relative humidity** - is used in the calculation of both creep coefficient and, where required, shrinkage. Typical percentage values according to Figure 3.1. would be 50 for indoor, dry atmospheric conditions, or 80 for outdoor, humid atmospheric conditions
- **Internal perimeter factor** or **Nominal size** appear depending upon selections made on the dialog. See Calculation of nominal section size for more details.

Creep calculations

Creep calculations are carried out to EN1992-1-1:2004 clause 3.1.4 and Annex B.1 with the following limitations:

- Elevated or reduced temperatures within the range 0 to 80°C are not considered. According to clause B.1(3), the concrete age should be adjusted according to expression B.10. This affects the modulus of elasticity (expression 3.5) and the notional creep coefficient (expression B.5). This adjustment for temperature is not implemented in the LUSAS model, therefore the calculations would be less accurate where ambient temperatures fall outside the range -40°C to +40°C (this range is inferred from clause 3.1.4(5).)
- Nonlinear effects of high stresses are not considered. According to clause 3.1.4(4), a nonlinear notional creep coefficient (expression 3.7) should be used for areas where compressive stress σ_c>0.45f_ck (t_0). This is not implemented in the LUSAS model code, therefore the calculations are only strictly applicable for compressive stresses which do not exceed this threshold.

Shrinkage

The effects of shrinkage can be included in accordance with clause 3.1.4(6) and Annex B clause B.2. Clause 3.1.4(6) states: "The total shrinkage strain is composed of two components, the drying shrinkage strain and the autogenous shrinkage strain. The drying shrinkage strain develops slowly, since it is a function of the migration of the water through the hardened concrete. The autogenous shrinkage strain develops during hardening of the concrete in the early days after casting and should be considered specifically when new concrete is cast against hardened concrete".

When shrinkage calculations are switched on, an option exists for the inclusion or exclusion of autogenous shrinkage.

Calculation of nominal section size

Nominal section size can be calculated automatically from the section dimensions using either a full perimeter (outer and inner, if voids are present) or by specifying an internal perimeter factor. Alternatively a nominal section size may be specified directly. For beams the nominal size of member is computed automatically by LUSAS for each element, and for tapered sections will vary along the beam.

- ❑ **Automatically calculate the nominal size from section dimensions** and **Use full perimeter** when selected together cause the nominal size of section to be calculated according to clause 3.1.4(5) as $2A_c/u$ where A_c is the area of the cross section and u is the length of the perimeter of the cross section that is in contact with the atmosphere - both externally and internally if voids are present.
- ❑ **Automatically calculate the nominal size from section dimensions** and **Specify internal perimeter factor** when selected together, cause the internal perimeter factor,

termed k_s here, to be used in the calculation of, u, the total perimeter of the member in contact with the atmosphere. The calculation of u takes the form: $u = u_o + k_s \times u_i$ - where, for sections calculated using LUSAS Section Property Calculators, u_o is the outer perimeter and u_i is the internal perimeter of any voids (e.g. the cell of a box girder section). The value of u determined in this way will be used in the calculation of nominal size, h, following clause 3.1.4(5). For voided cross-sections a default internal perimeter factor of 0.5 is entered into the dialog (and can be modified to suit the situation). For solid sections the value of internal perimeter factor is irrelevant.

- When **Specify nominal size** is selected the Nominal size entered should be compatible with the modelling units in use.

Notes

- The EN1992-1-1:2004 creep and shrinkage model can be used with beam elements, 2/3D continuum elements, semi-loof and thick shells, and composite solid elements. EN1992-1-1:2004 creep and shrinkage is only strictly applicable to beams, however, in LUSAS the creep equations have been extended to 2D and 3D stress states. The assumptions made in the derivation of this extension can be found in the LUSAS Theory Manual. It must be noted that it may be difficult to establish an appropriate value of "nominal size" for anything other than beam elements.

- The EN1992-1-1:2004 creep and shrinkage model must be run in a transient nonlinear analysis in which the time step and analysis termination times must be specified in units that represent days. The meaning of the time step and total response time values is specified on the Model Properties dialog. An option exists in the advanced time step options to use an exponent to increase the time step as the analysis progresses

- The age of concrete at the time an element is introduced to the analysis may be defined using the **Age** attribute.

- Nonlinearity of creep due to high compressive stress (eqn 3.7 in EN1992-1-1:2004); the effect of type of cement and curing temperature (eqn B.9 in EN1992-1-1:2004); and the effect of temperature on the maturity of concrete (eqn B.10 in EN1992-1-1:2004) are not currently considered and should be assessed using code requirements.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Concrete Creep: Chinese Code (2007 Draft)

This material model is accessed via the **Attributes> Material> Specialised** menu.

It allows the creep of concrete to be computed as concrete ages while subjected to external forces including thermal stresses that may be due to the hydration process.

The equations used to compute the creep strains are taken from the Chinese Creep Code for Dams. These equations are based on linear visco-elastic theory and hence may not predict the effects of unloading or load cycling accurately. Thermal stresses due to the heat of hydration can be included in the computation by running a semi-coupled thermo-mechanical analysis. Further information on both these facilities can be found in The LUSAS Theory Manual.

Material Properties

The following material properties are required

- **Long term Young's Modulus**
- **Poisson's ratio**
- **Mass density**
- **Coefficient of thermal expansion**
- **Parameters for controlling the evolution of Young's modulus with time (a, b)**
- **Parameters for controlling variation of creep coefficient with time (fi, gi, pi, ri...)**
- **Thermal properties if running a thermo-mechanical coupled analysis**

For full details and descriptions of these input parameters please refer to the LUSAS Solver Manual.

Notes

- If there is a lack of experimental data to complete the LUSAS dialog the Chinese Creep Code for dams recommends the following default input parameters:

$^{\infty}E = 1.05\,^{360}E$ or $^{\infty}E = 1.20\,^{90}E$, A = 0.40, B = 0.34

where ^{360}E and ^{90}E are instant elastic modulus at 360 and 90 days, respectively, and

F1 = C1, G1 = 9.20C1, P1 = 0.45, R1 = 0.30, C1 = 0.23/ $^{\infty}E$

F2 = C2, G2 = 1.70C2, P2 = 0.45, R2 = 0.005, C2 = 0.52/ $^{\infty}E$

F3 = R3 = 0

- The Chinese creep model must be run in a transient analysis in which the time step and analysis termination times must be specified in units that represent days. An option exists in the advanced time step options to use an exponent to increase the time step as the analysis progresses.
- The age of concrete at the time an element is introduced to the analysis may be defined using the **Age** attribute

Generic Polymer (Model 87 and 88)

The Generic Polymer model is accessed from the **Attributes> Material> Specialised** menu item.

The model consists of a set of Maxwell dampers which are used to model visco-elasticity, an Eyring dashpot which is used to model viscoplasticity and a linear spring. These components are placed in series. The Properties of the Maxwell elements, Eyring dashpot and linear spring can be different in tension and compression.

Model 87 is defined using an linear spring, an Eyring damper, and a number of parallel Maxwell elements. Model 88 is used when behaviour differs in tension and compression

Material Properties

- **Bulk Modulus**
- **Mass density**
- **Coefficient of thermal expansion**
- **Mass Rayleigh damping constant**
- **Stiffness Rayleigh damping constant**
- **Eyring damper activation energy**
- **Eyring damper activation volume**
- **External spring stiffness**
- **Maxwell element spring constant**
- **Maxwell element Newtonian dashpot viscous constant**

See the *Solver Reference Manual* and the*Theory Manual* for further details.

Generic Polymer with Damage (Model 89 and 90)

The Generic Polymer with Damage model appears under the **Attributes> Material> Specialised** menu item. The model accounts for strain rate behaviour and irrecoverable damage in the modelling of polymers and other polymer-like materials.

- **Model 89** consists of a linear spring, a spring that includes damage, an Eyring dashpot and a number of parallel Maxwell elements. Different sets of material properties can be defined to model tensile and compressive behaviour, except for the Visco-Scram damage model whose properties apply in both tension and compression.
- **Model 90** includes two additional features. The first feature is a set of failure criteria that are used to remove individual Maxwell elements when the criteria are met. The second is a switch between tensile and compressive material properties based on the stress state in an individual Maxwell element.

Note that for the Visco-Scram Damage model the expected input units for the Threshold Stress Intensity parameter cannot be displayed when the cursor is hovered over the input cell. This is because the power value concerned cannot be created with this display method.

Material Properties

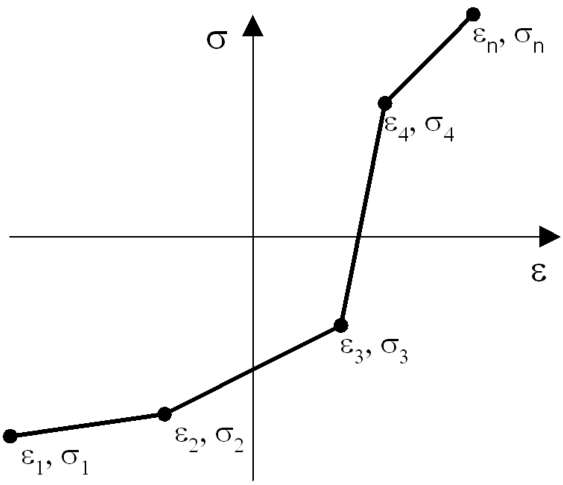
See the *Solver Reference Manual* and the*Theory Manual* for further details.

Piecewise Linear Bar Material (Model 104)

Piecewise linear bar material is for use with 2-noded bar elements to meet the requirements of no compression (tension-only) or no-tension (compression-only) members. The nonlinear elastic behaviour is recoverable, and during the analysis the stiffness (and stress) will be computed from the strain value using the relationship shown in the image below.

General usage

For general use the piecewise linear bar material model is accessed via the **Attributes> Material> Specialised> Piecewise Linear (Bar)** menu item. Any number of piecewise linear segments can be defined to represent material behaviour. The stress-strain curve defined does not have to pass through the origin. By defining curve data that runs along the positive (or negative) strain axis, no compression (tension-only) or no tension (compression-only) behaviour can be simulated.



General piecewise linear definition

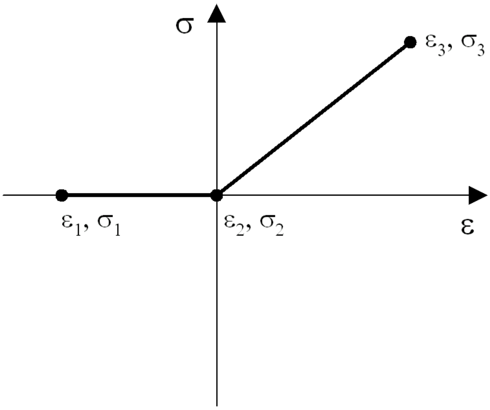
Editing curve data

Once defined, piecewise linear bar material attributes are held in the Attributes Treeview.

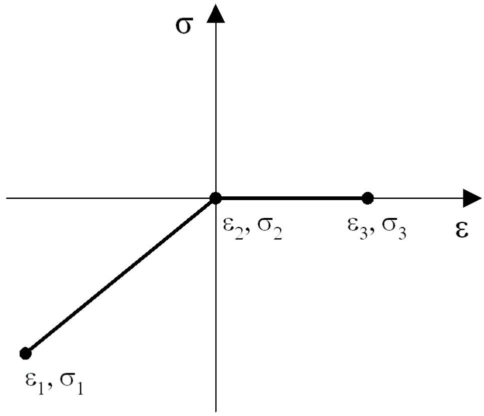
- Selecting the **Edit Attribute...** context menu for a bar material attribute displays the piecewise linear bar material data for editing.

Tension-only and compression-only usage

For simplified use, the piecewise linear bar material model is also used by the **Attributes> Material> Tension / Compression Only (Bar)** menu item. For a selected or user-defined material, automatically created curve data, that runs along the positive (or negative) strain axis, is used to define a simple no compression (tension-only)or no tension (compression-only) or behaviour.



Tension-only bar material



Compression-only bar material

Editing curve data

Once defined, piecewise linear bar material attributes are held in the Attributes Treeview, and have context menu entries named Edit Definition... and Edit Attribute.

- ❑ Selecting the **Edit Definition...** menu entry for a bar material attribute, or double clicking the attribute name displays the original definition dialog with all the original input data intact for modification.
- ❑ Selecting the **Edit Attribute...** menu entry for a bar material attribute displays the equivalent piecewise linear bar material data for the defined data. These values may be changed but this breaks the link to the original definition dialog and a warning message will be displayed.

Notes

- The input data must not contain a curve segment with a negative or vertical slope.
- Loading and unloading is assumed to occur along the same stress/strain path.
- If strains exceed the bounds of the curve data the first or last curve segment will be used to extrapolate the stress.
- A curve tolerance parameter is used to help prevent chatter (oscillations between adjacent segments when a computed stress/strain is very close to an apex on the curve). If the point falls within this tolerance an average Young's modulus value to form the stiffness is computed from the adjacent segments.
- Defining a non-zero stress for zero strain has the same affect as applying an initial stress. Under these conditions, if an initial stress is also applied, the compound effect

of both stresses will be applied. In other words, an applied initial stress loading will act as an offset on the stress data entered for this material model.

- Tension only /compression only material requires the use of a nonlinear material definition and suitable nonlinear controls.
- Carrying out an eigenvalue analysis following a nonlinear analysis that uses this model can lead to unrealistic bar stresses for some modes. If a no-compression bar element has zero stiffness prior to the eigenvalue analysis, then some modes will inevitably see that element put into tension.

Resultant User

Used to specify user material parameters for the user defined nonlinear resultant model.

See the *Solver Reference Manual* for further details.

Nonlinear User

This is used to specify user material parameters for the user defined nonlinear material model.

See the *Solver Reference Manual* for further details.

Material Properties Nonlinear Thermal User

This is used to enter material parameters for user nonlinear thermal material models. Required inputs are:

- ❑ **User material model number** The defined material model number
- ❑ **Number of state variables** The number of variables required for storing results at gauss points
- ❑ **Values** Input parameters for the user-defined material model listed in the order that they are required

Joint Material Models

Joint material models are used in conjunction with joint elements to define the material properties for linear and nonlinear joints. The following joint models are available:

Linear Joint Models

- ❑ **Spring stiffness only** corresponding to each local freedom. These local directions are defined for each joint element in the *Element Reference Manual*.
- ❑ **General Properties** advanced specification of joint properties of spring stiffness, mass, coefficient of linear expansion and damping factor.

Nonlinear Joint Models

- ❑ **Elasto-Plastic (Tension and compression equal)** with isotropic hardening and equal tension and compression yield conditions.

- **Elasto-Plastic (Tension and compression unequal)** with isotropic hardening and unequal tension and compression yield conditions.
- **Smooth Contact** with an initial gap. See notes below.
- **Frictional Contact** with an initial gap. See notes below.
- **Trilinear Earth Pressure** creates a piecewise linear joint material whose properties vary with depth, such as those used to represent a layer or stratum in a soil-structure interaction or geotechnical analysis.
- **Piecewise linear** allows a single force/displacement (moment/rotation) curve to be defined for each freedom by defining a number of line (curve) segments.
- **Piecewise linear elastic (Axial force dependent)** Allows any number of force/displacement (moment/rotation) curves representing the behaviour under different axial forces. The joint is typically used for modelling plastic hinges in pushover analysis.
- **Nonlinear user-defined** A user-defined joint model.

Seismic Isolator Joint Models

- **Viscous dampers** - Kelvin and Four Parameter Solid modules available.
- **Lead Rubber Bearings** with plastic yield and biaxial hysteric behaviour.
- **Friction Pendulum System** with pressure and velocity dependent friction coefficient and biaxial hysteretic behaviour.

Joint material properties

Joint material properties are assigned to model geometry using joint elements to define the joint behaviour. Joint input parameters and the number of degrees of freedom will vary according to the joint model and element type chosen

- **Joint type** selection of a joint type adds the properties relevant for that joint type to the tabular grid on the dialog.
- **Mass position** specifies that the joint mass will be at the first node, second node, or between nodes

Properties for each degree of freedom must be specified in the properties grid of the dialog. Some joint materials have properties that can be entered to be common for all freedoms. Additional **Thermal expansion**, **Varying friction coefficient** and **Damping** options are selectable according to joint type.

See **Joint and Interface Elements** for general information about using joints.

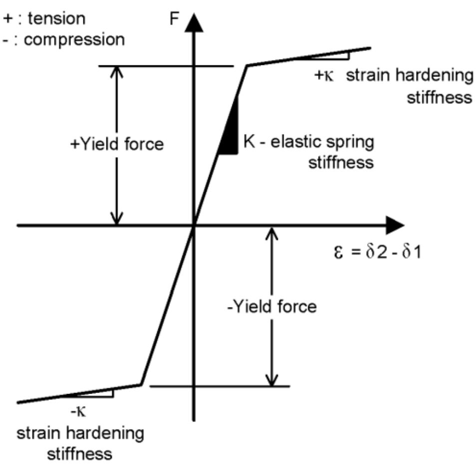
Refer to the *Solver Reference Manual*for a full description of the joint material input parameters required for these joint models.

Notes

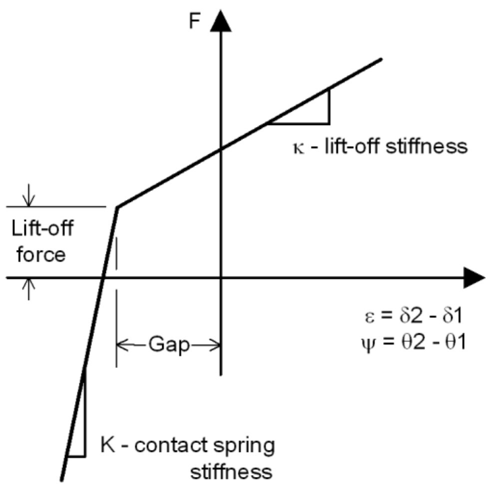
- When defining joint properties for single joint elements the total stiffness or yield force should be defined. When using interface joint meshing the stiffness and yield force defined in the joint properties should be defined per unit length when using interface joints assigned to lines or per unit area when using interface joints assigned to surfaces.
- Initial gaps are measured in units of length for translational freedoms and in radians for rotational freedoms.
- **Smooth Contact**. If an initial gap is used in a spring, then the positive local axis for this spring must go from node 1 to 2. If nodes 1 and 2 are coincident the relative displacement of the nodes in a local direction (d2- d1) must be negative to close an initial gap in that direction.
- **Frictional Contact** Friction contact joints can be specified to permit lift-off in a user-defined direction. If an initial gap is used in a spring, then the positive local x axis for this spring must go from node 1 to 2. If nodes 1 and 2 are coincident the relative displacement of the nodes in the local x direction (δx2- δx1) must be negative to close an initial gap.
- Both **Smooth Contact** and **Frictional Contact** joints can be used for lift-off or hook contact by using appropriate stiffnesses, gap and yield force.

Joint Model Force Response Curves

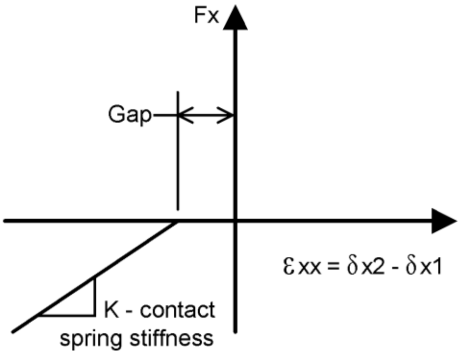
Elasto-Plastic Joint Models



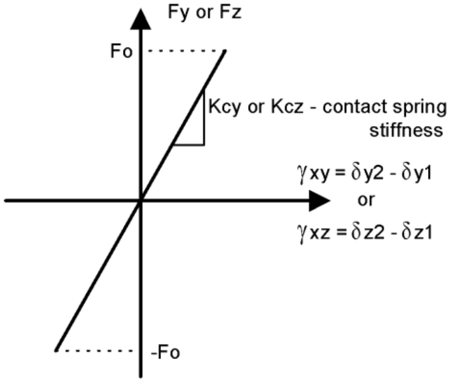
Smooth Contact



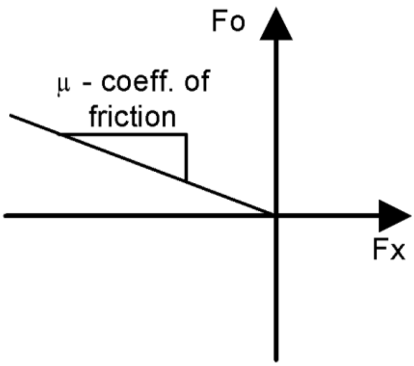
Frictional Contact 1



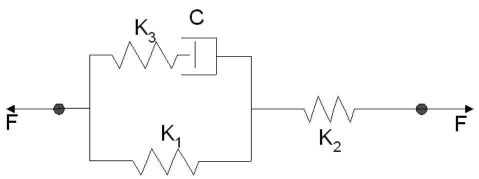
Frictional Contact 2



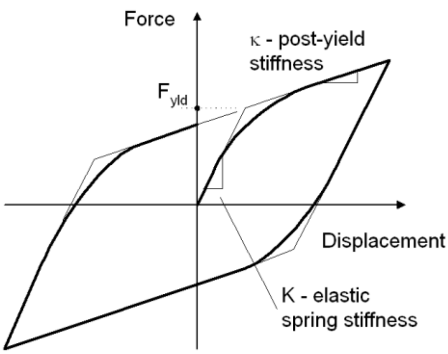
Frictional Contact 3



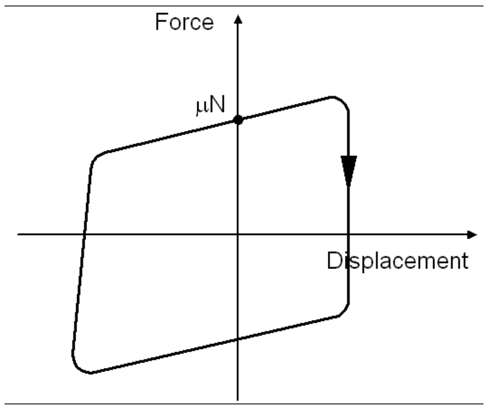
Viscoelastic Damper Joint Model



Lead Rubber Bearing Joint



Friction Pendulum System Joint



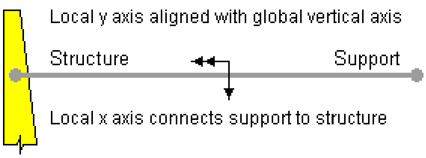
Tri-linear Earth Pressure Joint Material

Tri-linear earth pressure joint material simplifies the modelling of a variety of soil-structure interaction problems. It creates a piecewise linear joint material attribute that generates earth pressures in relation to deflections of the structure. Earth pressures increase with depth along the vertical axis. Multiple attributes can be defined to represent layers of soil or changes in properties due to the presence of water.

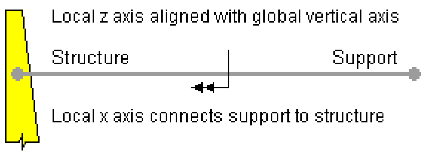
Defining Earth Pressure Joint Material

Tri-linear earth pressure joint material is defined using the **Attributes> Material> Joint** menu item. The input parameters and the number of degrees of freedom will vary according to the selected joint and model geometry.

- **Joint type** The Tri-linear soil joint material attribute must be compatible with the joint element to which it will be assigned. It is intended for use primarily with JNT3 and JNT4 elements, for 2D and 3D applications respectively. It is important that the local axes of the joint elements are orientated correctly for the intended use. For both 2D and 3D joint elements the joint element local x axis must align with the horizontal and connect the support to the structure. For 2D applications the joint element local y axis must be aligned to the vertical axis. For 3D applications the joint element local z axis must be aligned with the vertical axis. **Local coordinate systems** may be required to achieve correct element orientation.



Joint element axes required for 2D modelling applications



Joint element axes required for 3D modelling applications

- **Defining geometry**Joint material parameters can be defined for assignment to either a point, a line or a surface. The defining geometry refers to the underlying geometry to which the slave nodes of the joint elements are assigned. The defining geometry is used to determine the angle of the structure for lines and surfaces. Note that the angle of the structure cannot be determined when joint material attributes are applied between points.

Vertical and lateral pressure parameters

Parameters relating to vertical and lateral pressures are entered in tabular grids. Diagrams and equations are provided on the Trilinear Earth Pressure dialog to clarify the meaning of the material parameters.

Local axes representations

Because **joint mesh assignments** are made using master to slave assignments, and the joint element orientation may differ on each side of a line or surface representing a retaining structure, it is important to correctly specify what the positive x and y axis of each joint material attribute is representing. With reference to an observed element axes for an assigned joint element:

- ❑ The positive displacement of the local x axis of the joint element can be set to represent (mobilise) **Active pressures** or **Passive pressures**.
- ❑ The positive direction of the local y axis of the joint element (for 2D) or local z axis (for 3D) can be set to **Opposes gravity (points up)** or **Matches gravity (points down)**.
- ❑ **Consider angle of structure** The angle of the structure relative to the retained soil changes the effective direction of the soil pressures. Selecting this option (for lines and surfaces only) considers the orientation of the structure relative to the global vertical axis when resolving soil pressures. For lines, the angle of the structure is defined from the vector of the line. For surfaces, the angle is defined from the normal to the surface.

Notes

- All joints must be supported at the non-structure end by a fully fixed support condition.
- Modelling with tri-linear earth pressure material requires a nonlinear analysis.
- The pressure generated in the joint elements is interpolated from nodal displacement. The direction of the displacements determines from which part of the soil lateral response curve the forces are interpolated. The direction (either positive or negative) depends on the orientation of the local element axis representing the horizontal.
- By default soil pressures are defined per unit area. Where the material is applied to geometry other than surfaces a factor (W_{eff}) may be specified to define pressures over a fixed width or area. For example where the material is applied to a pile of fixed width, a factor may be specified to define pressures acting over a specified width rather than per unit width.
- When creating soil layers, the overburden pressure from any soil layers above must be calculated manually and entered as a surcharge for the material currently being defined.
- The material properties are assumed to vary with the vertical axis currently set in Modeller. The vertical axis can be set by selecting the **Utilities > Vertical Axis…** menu item.
- The material definition assumes soil depth and therefore pressure increases in a negative direction.

- The datum used for the calculation of effective overburden or ground level as entered into the grid must coincide with the geometry to which it is assigned. For example if the top layer of soil is defined by a line between (0, 0, 0) to (0, 0,-2) (Z is vertical) the datum level must be set to 0. For the layer beneath defined by a line between (0, 0,-2) to (0, 0,-8) the datum level must be set to -2, and a surcharge representing the above strata applied. If the model geometry is altered the soil definition values must also be updated to suit the new configuration.
- For all joint elements negative forces represent compression. Lateral pressures are therefore negative; positive forces are prevented by the defining variations (tension cut-off).
- The use of the tri-linear earth pressure material is described in the worked example Embedded Retaining Wall. See *Application Examples Manual (Bridge, Civil&Structural*

Editing of Earth Pressure Joint Material data

Earth pressure joint material attributes are held in the Attributes Treeview. Context menu entries named Edit Definition... and Edit Attribute... can be seen by right clicking on an attribute.

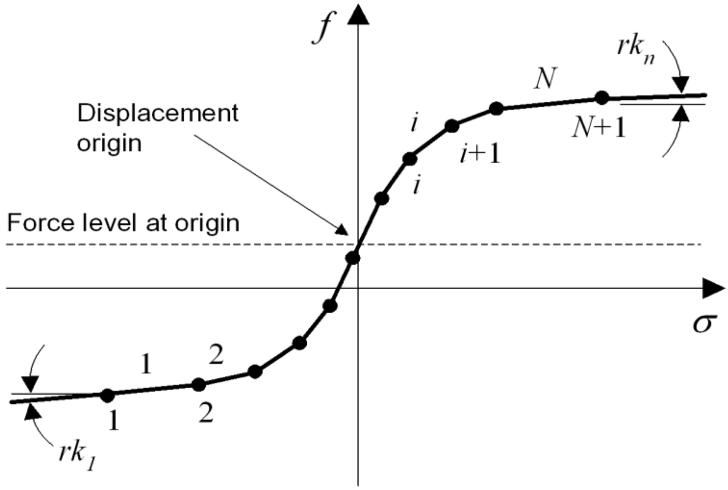
- Selecting the **Edit Definition...** menu entry or double clicking the attribute displays the original definition dialog with all the original input data intact.
- Selecting the **Edit Attribute...**menu entry displays the stored piecewise linear joint material values that are used to define the earth pressure joint material. These values may be changed outside of the Earth Pressure Material definition dialog but it is not recommended to do so. If edited, the link to the original definition dialog will be broken.

For a description of the piecewise linear joint material parameters that are required for this joint model please refer to the *Solver Reference Manual*.

Piecewise Linear Joint Material

This joint material is specified by a single Force/Displacement curve defined in a piecewise linear way, either by coordinate input or by spreadsheet import. Variations can also be used in the definition of curves. A stiffness factor governs how force/displacement values should be extrapolated when they fall outside the range of those defined. A factor of 1 (the default) keeps the same slope as the adjacent force/displacement line segment. A factor of 0 keeps the force the same.

A typical piecewise linear load-deflection curve with N linear segments is shown below.



Note

- It is possible to define segments with negative stiffness/gradient on the force-displacement curve to the positive side of the displacement origin. However, the curve cannot extend below the force level at this origin as the force-displacement relationship would then become non-unique because of the curve data defined (or extrapolated) for negative displacements. This is also true for forces on the negative side of the displacement origin; in this case the curve cannot extend above the force at the displacement origin.

Piecewise Linear Joint Material (Axial Force Dependent)

This joint material is similar to the generic piecewise linear joint material, but allows any number of Force/Displacement curves to be optionally specified by defining a number of curve segments for a stated axial force. Variations can also be used in the definition of curves. The joint is typically used for modelling plastic hinges in pushover analysis.

The curves defined are automatically interpolated by a LUSAS Solver to arrive at suitable force/displacement values for a particular axial force imposed upon a joint to which the piecewise linear joint material has been assigned. A stiffness factor governs how force/displacement values should be extrapolated when they fall outside the range of those defined. A factor of 1 (the default) keeps the same slope as the adjacent force/displacement line segment. A factor of 0 keeps the force the same.

Notes

- A force/displacement curve can be defined for an axial component axis (u) of an axial force dependent piecewise linear elastic material. If multiple curves defining the behaviour of the axial component are defined for various values of axial force, the behaviour between the singular points on each curve that matches the axial force it was defined for will be cross-interpolated between both bounding piecewise linear curves. This behaviour is complex, and it is therefore recommended that only a single curve is defined for the axial component (u).

- It is possible to define segments with negative stiffness/gradient on the force-displacement curve to the positive side of the displacement origin. However, the curve cannot extend below the force level at this origin as the force-displacement relationship would then become non-unique because of the curve data defined (or extrapolated) for negative displacements. This is also true for forces on the negative side of the displacement origin; in this case the curve cannot extend above the force at the displacement origin.

Support Conditions

Support conditions describe the way in which the model is supported or restrained and are defined from the **Attributes > Support** menu. Supports can be defined for Structural or Thermal analysis uses.

Structural supports

A summary of the structural support types is given here:

- **Free** For structural analyses the degree of freedom is completely free to move. For pore water analyses no flow will occur through the node. For thermal supports no heat will flow through the node.
- **Fixed** For structural analyses the degree of freedom is completely restrained from movement. For pore water analyses flow will occur through the node. For thermal supports heat will flow through node to maintain temperature at zero (or at a prescribed temperature).

Spring Stiffness Distribution

Spring stiffness is applied either uniformly, per unit length or per unit area. Spring stiffness values can be applied uniformly to all nodes meshed on the assigned feature, or their values may vary over a feature by applying a **variation**.

- **Stiffness** applies the spring stiffness uniformly to all nodes meshed on the assigned feature.
- **Stiffness/ unit length** and **Stiffness/unit area** distribute the total spring stiffness between the nodes on the assigned feature.

Simple lift-off supports

A simple lift-off support can be specified by pressing the **Lift-off** button and selecting the appropriate radio button for the degree of freedom that will control the lift-off condition for the support. For this a single value for the lift-off freedom needs to defined, where the lift-off value to be stated is the value that would be required to overcome the reaction or restraint at that support. This means that a positive lift-off value would be entered to overcome a negative reaction. Entering a negative lift-off control value effectively reverses the lift-off behaviour.

- ❑ A **Lift-off** value is set for either **Force**, **Moment**, **Hinge rotation**, or **Flow** for a chosen degree of freedom. Lift-off will take place once a stated value has been exceeded.

Additional behaviour can be specified for when lift-off has occurred:

- ❑ **Release restraint in lift-off direction only** releases only the specified restraint (controlling freedom), leaving all other restraints applied.
- ❑ **Release all restraints** releases all restraints.

Advanced lift-off supports and touchdown (contact)

More advanced lift-off behaviour can be modelled and specified using the **Contact** button. This provides for both lift-off and touch down and requires setting the appropriate contact conditions for the degree of freedom that will control the lift-off/contact behaviour.

- For lift-off (whether for all directions or the chosen direction), the support fixity part of the dialog describes the behaviour <u>before</u> the lift-off has occurred.
- For contact/touchdown, the support fixity part of the dialog describes the behaviour <u>after</u> the touchdown has occurred.

Contact control settings for Force, Moments or Flow

- ❑ Contact control can be set for either **Force**, **Moments**, or **Flow** values, and for a chosen degree of freedom. Upper and lower limits can be specified at which lift-off will occur. Contact will be deemed to be taking place once a stated value has been reached.

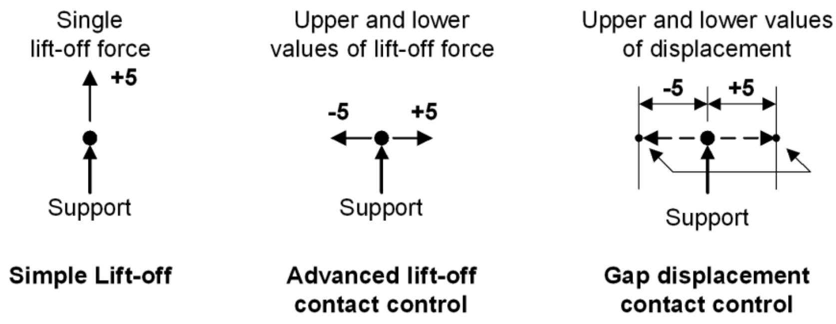
As for simple lift-off, additional behaviour can be specified for when lift-off or contact has occurred:

- ❑ **Release restraint in lift-off direction only** releases only the specified restraint, leaving all other restraints applied.
- ❑ **Release all restraints** releases all restraints.

Contact control settings for Displacements or Pore Pressure

Contact control for a **Displacement** or a **Pore pressure** based support condition is set using a **Gap** condition. For this, a supported node is free to move within an upper value and lower value. Touch down (contact) will occur when a limiting value is exceeded. This option would be used to restrict the movement of a pipe in a pipe support, or the movement of a bridge beam or deck supported by a displacement-restricted bridge bearing for example.

Lift-off / Contact examples



Notes

- The degrees of freedom which may be restrained for any analysis depend on the chosen element type. Those applicable for each element are defined in the *Element Reference Manual*.
- For structural problems Translation in X, Y, Z and Rotation about X, Y, Z refer to freedoms along and about the global axes unless a **local coordinate** is assigned in which cases the axis directions refer to local directions x, y, z.
- A hinge (loof) rotation is a local freedom which refers to rotation about the side of an element. Pore pressure is a special freedom type used in two phase elements.
- For thermal (field) problems there is only one freedom type, temperature (or the field variable).
- A support can only be set to have lift-off or contact properties, not both.
- If a particular freedom has been chosen as the freedom that controls lift-off / touchdown, it must either be either a restrained of spring freedom, it may not be 'free'.
- When loadcases that contain supports with lift-off or contact behaviour are used in load combinations or envelopes, a warning symbol is added to the load combination/envelope icon to signify that lift-off/contact supports are defined and that the principal of superposition is not true for these results. These combinations can be expanded to enable a nonlinear analysis to be carried out. See **Combinations and Envelopes** for details.

Additional settings for use with nonlinear analysis

Two system parameters ICNTCT and MXLFTO may be set to modify how the lift-off freedoms are handled in a nonlinear analysis. The parameters only affect lift-off freedoms which are lifting-off for the first time, that is, those that may have a bolt or anchor to break.

After the increment in which lift-off occurred has converged, a freedom may lift-off and re-ground without restriction.

- ❑ **ICNTCT =0** (Default value) Lift-off freedoms which lifted-off for the first time in the increment are not allowed to re-ground until the increment has converged. Once converged the initial fixity of the re-grounded freedom is restored and the increment is resolved
- ❑ **ICNTCT=1** No freedom is allowed to lift-off or re-ground for the first time until the increment has converged. On convergence the increment is resolved with the application of all pending lift-off/re-grounding conditions.
- ❑ **ICNTCT=2** No freedom is allowed to lift-off or re-ground until the increment is solved. On solution all pending freedoms may re-ground but only the freedom with the largest lift-off force is allowed to lift-off. The increment is then resolved.
- ❑ **ICNTCT=3** All freedoms are allowed to lift-off and re-ground at any point in the increment.

After a freedom has lifted-off for the first time, possibly breaking a restraining bolt or anchor, it is allowed to re-ground and lift-off but the lift-off force is set to zero in subsequent increments reflecting the fact that lift-off has in some way damaged the restraint. If all nodal freedoms were released on lift-off, these remain free regardless of future touch downs. ICNTCT can be set to 3 if there are no restraints to be broken.

- ❑ **MXLFTO** limits the number of times in an increment that the lift-off variables can be released or restored. It is used to prevent an infinite loop developing in which a node may repeatedly come into and then lose contact, a process known as chatter.

See **Model Properties** for details about setting these system parameters.

Thermal Supports

Thermal support conditions describe the heat flow at the model boundaries. Degrees of freedom can be made Free or Fixed. By default all freedoms are Free.

- ❑ **Free** no heat will flow through the node (insulated)
- ❑ **Fixed** heat will flow through the node to maintain temperature at zero (or at a prescribed temperature).

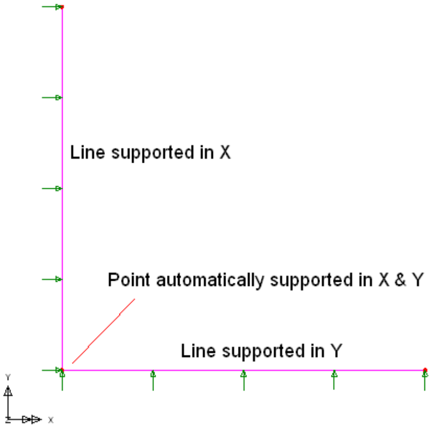
Assigning Support Conditions

Supports can be assigned to a model in a number of ways:

- By dragging a support attribute from the Attributes Treeview onto previously selected geometric features (or to **mesh objects** in a mesh-only model).
- By assigning a support type to all relevant features in a model by use of the **Assign to All** context menu available for the loading type.
- By copying and pasting a support type between Analysis entries in Analyses Treeview.

For linear static analyses all support conditions are assigned to the first loadcase. For a nonlinear or transient analysis, in which the support conditions change, the loadcase after which the support is applied is specified when carrying out the assignment.

Features that are supported automatically support common lower-order features. For instance,two lines supported in X and Y directions respectively, will automatically result in a common point being restrained in those directions also.



Support dependency across analyses / loadcases

- Supports defined for the first loadcase within a **base analysis** can be optionally inherited by all other analyses.
- For a linear analysis, supports assigned to the first loadcase of the analysis will be used by all following loadcases within that analysis. Supports cannot be changed for different loadcases within a linear analysis.
- Assigning a different support to selected features in a following analysis supercedes any inherited supports but for only those assigned features.
- For a nonlinear analysis, or a new time step of a transient problem, supports may only be reassigned on a new loadcase increment. All support assignments from previous increments or time steps that are not reassigned will remain unchanged.
- For an eigenvalue analysis support conditions may be omitted provided a shift is used in the **eigenvalue control**.

Notes

- Support assignments on lower order features override those on higher order features.
- On the common feature where support assignments meet, the support condition applied is additive.

- Assigning a local coordinate to a feature changes the freedom directions of the underlying element nodal freedoms and will hence also affect any global loads applied to that feature.
- Supports which act in tension, but not in compression, may be modelled using **joints or interface meshes**

Visualising Support Conditions

Support visualisation can be toggled on and off using the support visualisation button.

Support attributes can be visualised individually by selecting an attribute name in the Attributes Treeview and clicking the right mouse button to choose **Select Assignments** or **Visualise Assignments** from the context menu. This is the easiest way to interrogate the assignment of a single attribute.

Viewing support assignments

Loadcases and analyses to which attributes such as supports have been assigned can be viewed in a tabular form by selecting **View Assignments** from the context menu for the attribute.

Deassigning Supports

Supports may be deassigned from all or selected features by selecting the support attribute in the Attributes Treeview with the right hand mouse button and picking the **Deassign** entry from the context menu. The menu item entries **From Selection** or **From All** may then be chosen to deassign from the items in the current selection or from all the features in the model. Unassigned attributes will be denoted with a greyed-out bitmap .

Support attributes may also be defined by selecting **View Assignments** from the context menu for the attribute and unchecking the relevant loadcase.

Support Styles

The Attributes layer properties define the styles by which supports are visualised and also permits visualisation of supports on an individual basis. The attributes layer properties may be edited directly by double clicking the layer name in the Layers Treeview. Loading can be visualised in three ways:

Support conditions can be visualised in three ways:

- ❑ **Arrows** Visualises restraints as straight arrows representing translational freedoms, and circular arrows for rotational freedoms. Spring supports are visualised as spring representations. Hinge freedoms are not visualised. For supports defined using lift-off or contact criteria the controlling freedom is, by default, coloured orange with other support freedoms remaining the default green.
- ❑ **Symbols** Places a symbol on each supported node.

- ❑ **Codes** Writes a code next to each supported node representing the type of support assigned. The code uses F = Free, R = Restrained (Fixed), S = Spring. For example, a code RRSFFF represents a six degree of freedom node that is restrained in X and Y directions, supported with a spring in Z direction and free in all three rotational freedoms.

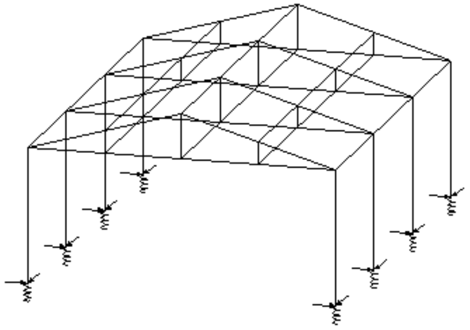
See **About Model Attributes** for more details.

Notes

- For nonlinear and transient problems, by default, supports are visualised for the active loadcase by combining the assignment in the loadcase history. To view the supports assigned to the active loadcase only, select the **Show only assignments in the active loadcase** option on the support visualisation dialog accessed from the attributes layer properties.
- Support visualisation may be drawn using the parent feature colour by selecting the **Colour support by geometry** option on the support visualisation dialog accessed from the attributes layer properties. This is useful to identifying which feature type a support attribute is assigned to.

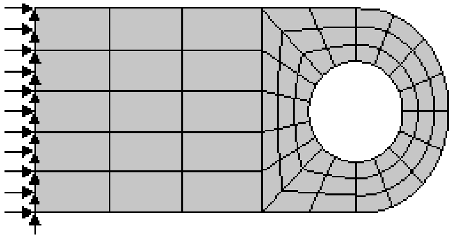
Example: Translational Fixed and Spring Supports

This 3D structure is restrained from any lateral movement at the base of all legs. The same points are also sprung vertically to represent a non-rigid base support.



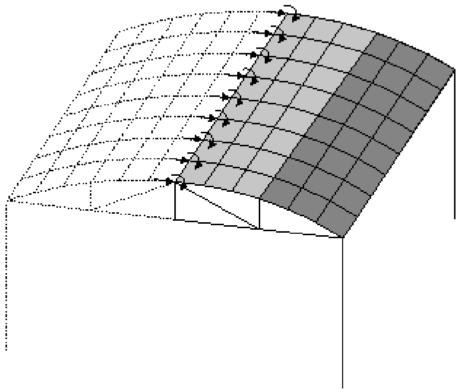
Example: Translational Supports

This 2D structure is restrained horizontally and vertically at the left edge with a single restraint in both in-plane translational directions. This rigidly fixes the body along the edge shown while allowing the rest of the model to move.



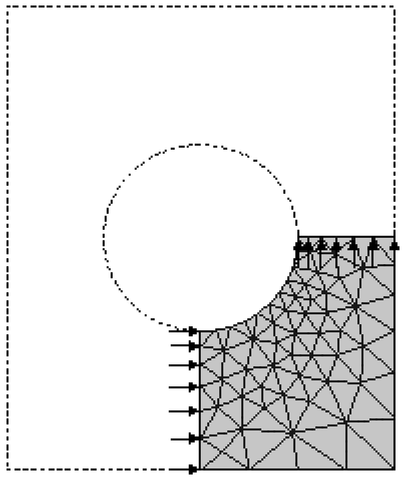
Example: Symmetry

Only the right half of this structure is modelled using shell elements but the full structure is represented by assuming symmetry at the centre-line. Symmetry assumes the same behaviour for both sides of the model therefore a translational restraint is applied to stop movement across the symmetry boundary and a rotational restraint is applied to force zero rotation at the boundary.



Example: Symmetry

This quarter plate membrane model uses symmetry restraints to effectively model the whole plate. Supports are positioned in order to prevent any movement at lines of symmetry.



Loading Attributes

Loading attributes describe the external influences to which the model is subjected. Structural and Thermal loading options are provided on separate **Attributes > Loading** menu items according to the user interface in use. A summary of the loading types is given here:

Structural Load Types

- **Structural Loads** are concentrated, body force, distributed, face, temperature, stress/strain, and beam loads.
- **Prescribed Structural Loads** are used to specify initial displacements, velocity or acceleration at a node.
- **Discrete Structural Loads** are used to distribute a given loading pattern (such as for a type of vehicle) over full or partial areas of the model, independent of the model geometry. Point and Patch loads are discrete loads - also known as general loads. Compound discrete loads permit sets of point and patch (and compound loads) to be defined.

Thermal Load Types

- **Thermal Loads** describe the temperature or heat input to a thermal analysis.

All Structural and Thermal loads are feature-based loads that are assigned to the model geometry (or to **mesh objects** in a mesh-only model) and are effective over the whole of the feature to which they are assigned. Discrete loads are feature independent. Further control over how discrete loads are applied is available by using a **Search area**. Once defined. loading attributes are added to the Attributes Treeview.

Assigning Loading

Loading can be assigned to a model in a number of ways:

- By dragging a loading attribute from the Attributes Treeview onto previously selected geometric features (or to **mesh objects** in a mesh-only model).
- By assigning a loading type to all relevant features in a model by use of the Assign to All context menu available for the loading type.
- By copying and pasting a loading type between Analysis entries in Analyses Treeview.

When a load is assigned, an analysis and a loadcase must be defined. A load factor may also be specified. If a load factor is entered the loadcase name will include this load factor. If the load is to be assigned to a new loadcase the new loadcase name may be entered into the loadcase combo and the new loadcase may be set active if required using the checkbox provided. Setting the loadcase as the active loadcase ensures that immediate load visualisation will take place. Additional options are available when applying discrete loads see **Assigning Discrete Loads** for more information.

Some loads act in global directions, others in local element directions. The defined loading value will be assigned as a constant value to all of the nodes/elements in the feature unless a **variation** is applied. Variations can be applied to all feature load types except for Beam Distributed loads that have a variation built into the definition.

Tip. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

Notes

- Load factors of assigned loads can be changed by selecting the **Change load factor** menu item accessed from the loading name context menu.
- In nonlinear and transient analysis feature-based loads can be factored using **load curves**.
- Consult the *Element Reference Manual* in order to check that the required loading is available for a particular element.

Gravity loading

Gravity loading can be applied on an analysis or loadcase by loadcase basis. Gravity loading is applied as a Body Force load type.

Loading dependency across analyses / loadcases

Unlike supports, loadings defined for one analysis or loadcase are not inherited by following analyses or loadcases. Loadings can however be copied and pasted between analyses in the Analyses Treeview.

Visualising Assigned Loading

Loading visualisation is done on a loadcase-by-loadcase basis requiring each loadcase to be set active in turn. Load visualisation can be toggled on and off using the load visualisation button.

Loading attributes can be visualised individually by selecting an attribute name in the Attributes Treeview and clicking the right mouse button to choose **Select Assignments** or **Visualise Assignments** from the context menu. This is the easiest way to interrogate the assignment of a single attribute.

Viewing loads by definition and by effect on the mesh is also possible via the load visualisation button's drop-down menu.

Viewing loading assignments

Loadcases and analyses to which attributes such as loading has been assigned can be viewed in a tabular form by selecting **View Assignments** from the context menu for the loading entry in the Attributes Treeview.

Styles

The Attributes layer properties define the styles by which loading is visualised and also permits visualisation of loadings on an individual basis. The Attributes layer properties may be edited directly by double clicking the layer name in the Layers Treeview. Loading can be visualised in three ways:

- **By colouring geometry** Colours the geometry according to which loading attributes are assigned to which features.
- **Using symbols** Marks the geometry with symbols according to which loading attributes are assigned to which features.
- **Using arrows** Visualises the loading attribute using arrows. For discrete loads the definition and the effect on the mesh may be different.

See **About Model Attributes** for more details.

Structural Loading

Structural loading is feature based and hence it is assigned to the model geometry (or to **mesh objects** in a mesh-only model). Variations in loading on a feature can be specified using a **Variation**. For information on which load types can be applied to which element types, see the *Element Reference Manual.*

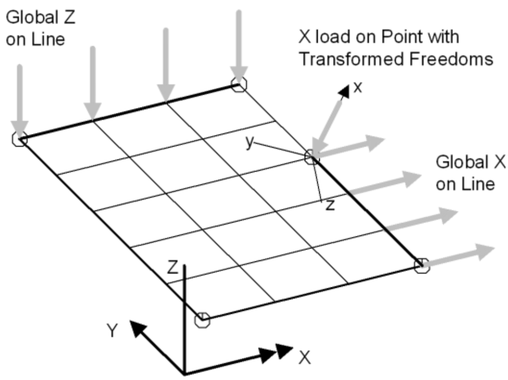
Structural loading is accessed via the **Attributes > Loading** or **Attributes > Loading > Structural** menu items depending upon the type of user interface in use.

Structural load types available are:

- **Concentrated Load**
- **Body Force**
- **Global Distributed Load**
- **Face Load**
- **Local Distributed Load**
- **Temperature Load**
- **Stress and Strain Loading**
- **Internal Beam Point Load**
- **Internal Beam Distributed Load**
- **Initial Velocity Load**
- **Initial Acceleration Load**

In the following headings the Solver name for the loading is showed in parentheses.

Concentrated Load (CL)



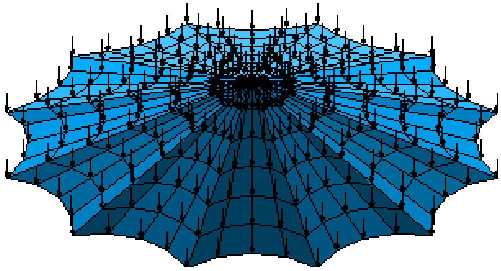
A **Concentrated Load** defines concentrated force and moment loads in global (or transformed) directions.

A Concentrated Load is applied per node of the underlying feature onto which the load attribute is assigned. A Concentrated Load is therefore normally only used to assign a load to a point as the total applied load would otherwise be dependent on the mesh density of each feature assigned to.

Concentrated loads are defined relative to the nodal coordinate system. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

Concentrated loads can be applied in cylindrical coordinates for Fourier elements by setting the option on the **Attributes** tab of the **Model Properties** dialog.

Body Force (CBF)



A **Body Force** defines an acceleration or force per unit volume loading in global directions. A typical example of body force loading is self weight, which requires the specification of gravitational acceleration and mass density (in the material properties).

By default, Body Forces define accelerations, but an option on the **Attributes** tab of the **Model Properties** dialog, can be set so that Body Forces define a force per unit volume.

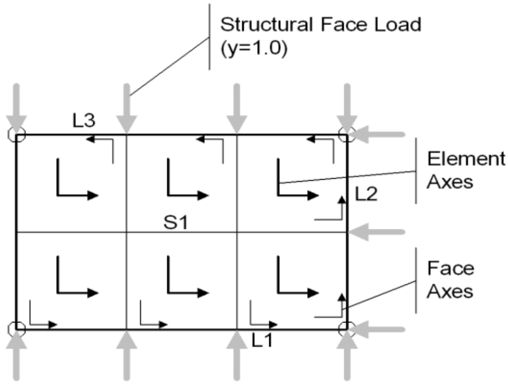
Note that gravity loading can be defined either by directly specifying a constant body force load or by defining its existence as a property of a structural loadcase.

Global Distributed Load (CL)

Defines concentrated force or moment loads in global (or transformed nodal) axis directions. Concentrated force loads are applied to all nodes underlying the feature onto which the load attribute is assigned. Nodal freedoms can be transformed using local coordinate sets. The following sub-types are supported:

- ❑ **Total** applies nodal load values calculated according to contributions from surrounding elements and to element nodal weighting values, e.g. loads are weighted with ratios 1:4:1 at nodes along the edge of a quadratic shell in such a way as to make the shell strain equally.
- ❑ **Line (per unit length)** applies nodal loads using the specified values per unit length loads. Must be assigned to Lines.
- ❑ **Surface (per unit area)** applies nodal loads using the specified values per unit area loads. Must be assigned to Surfaces.

Face Load (FLD)



Defines face traction values and normal loading applied in local element face directions. Face loads are applied to the edges of plane elements or the faces of solid elements. This type of loading is applicable to 2D and 3D continuum elements, and certain shell, membrane and thermal elements.

In the example shown, a local y direction structural face load is assigned to the Surface boundary Lines. Note the direction of the axes of the local element faces.

Where a loaded Line or Surface feature is common to two or more higher order features, it is possible to specify to which higher order feature elements the load is assigned.

See the *Element Reference Manual* for details of element face directions.

Local Distributed Load (UDL)

Defines a load per unit length or area for line or surface elements in the local element directions. Typically, local distributed loading is applied to beam elements and shell faces. An example of a local distributed load is internal pressure loading. For beam elements, when the element type permits, uniformly Distributed Load will be written to the LUSAS data file as Beam Element Loading (ELDS).

Temperature Load (TEMP, TMPE)

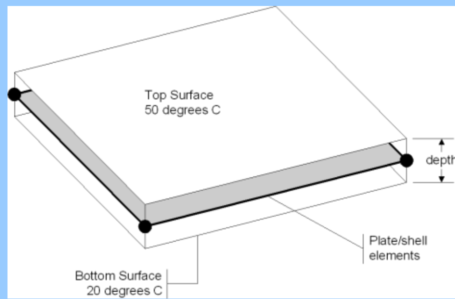
Nodal and **Element** temperatures define the LUSAS Solver TEMP and TMPE load types respectively. These loads apply temperature differences on a nodal and element basis. Temperature gradients in X, Y and Z directions may also be input. This load type can be used in conjunction with temperature dependent material properties to activate a different set of properties at a specified point in the analysis. The thermal expansion coefficient is normally set to zero in this case.

Notes

- Nodal temperatures apply to all elements connected to that node, except joints, in which temperature loading is invoked using Option 119.
- Elemental temperature is only applied to the node of the element specified.
- For step by step problems, the (initial) temperature values need only be specified on the first load step.

- The Temperature load may be used to provide a temperature field for computing initial material properties in a nonlinear analysis. To initialise the temperature field in a nonlinear field analysis, the temperature loading must be applied using a manual loading increment.

Case Study. Temperature Gradient Through Slab Thickness



Nodal and element temperature values accept gradient values for some element types. This gradient applies a differential thermal load across the top and bottom surfaces of a Surface element. The effect of this gradient is to cause bending in the structure. See the *Element Reference Manual* for temperature load input variations on an element basis.

In this example (which assumes no slab eccentricity) a 0.5m thick concrete slab is at 20 degrees Celcius. The top surface is subjected to a temperature of 50 degrees Celsius and the bottom surface remains at 20 degrees Celsius.

To model this enter the following on the structural temperature loading dialog:

- The final slab mid-surface temperature of **(50+20)/2** should be entered in the **Final temperature** field
- The temperature gradient through the slab of dT/dZ should be entered as **(50-20)/0.5** in the **Final Z temperature gradient** field
- The initial slab mid-surface temperature of **20** is entered in the **Initial temperature** field

When the analysis is run, LUSAS will multiply the temperature gradient by the thermal coefficient of expansion specified in the material property attribute to calculate the thermal bending strain.

This method assumes a linear temperature distribution through the depth of the slab. If a known nonlinear variation is required, solid elements must be used with a **variation** defining the nonlinear through-thickness behaviour.

Stress and Strain Load (SSI, SSR)

The input values that are required in order to define stress and strain loading for particular elements can be seen by selecting the either the element description or by entering the element name in the Stress and Strain loading dialog.

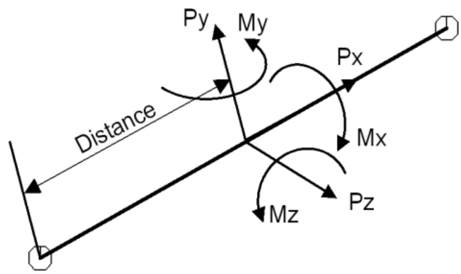
- ❑ **Initial** defines an element initial stress/strain state in local directions. Initial stresses and strains are applied as the first loadcase and subsequently included into the incremental solution scheme for nonlinear problems. Initial stresses and strains are only applicable to numerically integrated elements.

- **Residual** defines element residual stress levels in local directions. This can only be done for elements with a nonlinear capability. Residual stresses (unlike initial stresses) are assumed to be in equilibrium with the undeformed geometry and are not treated as a loadcase as such. They are considered as a starting position for stress for a nonlinear analysis. Failure to ensure that the residual stresses are in equilibrium will result in an incorrect solution. There is no concept of residual strains and therefore when the residual button is chosen a reduced number of components are presented.

Refer to the individual element descriptions in the *Element Reference Manual* for full details of the initial stress and strain, and residual stress components.

LUSAS Modeller will automatically write an appropriate initial stress and strain, or residual stress type to the datafile when a solve is requested. See the *Solver Manual* for more detailed information regarding the tabulation of initial stresses and strains, and residual stresses in LUSAS datafiles.

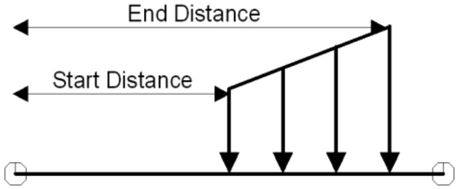
Internal Beam Point Load (ELDS)



The **Internal Beam Point** load is a point load applied to lines in the local or global direction. The distance to the point load from the start of the line may be defined as either a parametric distance (0 to 1 with respect to the line length) or an actual distance. Force and/or Moment components are defined for each point location. Several point loads may be defined in one load attribute if required.

See the *Element Reference Manual - Appendix A* for examples of internal beam loading (ELDS) types. Refer to each element's main listing to see if an element supports this loading type.

Internal Beam Distributed Load (ELDS)



The **Internal Beam Distributed** load is a distributed load that can be applied to lines in a model in a local, global or projected direction. The distance from the start of the line to the start of the distributed loading, and the distance from the start of the line to the end of the distributed loading are defined along with the load component and the start and end load values. For local and global loading the start and end distance may be defined as either an actual distance or a parametric (0 to 1) distance - both measured along the line. For projected loading only an actual distance measured along a global axis is permissible. Several distributed loads may be defined in one load attribute if required. As an example, trapezoidal loading would be defined as 3 separate load sets with a load varying from zero to a load value over an initial stated distance, then remaining at the same load level up until a second stated distance, and then varying from the same load value back to zero over from that point to a distance equal to the length of the line.

Note that for projected loading on a line not sitting in a primary plane the start and end points of each load set are based upon projecting the line back into either an XY, YZ, or ZX plane (depending upon the load component selected) with the start of the internal loading pattern being measured from the start of the line along the shortened plane length of line and not along the actual line length itself.

See the *Element Reference Manual - Appendix A* for examples of internal beam loading (ELDS) types. Refer to each element's main listing to see if an element supports this loading type.

Initial Velocity / Initial Acceleration Load (VELO/ACCE)

In dynamic analyses, velocities or accelerations at a nodal variable can be defined. These values can be used to specify an initial starting condition or they may be prescribed for the whole analysis. If values are to be prescribed throughout the analysis load curves must be used and the appropriate freedoms must be restrained.

Prescribed Structural Loads

Structural loading is feature based and hence it is assigned to the model geometry (or to **mesh objects** in a mesh-only model). Variations in loading on a feature can be specified using a **Variation**. For information on which load types can be applied to which element types, see the *Element Reference Manual*.

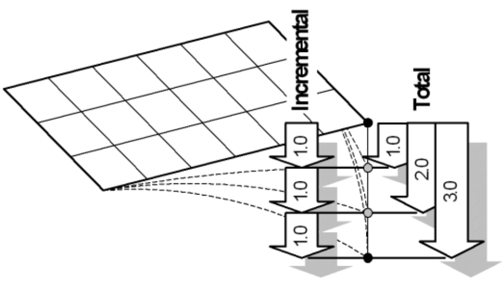
Prescribed Structural loading is accessed via the **Attributes** > **Loading** or **Attributes** > **Loading** > **Structural** or menu items depending upon the type of user interface in use.

Prescribed Structural load types available are:

- **Displacement**
- **Velocity**
- **Acceleration**

Prescribed Displacement Loading

A prescribed **Displacement** defines a nodal movement by either a **Total** or **Incremental** prescribed distance in global (or transformed) axis directions. Freedoms which are assigned a non-zero prescribed value will be restrained.



The adjacent example shows two methods of applying prescribed displacement. Incremental loading adds displacements to a previous increment, whereas Total requires the full displacement to be specified on each increment.

Notes

- For linear analyses with multiple loadcases an automatic restraint is only assigned if the prescribed displacement is applied in the first loadcase. If a prescribed displacement is not assigned in the first loadcase but is assigned in subsequent loadcases a restraint must be assigned manually.
- Prescribed displacements can be assigned and turned-off on a loadcase basis.
- Total and incremental prescribed displacements should not be used in the same analysis.
- It is recommended that total prescribed displacements are used with load curves.
- Prescribed rotations should be specified in radians.
- The assignment of a prescribed displacement of zero to a freedom of a node (in order to return it to an original undeformed position) causes a 'hidden' support to be created that remains active in all following loadcases, until it is superceded either by another prescribed displacement load, or by another support assigned to a following loadcase. A Prescribed Loading entry within an analysis loadcase in the Attributes Treeview effectively represents a support condition.

Prescribed Velocity and Acceleration Loading

In dynamic analyses, prescribed velocities and accelerations may be defined for any nodal variable. These values can be used to specify an initial starting condition or prescribed for the whole analysis.

- ❑ A prescribed, or initial **Velocity** defines a velocity loading in global (or transformed) directions.
- ❑ A prescribed, or initial **Acceleration** defines an acceleration loading in global (or transformed) directions. If acceleration loads are required, the density must be specified in the material properties. Initial accelerations are only valid for implicit dynamic analyses.

Notes

- If the values are to be prescribed throughout the analysis load curves must be used, see **Load Curve Definition**.
- Initial velocities and accelerations should only be applied to the first loadcase.
- In general, load curves should be used to prescribe velocities and accelerations in an analysis. However, initial values may be defined without using load curves if no other load type is controlled by a load curve.
- If velocities and accelerations are prescribed for the same variable at the same point in time in an analysis, the acceleration will overwrite the velocity and a warning will be

output. An exception to this rule occurs for implicit dynamic analyses where an initial velocity and acceleration may be used to define an initial condition for the same variable.

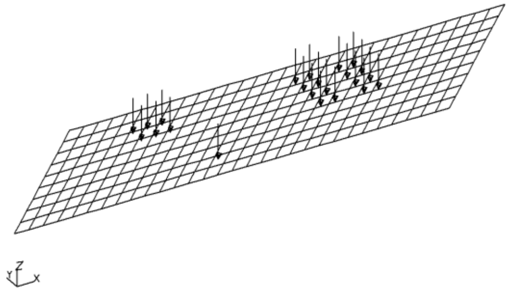
- If initial conditions are to be applied, refer to **Transient Dynamic Analysis** for details on how to compute the data input required for the appropriate integration scheme.

Discrete Structural Loads

Discrete structural loads are used to distribute a given loading pattern (such as for a type of vehicle) over full or partial areas of the model, independent of the model geometry. Discrete structural loads are defined in relation to their own local coordinate system, the origin of which is given by the coordinates of the Point feature to which the load is assigned. Note that discrete loads are always assigned to Points. Discrete structural loads differ from feature-based loads in that they are not limited to application over whole features. Discrete structural loads may be projected over an area, onto Lines or into Volumes. Separate discrete loads may be applied to a model as a set or load train using the Compound load option. Search areas can be used to control load assignment.

To identify critical vehicle loading patterns on bridges (using discrete point and patch loads) **vehicle load optimisation** is available for selected software products only. See *Application Manual (Bridge, Civil & Structural)* for details.

Discrete loads are useful for applying a load that does not correspond to the features underlying the mesh. A patch load may be spread or skewed across several features. LUSAS automatically calculates the nodal distribution of forces that is equivalent to the total patch load. This example shows a typical set of point loads assigned to a grillage model. A single point, a group of 6 and a group of 16 point loads are shown.



The coordinates of the vertices defining the patch are relative to the Point to which the patch load is assigned, i.e. a load definition is defined in a local coordinate system, the origin of which is given by the coordinates of the Point to which the load is assigned. The Point does not have to lie on the Surface to which the load will be applied as the patch load is projected in a specified direction.

Using Search Areas with Discrete Loads

A discrete load is distributed onto the elements over which the load lies. A **Search Area** is a way of controlling the load distribution onto these elements. If no search area is specified when assigning the load, then all of the underlying elements will be eligible for load distribution.

Notes

- While projecting the loads into the search area a check is made for multiple intersections of the load and the search area. Multiple intersections indicate an ambiguity in the location of the load. This ambiguity may be resolved with a more specific search area.
- The distribution of load to the nodes follows the shape functions of the particular element. In quadratic elements, this distribution can appear at first unlikely. For example, a unit positive load at the centre of an 8-noded quadratic element, results in negative 0.25 loads at the corners and positive 0.5 loads at the mid-side nodes.
- Search areas are automatically created and used by the **prestress wizards** to define the target to be loaded.

LUSAS generated discrete structural loads

Examples of discrete structural loads that are created automatically by LUSAS include patch and point loadings created by **vehicle load optimisation** software to represent vehicle and lane loading, and equivalent nodal loading defined as a result of using a **prestress wizard**. Discrete loads generated by LUSAS for these applications cannot be deleted or edited of them.

Discrete Structural Load Types

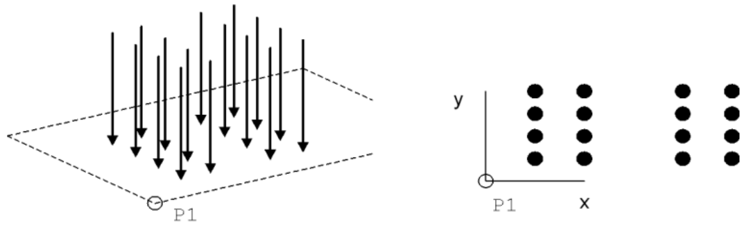
A discrete load consists of coordinates defining the local x, y and z position and a load intensity. Any Points selected prior to the Discrete loading dialog being displayed are entered as coordinates into the grid and are ordered to an expected point order where possible. Discrete load types available are:

- ❑ **Point load**
- ❑ **Patch load**
- ❑ **Compound load.**

Point Load

Point loads provide concentrated loading at specified locations. Point loads can be defined singly or as a set of discrete point loads in 3D space. Each individual load can have a separate load value. Point loads can be defined via Arbitrary input or by specifying a Grid input.

This example uses 16 distinct load values. The loads are applied to the model as distinct loads.



Patch Load

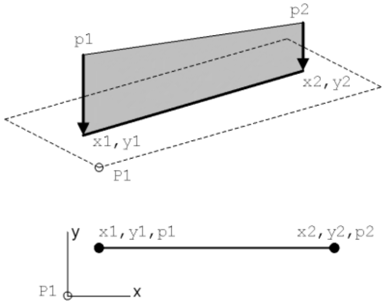
Patch loads provide a loading per unit length or per unit area. Loads can be defined in global or local untransformed load directions. Loading types are:

- **8 node patch** - used to define a curved sided quadrilateral
- **4 node patch**- used to define a straight sided quadrilateral
- **Multi-node patch** - used to define a series of straight sided patches of any number of points.
- **Straight line** - requires 2 specified coordinates to define a straight line load
- **Curve** - requires 3 specified coordinates to define a curved line load

The following examples show patch loads assigned to Point 1. Once assigned, the load origin is located at Point 1. The discrete load point order (P1, P2 etc) shows the order in which the coordinate definitions should be made.

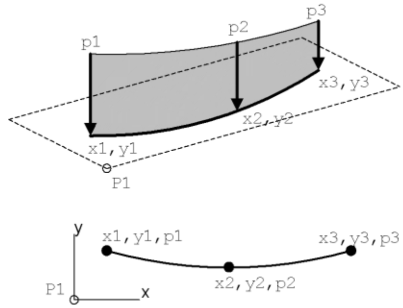
Example 1.

A straight line load defined using 2 coordinates.



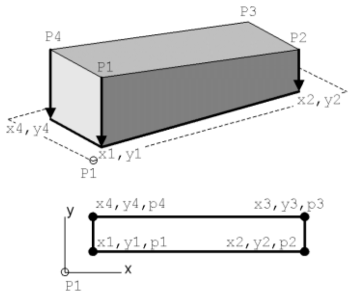
Example 2.

A curved line load is defined using 3 coordinates.



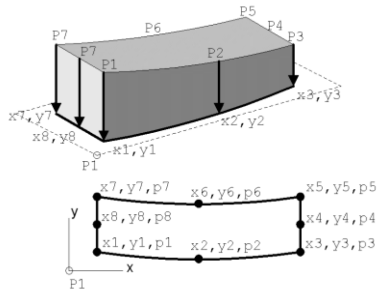
Example 3.

A straight sided quadrilateral defined using 4 coordinates.



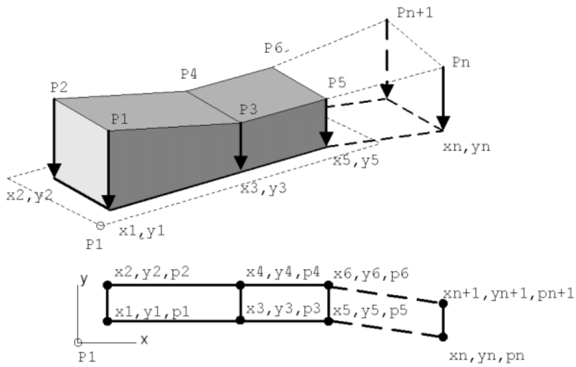
Example 4.

A curved sided quadrilateral defined using 8 coordinates.



Example 5.

Multi node patch loading.

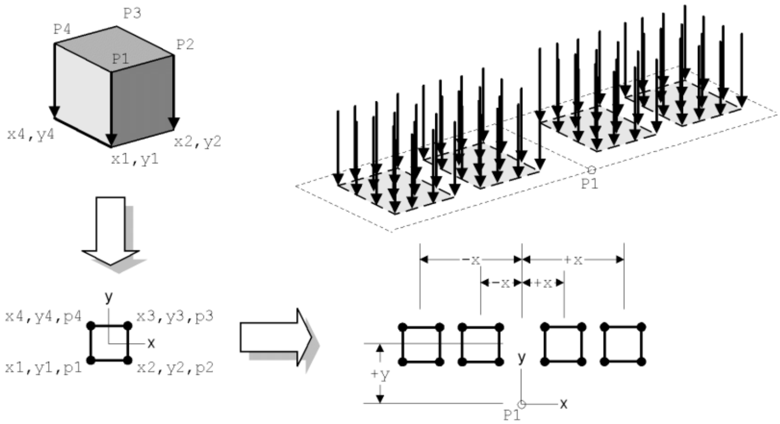


Notes

- The mid-side nodes for a curved line load and for a curved sided quadrilateral load must lie with +/- 10% of the overall distance between the corner nodes.
- Multi node patch loading is always straight sided. A minimum of 4 coordinate points and an even number of coordinate points overall must be defined.

Compound load

Compound loads may be created to simplify the definition and assignment of more complex load patterns. A compound load can be formed from a set of previously defined discrete loads and be subsequently assigned to a model as one loading. A compound load may be defined from any combination of existing point, patch and compound discrete loads. For example a discrete patch load representing a vehicle may be included in a compound load twice with the distance between the vehicles specified. x, y, z offsets and a translation can be specified to locate the compound loading from away from an assigned point. When created, compound discrete loads are held in the Loading section of the Attributes Treeview in their own section named Compound.



Single patch load Compound load

Defining Discrete Point and Patch Loads

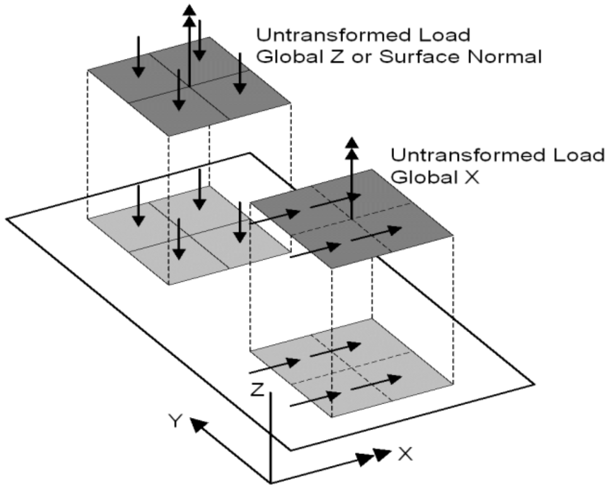
- **Arbitrary** permits the definition of any number of Point loads. If the points are defined in an expected grid order (see Grid entry), then the loading components can be optionally projected along the columns and rows of the patch load grid into a search area (if used), otherwise the loading is applied to the nearest load location in the search area.
- **Grid** adds the correct number of rows to the point load definition grid for the number of x and y values defined and expects the coordinate points to be defined in a particular order so that any loads that may overhang a search area (if used), can be optionally projected to be included into the search area. For a grid of points of x=3, y=2 the expected order of coordinate input is x1,y1, x2, y1, x3, y1, x1, y2, x2, y2, x3, y2.

Coordinates and magnitude

- For a **point** load each attribute defines multiple loads, one concentrated load at each given vertex.
- For **patch** loads the vertices combine to specify the shape of a line or patch load. The load is specified at each vertex allowing the load intensity to be varied. **Patch types** include 8 node, 4 node, multi node, straight line and curve.

Load direction

- **Load Direction** (an untransformed load direction) defines the direction of the loads in the patch before any transformation is carried out at the assignment stage. Options are: **X**, **Y**, **Z**, **XYZ**, **Patch x**, **Patch y**, or **Surface normal**. The Patch x and Patch y load directions apply a patch load in the local x or local y direction of the patch (and not the local coordinates of the assignment object.) Patch y is not available for the line patch type.



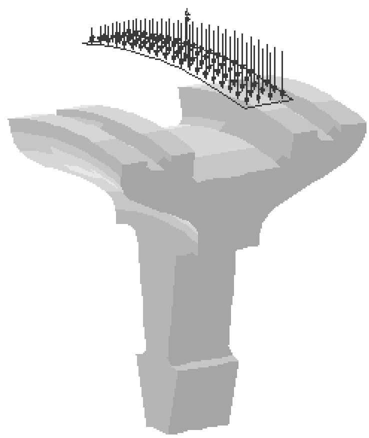
In the example shown, untransformed global loads are projected onto a model normal to the patch definition. The projection vector is denoted by a double-headed arrow on a visualised patch. The direction of the load applied to the model is defined using the untransformed load direction.

Use of a Patch x load direction in conjunction with any of the patch types would, for example, permit modelling of train or vehicle braking effects on straight or curved paths.

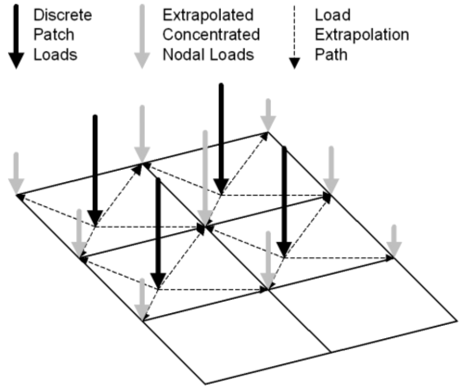
Projection Vector

- **Projection Vector** is used to work out which features are actually loaded. The vector is followed (in both directions) and any features intersected by the assigned discrete load vertices projected in the direction of the projection vector are loaded in the direction specified by the untransformed load direction. For patch loads defined by 4 or 8 vertices the projection vector is always perpendicular to the patch.

  This example shows a typical 3D patch load where the patch is defined in space and projected onto the model.



Patch load divisions



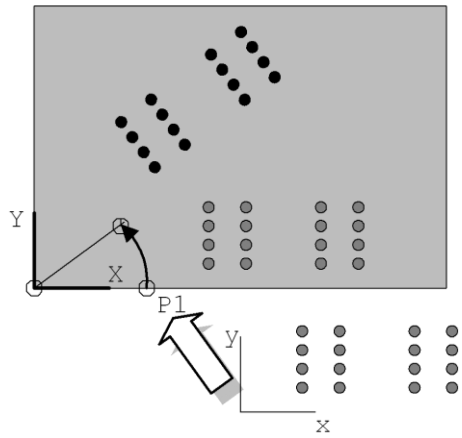
- **Patch load divisions** specifies the numbers of divisions in the local x and y directions of the patch being assigned. The divisions are used to split the applied patch into individual component loads before they are in turn used to calculate equivalent nodal loads on the model. By default, the patch load division are based upon the values set in the *Patch divisions* object which is created when a discrete loading is added to the Attributes Treeview. Again by default,10 patch load divisions are used in the local x direction and the aspect ratio of the patch is used to calculate the divisions in the y direction. When creating a patch ideally at least one division should be used per element division. The more individual loads a patch is split into, the more accurate the solution obtained. Patch load divisions can also be explicity defined on the main patch loading dialog as a number in x and a number in y. In this case if X=0 and Y=10 is entered the number in the x direction will be calculated proportionally to the patch shape. Equivalent weighting values are used to calculate the portion of each discrete load that is applied to each corner of the element that it lies within. The load is then applied as Concentrated Loads. These weighting values are based on element shape functions and may vary with element type.

Assigning Discrete Loads

Discrete loads are independent of features therefore their application can be more flexible. The load assignment parameters are explained below:

- **Patch Transformation** Discrete load patches can be mirrored, rotated, scaled and generally translated from their initial defined orientation during their assignment to a feature on a model. For example, a patch load may be skewed by applying a rotation transformation when assigning the load.

  In the example shown right the Point load defined about local xy axes is assigned to Point 1 subject to a patch direction transformation using a 30 degree xy rotation about the global origin. Note that the local origin of the patch load is rotated and repositioned as well as the patch itself.



  To rotate a patch about its centre, define the patch with its local origin at its centre. Note that static vehicle loadings (as supplied in the vehicle libraries) are generally defined with a loading origin at their centre.

- **Load transformation** Changes the load orientation from the (untransformed) direction given in the load definition. The transformation applies to the direction of the individual load components rather that to the patch as a whole. For example, it can be used to model braking loads on a 3D model that have horizontal and vertical components by specifying a transformation that will rotate the loads out of the vertical direction and into an inclined plane in the direction of vehicle travel.
- **Search area** A **search area** restricts loading to a specified portion of the model. If a search area is not specified, the load is projected onto the whole model. For 2D models it is usually acceptable to default to the whole model, but for 3D models where multiple intersections of the load projection onto the model may occur it is safer to restrict the loading to the required face using a search area. In either case the time taken to assemble the loads is significantly improved by using a search area to restrict the number of elements tested for intersection with the load. Search areas are automatically created and used by the **prestress wizards** to define the target to be loaded.

In addition, the discrete load can be specified to:

- **Project onto line** This option is used to project discrete loads onto 2D line beam structures and frame models. Discrete loading is applied to the beam as corresponding forces and moments along the beam.
- **Project over area** This option is used to project discrete loads over an area. The area may be defined by a grid of beam elements (a grillage), a plate or shell structure (slabs), or the face of a solid model.
- **Project into volumes** This option is used to project discrete loads into volumes (solid models) and is primarily for use with tendon loading.

Loads that extend beyond a search area can be included or excluded using:

- **Options for loads outside search area** Loads that fall outside the search area can be moved into the search area or be excluded entirely using a variety of options. See **Processing Loads Outside Search Area**.

The analysis in which the loading applies must be stated and general loadcase information that can be entered includes:

- **Loadcase** specifies in which loadcase the loading is to be applied. Loadcases can themselves be manipulated. See **Loadcase Management** for more details.
- **Load factor** specifies a factor by which the loading is multiplied before the equivalent nodal loads are calculated.

Editing of Discrete Loading Data

Editing of pre-defined discrete loading data (such as that used for supplied vehicle loads) allows users to view both the original vehicle definition input data, as well as the actual loading applied (the vehicle load converted into a discrete load format), for any and all vehicles within LUSAS. Editing of user-defined discrete loading data only permits viewing and editing of the discrete loading data.

So, for the case of creating a vehicle load from a pre-defined vehicle, the resulting attribute in the Attributes Treeview has context menu entries named Edit Definition... and Edit Attribute... These menus can be seen by right clicking on the attribute.

- Selecting the **Edit Definition...** menu entry or double clicking the attribute displays the original definition dialog with all the original input data intact. The user can change these inputs that may be either loading parameters such as width, length and intensity etc or even the type of vehicle, at any time. For each modification, the name of the attribute and the equivalent discrete load values are modified. Although the name of the attribute is altered, the attribute itself is merely modified and so the assignment links between the bridge structure and the load will not be lost.
- Selecting the **Edit Attribute...**menu entry displays the equivalent discrete loads. These values may be changed but this breaks the link to the original definition dialog and a warning message will be displayed.

Editing of automatically generated discrete loading data (such as that created by the use of the **prestress wizards**) is not permitted.

Editing patch load divisions

- When the first discrete load type is added to the Attributes Treeview, a *Patch divisions* object is also created. Double-clicking on this object displays a dialog which allows the type and default number of patch divisions used on discrete patch loading to be edited.

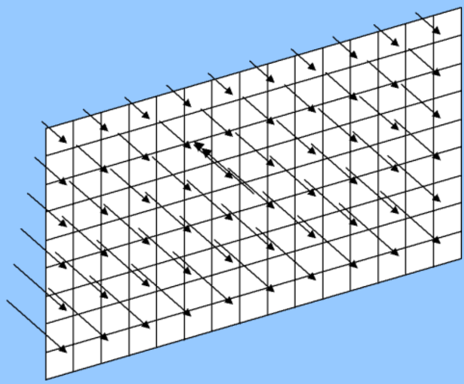
Case Study. Hydrostatic Loading

In this example, a discrete patch load will be used to apply a hydrostatic load to the side-wall of an underground box culvert.

1. The box culvert wall is defined using a Surface in the global XZ plane with corners at coordinates (0,0,0), (5,0,0), (5,0,3) and (0,0,3). Define a Surface using the **New Surface** button at the specified corner positions.
2. Rotate the view using the **Dynamic Rotate** button until the Surface can be visualised in 3D.
3. Using **Attributes> Mesh> Surface**, define a mesh using **Thick Shell**, **Quadrilateral**, **Linear** elements. Specify the spacing as **15** divisions in the local x and **9** divisions in the local y directions. Since only one Surface is present in the model, the divisions for the mesh can be entered directly onto the Surface mesh dialog.
4. With the cursor in normal mode, **assign** the mesh to the Surface by dragging the attribute from the Treeview onto the selected Surface.
5. To define a patch load that is coincident with the side-wall Surface, first select the four Points defining the Surface in the order they were defined. Choose the **Attributes> Loading** menu item and pick the **Discrete Patch** option. Note that LUSAS has selected a **4 node patch** and filled the Point coordinates into the dialog.
6. The load direction coincides with the global Y axis direction so select **Y** from the **Untransformed Load Direction**. Specify patch corner load intensity values of -3, -3, -1, -1 respectively.
7. The patch definition uses a coordinate system that is coincident to the global Cartesian axis system, so the load can be assigned to the Point at the origin (Point 1). **Assign** the load to Point 1 (0,0,0) leaving all dialog entries as default and press **OK** to assign the load. Note that the patch is drawn as discrete point loads. This is because the patch load is automatically split into point loads.
8. The number of discrete loads in each direction is dependent on the numbers of divisions entered in the Assign Loading dialog. In this case, the default number of divisions (10) is insufficient as there are insufficient loads to apply at least one load per element along the culvert. To improve the load application accuracy, deassign the load from the Point, and reassign using **15** divisions in the local X direction. Leave the Y divisions field blank. Note that LUSAS has automatically used the aspect ratio of the patch load to calculate a suitable number of divisions in local Y.

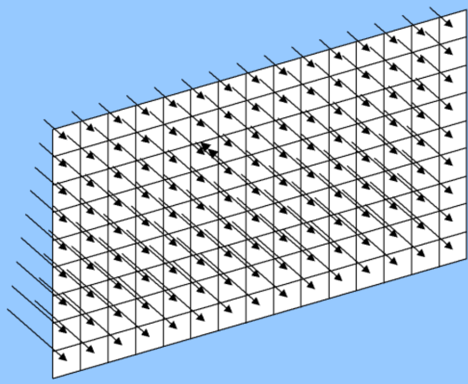
Hydrostatic Patch Load

Default number of divisions showing insufficient discrete point loads.



Hydrostatic Patch Load

Increased number of divisions on assignment. The double arrow vector indicating patch orientation.



Thermal Loading

Thermal loading is feature based and hence it is assigned to the model geometry (or to **mesh objects** in a mesh-only model). Variations in loading on a feature can be specified using a **Variation**. For information on which load types can be applied to which element types, see the *Element Reference Manual*.

Structural loading is accessed via the **Attributes** > **Loading** or **Attributes** > **Loading** > **Structural** or menu items depending upon the type of user interface in use.

Thermal load types available are:

- **Flux**
- **Distributed Flux**
- **Internal Heat generation**
- **Prescribed Temperature**
- **Environmental Temperature**
- **Internal Heat User**

Flux

- A **Flux** loading produces the LUSAS CL load type which in a field analysis applies a rate of internal heat generation (Q). Positive Q defines heat input.
- A total flux, a flux per unit length or a flux per unit area can be specified.
- Flux is defined relative to the nodal coordinate system. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

Distributed Flux

- ❑ A **Distributed Flux** loading produces the LUSAS FLD load type which in a field analysis applies a rate of flux.

Internal Heat Generation

Defines the internal heat generation for an element. Positive loading values indicate heat generation and negative values indicate heat loss.

- ❑ The temperature dependent internal heat loading (RIHG) defines the rate of internal heat generation. This load attribute requires a reference temperature for each set of properties.
- ❑ Defining temperature dependent properties turns a linear thermal field problem into a nonlinear thermal problem.

Notes

- **Load curves** can be used to maintain or increment the RIHG as a nonlinear analysis progresses.
- Automatic load incrementation under Nonlinear Control cannot be used with RIHG loading.

Prescribed Temperature

Defines a prescribed temperature for an element.

- ❑ The **Incremental** prescribed load type adds to any temperatures present from a previous increment.
- ❑ The **Total** prescribed load type defines the total temperature at a given node at a specified increment.

Environmental Temperature

Models external fluid temperature and associated convection and radiation heat transfer coefficients. If an element face does not have an environmental temperature assigned it is assumed to be perfectly insulated.

- ❑ The temperature dependent environmental temperature loading (TDET) models properties that vary with nodal temperature. This load attribute requires a reference temperature for each set of properties.
- ❑ Defining temperature dependent properties will turn a linear thermal field problem into a nonlinear thermal problem.

Notes

- The 'environmental temperature' is the temperature of the surrounding fluid to which the object is losing heat to (or gaining heat from).
- The 'convection heat transfer coefficient' is in units of 'power per unit area per unit temperature' (e.g. $J.s^{-1}.m^{-2}.K^{-1}$).
- The 'radiation heat transfer coefficient' is in units of 'power per unit area per unit temperature to the power four' (e.g. $J.s^{-1}.m^{-2}.K^{-4}$) and is determined by multiplying the emissivity of the material by the Stefan-Boltzmann constant.
- The 'reference temperature' is only required if more than one row of data is being entered for temperature-dependent properties. For example different convective and radiative coefficients could be specified at 0, 50 and 100°C. Linear interpolation will be used. If only one reference temperature is entered the others properties will be constant for all temperatures.
- If heat transfer coefficients vary on a specified face the values will be interpolated using the shape functions to the Gauss points.
- If a non-zero radiation heat transfer coefficient is specified, the problem is nonlinear and **Nonlinear Control** must be used.
- Load curves can be used to maintain or increment the environmental temperature as a nonlinear analysis progresses.
- Automatic load incrementation within the **Nonlinear Control** can be used to increment ENVT loading.

Internal Heat User

Allows user-defined input of internal heat generation for an element for use with user-written software programs. Values can be entered in multi-column format. Positive loading values indicate heat generation and negative values indicate heat loss.

Case Study. Temperature Dependent Loading

Temperature dependent **environmental** loading can be useful to model experimentally determined correlation for convective coefficients. For example, if the convective coefficient may be given by **C [deltat]** to the power one third where **C** is a constant, **deltat** is the temperature difference between the surface and the environment. To specify this loading in LUSAS you would define the convective coefficient at as many reference points as are required to give a good piece-wise linear approximation of the function. Each reference temperature point is defined in a loading attribute and collectively these attributes define a single loading table. The loading table is then assigned to the features as required.

1. Define a row of Surface features.
2. Use an incremental prescribed loading to fix the temperatures at one end of the model.
3. Define a convective coefficient function using environmental loading (temperature dependent).
4. Assign the loading and solve.
5. Since the problem is one-dimensional the solution may be checked to ensure that the convection coefficient has been correctly interpolated.

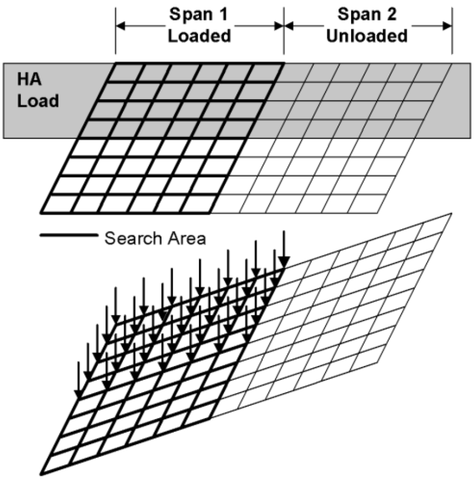
Search Areas

Search areas may be used to restrict the area of application of **discrete loads** (point and patch). This is useful for several reasons:

- **Improved Control of Load Application.** The search area will effectively limit the area over which the load is applied so that the effect of loads on certain features may be removed from the analysis. For 3D models it is possible that a chosen projected direction will cross a model in several locations. A search area is therefore used to limit the application of load to one of these multiple intersections. Restricting the area of application of discrete loads allows the same load attributes to be used to apply loads to different parts of the model.
- **Speed Improvement** the speed of calculation of equivalent nodal loads will be increased by cutting down the number of features considered in the calculation.

Search areas are automatically created and used by the **prestress wizards** to define the target to be loaded.

In the example shown, a multiple span grillage structure is defined with Span 1 as the search area. A discrete Patch load, indicated by the grey shaded region in the upper diagram, is applied across the whole structure, Span 1 and 2. The area of the structure coinciding with both the Search Area and the patch load will take the load as shown in the lower diagram.



Tip. Search areas should be used if the model is three dimensional and discrete loads are applied, as, for example, for box-section or cellular construction decks.

Defining and Assigning Search Areas

Search areas are defined from the **Attributes** menu then **assigned** to the required Lines or Surfaces. Control of loads lying **outside** the search area is available when the load is assigned, see **Assigning Discrete Loads**. If a search area is not specified when the load is assigned, all of the highest order features, excluding volumes, in the model will be used as a default search area. Valid search area configurations are shown below.

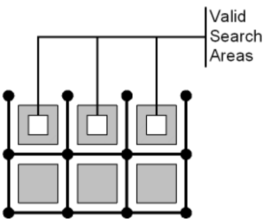
Note: The default maximum number of elements that can be used with search areas per grillage bay (each four-sided framing of a section of slab) is 30. In the unlikely event that a higher number is required this can be changed by setting a user-defined option in Modeller. Contact LUSAS technical support if you wish to do this.

Rules for Creating Search Areas for Grillages

The following general guidelines should be noted when assigning a search area to a grillage model.

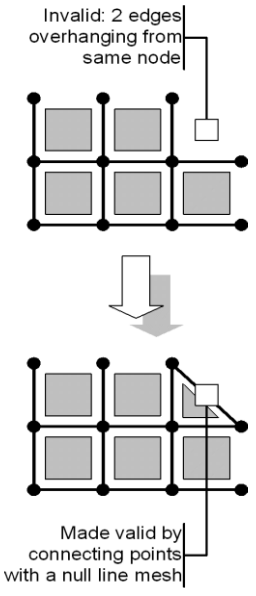
Overhanging elements defined such that only one side of a cell is missing are included in the search area, as shown.

In these cases LUSAS automatically 'closes' the cell.

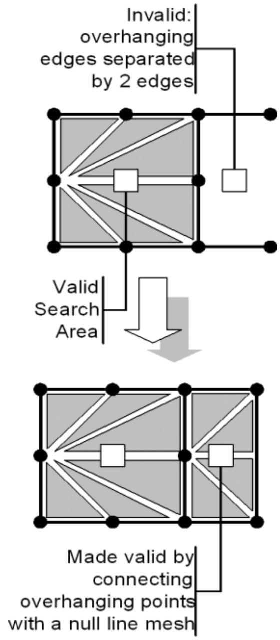


Elements cannot be included in the search area when they overhang from the same node, as shown.

In this case, a dummy Line can be added manually between the 2 overhanging points to close the bay to make the search area valid. When closing the bay in this way note that a single null-line mesh should be used having one mesh division.

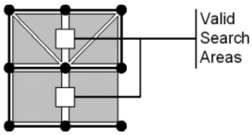


Cells of more than four edges are automatically subdivided into triangles, but overhanging elements are only included if divided by no more than one edge.



In this case, a dummy Line can be added manually between the 2 overhanging points to close the bay to make the search area valid. When closing the bay in this way note that a single null-line mesh should be used having one mesh division.

For the invalid region shown, a dummy Line can be added manually between the 2 overhanging points to close the bay and make the search area valid. When closing the bay in this way note that a single null-line element, having one mesh division should be assigned. If done, the resulting cell of five edges will be subdivided into 4 triangles.



There is no limit to the number of edges that may hang over the main body if the overhanging members are only separated by one edge (right).

In these cases LUSAS automatically 'closes' the cell and either sub-divides the resulting cells into triangles or uses a quadrilateral as appropriate.

Processing Loads Outside a Search Area

For **point** and **patch** loads any load outside a search area can either be excluded from the search area, or be projected to be included into the search area.

Discrete loads on lines

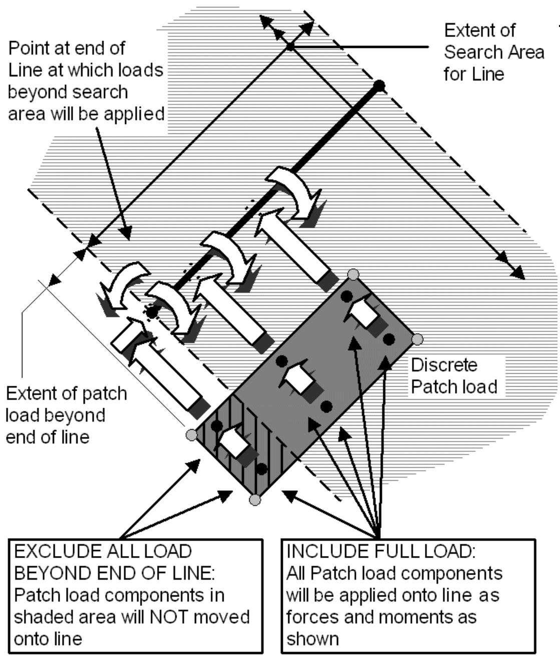
When discrete loads of patch or point loading are assigned onto Lines with an assigned Search Area the individual load components are projected onto the line, normal to the local x axis of the Line, and their effective loading is calculated and applied to the line as forces and moments along the line.

Options available when assigning onto Lines are:

- **Exclude All Load beyond the end of the Line (default)**patch load components beyond the end of line will be disregarded and all load components within the search area will be applied to the line with an appropriate force and moment to represent the positions of the loads.

- ❑ **Include Full Load** all load components within the search area will be applied to theline with an appropriate force and moment to represent the positions of the loads. Patch load components beyond the end of line will be applied to the point at the end of the line with an appropriate force and moment to represent the actual position of the loads.

Note that when a search area is assigned to a line the search area extends for the length of the line and for an infinite distance perpendicular to the line direction. See the diagram that follows or details.



Patch Load onto Line

Patch loads not lying on a line but within an assigned search area will be applied to the line as effective forces and moments. Loads outside the search area can be either included or excluded. If included, the applied moments will be computed by using the actual location of the defined loads

Discrete loads over areas

When discrete patch loads are assigned over an area the projection path(s) is/are defined by the **local x and y axes** of the loading patch. Each patch load component is 'moved' along a specified local x or y direction and added to the first loading positions found inside the patch in that projected direction. See the diagram that follows which illustrate the various options.

Options available when assigning onto areas are:

- ❑ **Exclude All Load (default)**
- ❑ **Include Local X Projected Load**
- ❑ **Include Local Y Projected Load**
- ❑ **Include Local X and Y Projected Loads**
- ❑ **Include Non-Projected Load**

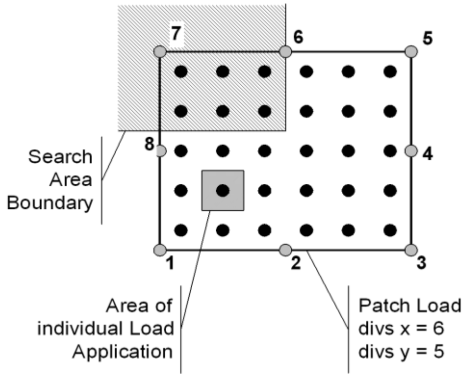
- ❑ **Include Full Local X Load**
- ❑ **Include Full Local Y Load**
- ❑ **Include Full Load**

Notes

- Loads will not be moved to the edge of the search area if the entire patch load lies outside the search area.
- Loads inside the search area are not moved.
- Discrete patch loads assigned over areas are not work equivalent as the discrete points are simply lumped at the nearest node.
- Patch loads outside the search area are lumped onto the nearest edge of the search area.

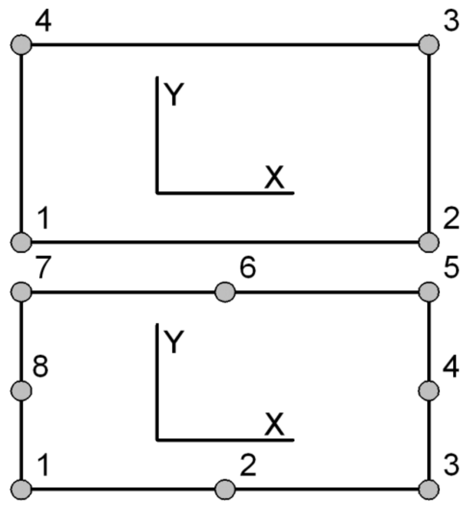
Patch Load Divisions

The number of patch load divisions in local x (div x) and y (div y) are specified at load assignment. The load intensity is then split into individual load components with an associated area of application.



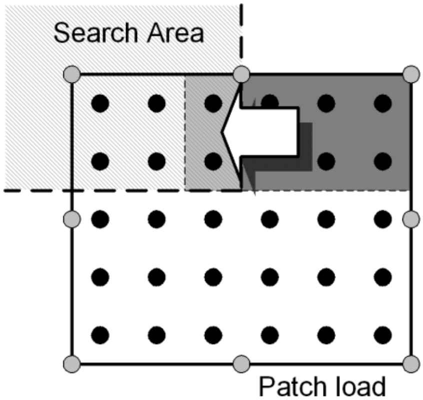
Patch Load Local Coordinates

The local coordinate set is dependent on the order in which the coordinates of the patch vertices are defined.



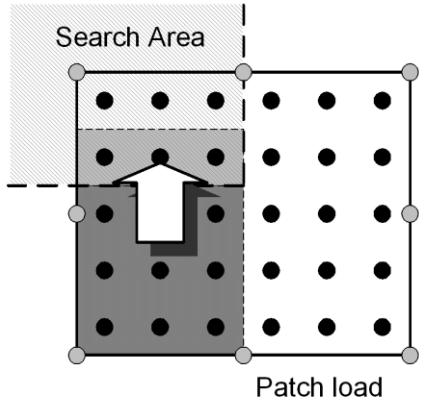
Local X Projected Load

Loads in the **local y** projected region (dark area) are lumped at nearest loading positions within the search area (light area).



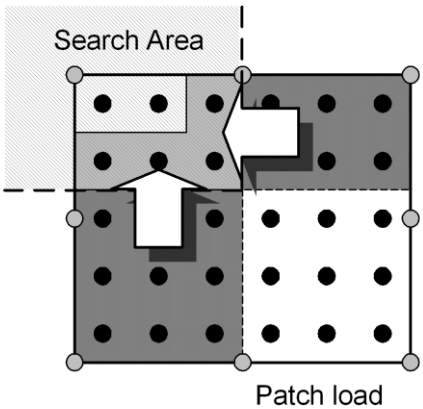
Local Y Projected Load

Loads in the **local y** projected region (dark area) are lumped at nearest loading positions within the search area (light area).



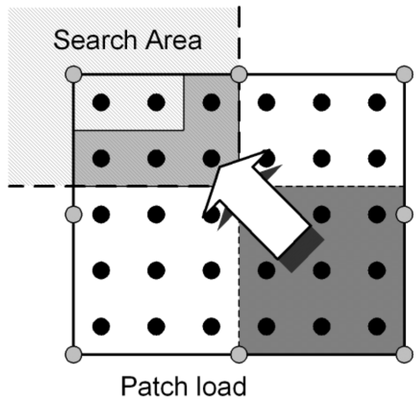
Local X and Y Projected Loads

Loads in the **local x and y** projected regions (dark area) are lumped at nearest loading positions within the search area (light area).



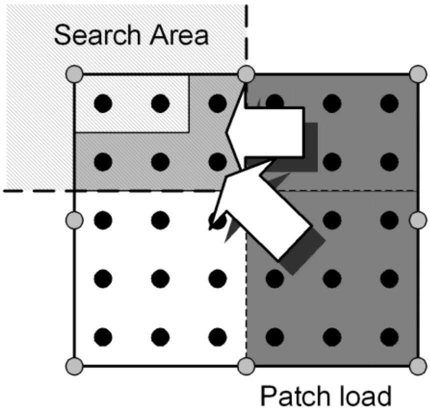
Non-Projected Load

Loads **not** in the **local x and y** projected regions (dark area) are lumped at nearest loading positions within the search area (light area).



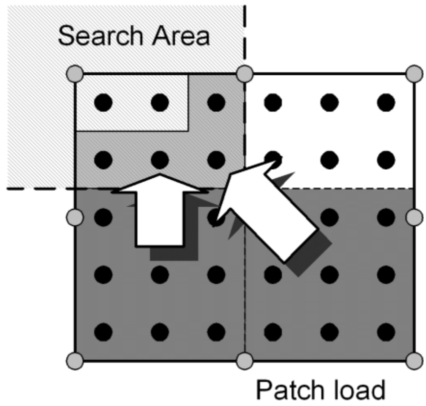
Full Local X Load

Loads in the **full local x** region of the patch (dark area) are lumped at nearest loading positions within the search area (light area).



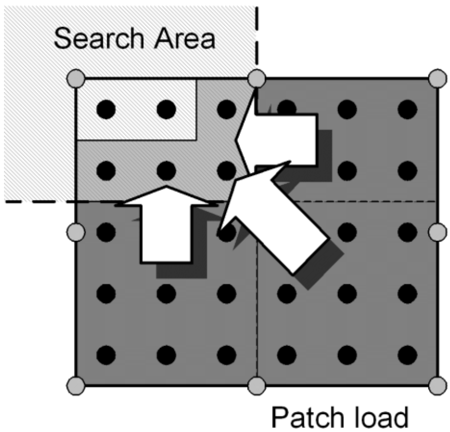
Full Local Y Load

Loads in the **full local y** region of the patch (dark area) are lumped at nearest loading positions within the search area (light area).



Full Load

All patch loads lying outside the search area (dark area) are lumped at nearest loading positions within the search area (light area).



Discrete point loads over areas

When discrete point loads are defined by specifying a Grid of points they can either be excluded from the search area, or be projected to be included into the search area in exactly the same way as for discrete patch loads. The expected coordinate input for a grid of points defined in this way for a grid of x=3, y=2 is x1,y1, x2, y1, x3, y1, x1, y2, x2, y2, x3, y2. If discrete point loads are defined by the Arbitrary option, and if the points are defined the expected grid format, then the loading is applied as per the patch loading (that is, the loading components can be projected along the columns and rows of the patch load grid into the search area), otherwise the loading is applied to the nearest load location in the search area.

Discrete point loads into volumes

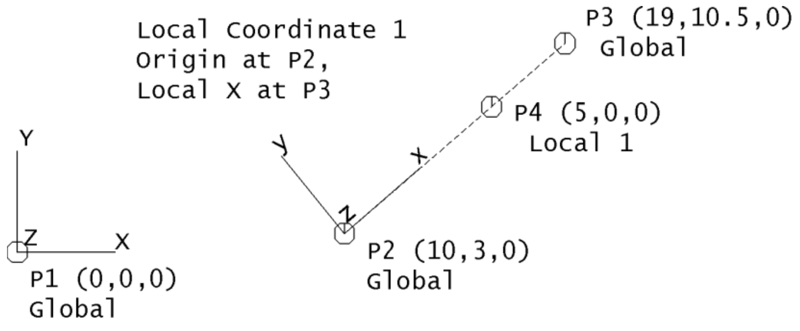
When discrete point loads are projected into volumes (by being assigned to a particular point on, or within the volume) the applied discrete loads are extrapolated within the elements to create equivalent concentrated nodal loads. When search areas are assigned to volumes the following options are available:

- **Exclude All Load** (default) Patch load components outside the volume will be disregarded and all load components within the search area will be extrapolated within the elements to create equivalent concentrated nodal loads
- **Include Full Load** Patch load components outside the search area will be applied to the nearest elements in the volume.

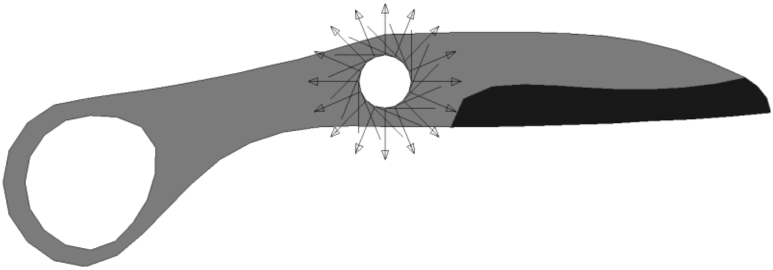
Local Coordinates

Local Coordinates define coordinate systems that differ from the default global Cartesian system. They have several uses:

- **Geometry Definition** Geometry features may be defined in a local coordinate system by set the chosen local coordinate active. When a local coordinate is active, all dialog entries relating to global X, Y and Z coordinate input use the transformed axis set as a basis for input.



- **Transforming Nodal Freedoms** When assigned to features the effect is to transform the degrees of freedom of the underlying element nodes. This has the effect of transforming the directions of applied global load and support conditions. In the example below, global freedoms are transformed to radial directions by assigning a cylindrical coordinate to the Lines around the hole. This method of transforming nodal freedoms is only valid for small deflections, since the freedom directions are not updated during analysis.



- **Materials** A local coordinate may be used to align orthotropic and anisotropic materials.
- **Variations** Variations may be defined using functions in terms of a local coordinate.
- **Composites** A local coordinate may be used to align composite attributes when they are assigned to the model.
- Element Orientation **A local coordinate may be used at the mesh assignment stage to orient beam and joint elements.**
- **Results Transformation** Results can be output relative to a local coordinate. For example, this is useful when looking at results on elements when the axes are not consistent.

Defining Local Coordinates

Local coordinates are defined from the **Attributes> Local Coordinate** menu item by specifying the local coordinate type and, for Cartesian, cylindrical and spherical types an **origin** and either a **rotation about a global plane** or a **rotation matrix.**

- **Rotation about a global plane** specifying angular rotations about the global planes, XY, YZ or XZ. When defining coordinate systems using this method, the local x axis is oriented parallel to the global X axis and rotated into position using the specified angle in the specified plane.
- **Rotation matrix** specifying a direction cosine matrix. A **Rotation matrix may be defined from selected Points** by first selecting 3 Points (1st Point defines the origin, 2nd Point defines the positive direction of the local x axis, 3rd Point defines the local xy plane) and clicking the **Use** button.

Notes

- Defining a new coordinate set does not automatically make it the active set, see Visualising Local Coordinates below.
- Local coordinate set types cannot be modified so, for example, a Cartesian set can not be changed to a cylindrical or spherical set.
- Local cylindrical coordinates defined by matrix are always defined with the local z axis along the cylinder.

Local Coordinate Types

Cartesian, cylindrical and spherical local coordinates are defined by indicating three positions in space defining a local xy plane (origin, x axis, xy plane). The type of coordinate chosen will dictate how the axes are defined.

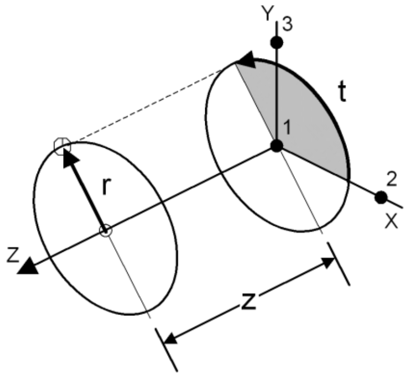
- ❑ **Cartesian** - Based on standard x, y and z coordinates arbitrarily oriented in space.
- ❑ **Cylindrical** - Based on the axes of a cylinder defined by a radius, angle and distance along the cylinder axis.

  - For a local cylindrical coordinate defined along the z axis a point is specified as (r, theta, z), where:

    r is the radius perpendicular to the local z axis

    theta is the angle in degrees measured from the positive x direction of the local xy plane, clockwise about the local z axis when looking in the positive z direction

    z is the distance along the z axis.



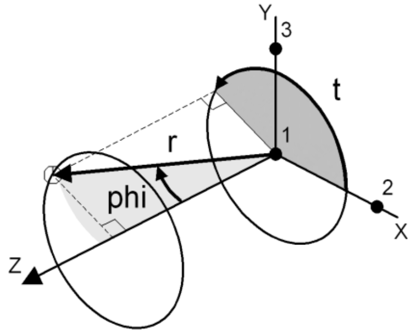
- ❑ **Spherical** Based on the axes of a sphere defined by a radius, tangential angle and angle around a meridian.

  Coordinates of a point are specified as (r, t, phi), where:

    r is the radius of the sphere on which the point lies from the local origin

    t is the angle in degrees measured from the positive x direction of the local xz plane, clockwise about the local z axis when looking in the positive z direction

    phi is the angle in degrees measured from the positive z axis to the radius line



Warning. There is no equivalent spherical coordinate set in Solver, therefore freedoms cannot be transformed using this type of local coordinate system.

- ❑ **Surface** local coordinate systems define a local axes which has the x and y axes in the plane of the surface and a local z axis normal to the surface. This is useful for extruding a volume normal to a surface and assigning supports normal to a surface. Surface local coordinates cannot be set active.

Visualising Local Coordinates

The active local coordinate system is defined from the Model Properties Geometry tab or from the local coordinate attribute context menu using the **Set Active** menu item. A black dot is shown next to a local coordinate attribute to indicate it is active.

By default the active coordinate is visualised on the graphics area, this can be switched off from the View properties. Click on the View Axes tab to change the view axes settings.

Local coordinates assigned to features may be **visualised** in the same way that other attributes are visualised.

Composites

Composite attributes allow previously defined materials to be collected together to define a laminate or composite lay-up. Layup definition methods allow for properties to be defined manually for use on solid and shell models and optionally include additional specific values for draping over model surfaces. Layup data can also be imported from a FiberSIM XML file, also for draping over selected model surfaces.

Composite Layup Methods

The following methods are available as a result of selecting the **Attributes > Composite** menu item:

- **Solids and Shells**
- **Draped Solids and Shells**
- **FiberSIM Solids and Shells**
- **Simulayt Solids and Shells**

Solids and Shells

This method allows a manual definition of the composite lay-up where orientations and thicknesses for the plies can be specified by stacking layers of differing materials at various angles and thickness. The orientation angles can be applied with respect to the local element x-axis (in the x-y plane) or with respect to the x-axis of a predefined Cartesian set. The z-axis defines the direction of the lay-up with ply 1 located at the bottom of the stack. The lamina thickness specified depends upon the element types used.

Notes

- Only orthotropic plane stress (for semi-loof shell) or orthotropic solid (for thick shell) materials can be used for structural shell composite lay-ups. Structural solid composites models must use the orthotropic solid material model and thermal solid composites models must use the orthotropic solid field material model. Isotropic materials may be used within any composite lay-up.

- For shell elements an appropriate plane stress nonlinear material model may be used whilst for solid elements a 3D nonlinear continuum model may be used (see the *Element Reference Manual*).
- The lay-up sequence is from bottom to top. In the case of a shell this will be in the direction of the Surface normal. In the case of a solid this will be in the direction of the local z.
- In cases where surface normals need correcting good use can be made of the cycling facility, where feature local axes can be cycled relative to a reference feature to ensure a consistent set of composite material axes.
- Composite attributes may not include materials that contain variations.

Draped Solids and Shells

This method makes use of the native draping functionality in LUSAS. A start point (which should lie inside or on the boundary of the surface to be draped) can be defined for each ply and the start direction is defined by the x-axis of a predefined Cartesian set. Prior to assigning a composite attribute of this type to a model a **draping surface** must be selected or specified.

The orientations of fibres following the drape are computed by LUSAS and are tabulated with respect to the x-axis of the local element axes. As with the Solids and Shells option, it is essential that the z-axis of the volumes to which the composite is to be assigned are consistently oriented. See Draping below.

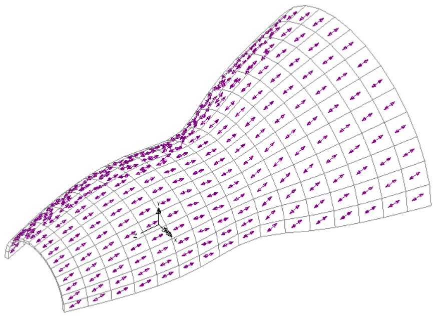
FiberSIM Solids and Shells and Simulayt Solids and Shells

Composite stack details can be read in from an external FiberSIM XMLfile or Simulayt LAYUP file. A default fibre volume fraction can be specified and by default it is assumed that all plies are of the same thickness but this can be modified. It is not necessary to select a draping surface or to define a start point when using this option.

Notes

- The coordinates of the ply data in the FiberSIM or Simulayt file must coincide with the coordinates of the drape surface.
- Any XML file should only contain lay-up data relating to a single drape surface. For example, if a non-composite core is sandwiched between two composite skins then at least two XML files will be required. The volumes defining each skin must be selected in turn and the appropriate XML file assigned to it.
- A draping grid can be extended by one grid row to ensure the edges of the component are fully enclosed. See **Extending the draping grid.**

Draping in General



Composite attributes may be orientated on a solid or shell model by specifying a start point and a **local coordinate** defining the drape direction of each lamina

Draping assumes the thickness remains constant and hence the volume fraction (the proportion of the volume in a lamina that is fibre with the remaining portion being the resin) is adjusted when the fabric is distorted. After assignment of a composite attribute that contains draping data to a model, the skew angle and fibre volume fraction may be contoured and the fibre orientations may be visualised.

Notes

- The native draping functionality in LUSAS is controlled by **Draping options**.accessed via the File > Model Properties menu item (Solution tab). FibreSIM and Simulayt draping options can be specified at the xml file import stage (accessed via the Attributes > Composite menu item).
- The Volume xy axes control the local element axes which must lie in the plane of the composite lamina. The local element axes may be visualised from the Mesh properties dialog.
- The local coordinate defining the drape direction must lie in the xy plane of the drape Surface at the start point.

Defining Composite Layups

Composite attributes require composite materials to be defined prior to defining composite layups. Composite attributes consist of a number of named layers where each layer contains specified material properties, and for certain element types, the angle of fibres and layer thickness. Composite layers can be defined using a Normal or a Grid method. Once composite attributes have been defined, they are assigned to the model on a feature basis.

For Solids and Shells and Draped Solids and Shells composites definition the procedure described below can be used to define a layup. For FiberSim Draped Solids and Shells definition the composite stack will already have been created using a default material and volume fraction for all laminae. If required, for this case, the Normal and Grid Tabs can be used to modify details for selected laminae.

Procedure

The procedure to define a composite layup using the **Normal** Tab is described, ending with details of how the Grid Tab can also be used to check or add layer data.

1. Define the Layup

Click on the **New** button to define a new layer. Enter a unique lamina name, select a composite material,and enter thickness and layup angle values. Note that a lay-up sequence is defined from bottom to top. The name may given a suitable prefix in the box provided. Click the **OK** button.

2. Enter details for the next lamina.

3. Repeat this process for each layer as required.

If a symmetric layup sequence is to be defined check the **Symmetric** button. This duplicates and reverses the layup sequence previously entered. The **Reverse** button is used to upturn the defined stack so the uppermost layer becomes the bottom layer. The **Insert** button can be used to add layers between existing layers.

4. Check the Layup sequence

There are two ways to check the composite layup sequence:

i) Select the Visualise button to display a representation of the defined composite layup. If desired this image can be annotated to the screen by clicking on the Create Annotation button from the visualise dialog.

ii) Select the Grid tab to display the layer properties in grid format. Data may also be created or edited using this option. Pressing the Tab key with the cursor sitting in the last row and cell of the grid creates a new row populated with the same data as the previous row. A right mouse click in a row opens a context menu that allows rows to be inserted or deleted.

Defining Lamina Thicknesses

The definition of lamina thickness depends upon the element types and model type used.

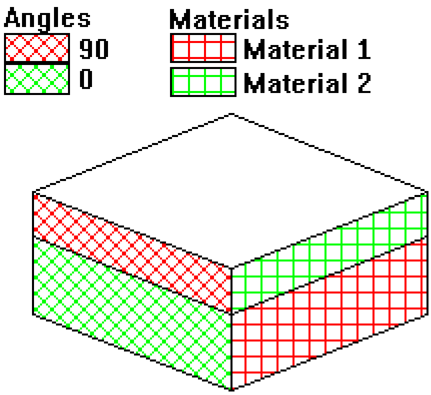
- The lamina thicknesses for shell models that have been assigned a geometric thickness are relative, not absolute, and represent the proportion of the total thickness (as specified by geometric surface properties) apportioned to each lamina.
- When assigning a draped layup to a shell model the assignment of geometric thickness properties to the shell is not always of use. In these cases, if a geometric property is not assigned to the model then the thickness of the assigned laminates is used to calculate the corresponding element (and hence geometric) thickness at any point. So, in this case the lamina thicknesses would be absolute values.
- The lamina thicknesses for solid models comprised of pentahedral and hexahedral composite elements are relative, not absolute, and represent the proportion of the total space that the elements of the volume represent apportioned to each lamina. For these models the number of laminate layers must correspond to, or exceed, the number of

elements through the Volume. Element nodal positions will be moved to correspond with laminate boundary positions if the node/laminate layer positions do not coincide.

- The lamina thicknesses for solid models comprised of tetrahedral composite elements are absolute values and represent the actual thickness of each lamina. For these models the total of all lamina thicknesses as measured from a tooling surface must exceed the space occupied by the tetrahedral elements.
- For a 'mesh-only' model (a model that has been created by importing a LUSAS datafile) the actual thickness of each lamina would be entered, so in this case the lamina thickness is absolute.

Visualisation of Composite Layup

The orientations and thicknesses for each lamina can be viewed by clicking on the **Visualise** button of the Composites dialog for a particular chosen entry method. This will display a layered representation of the composite stack with annotations. This representation may be used to create a bitmap annotation by clicking on the **Create Annotation** button.



Assigning Composite Properties

The method of assigning composite properties to a model differs according to the type of composite definition method used:

For the Solids and Shells definition method:

- The composite attribute created by this method is **assigned** to selected surfaces or volumes of a model by specifying the overall composite orientation. Options for orientation are: Local Coordinate, Local Element Axes, Axes From Surface. An angle of zero degrees aligns the laminate axis with the x axis from the orientation axes.

For the Draped Solids and Shells definition method:

- For solid and shell models the composite attribute needs to be assigned to a draping surface. This is done by selecting and placing the Surfaces defining the drape surface into **selection memory** and assigning the composite attribute to the model.

For the FiberSIM Solids and Shells and Simulayt Solids and Shells definition methods:

- No assignment to a draping surface is required because the layup data is already included and correctly positioned in the FibreSim or Simulayt XML files. However, the composite attribute must be assigned to the model to enable visualisation of other composite model data.

Visualisation of Composites Properties

To visualise assigned composites properties the surface or volume must be meshed. The following composite properties can be visualised:

- ❑ **Fibre (ply) directions**
- ❑ **Draping grid**
- ❑ **Lamina thickness, Skew angle, Offset layer, Fibre volume fraction**

Visualisation of Fibre (Ply) Directions

Once assigned to features which have a mesh assigned, the fibre directions of assigned composite data can be examined graphically as follows:

1. Right-click on the **Attributes** entry in the Treeview and select **Properties**.
2. On the Composite tab, click on **Settings** and select **Visualise ply directions**. The x and y axes define the warp and weft directions respectively; the item x&y displays both directions at the same time.

Lamina directions can be plotted as an x, y, z or x&y axes at any layer. For solids the axes may be placed at the top/bottom or middle of the chosen layer, for shells the axes are placed on the mid surface of the shell element. For details of how to choose a composite layer see **Setting The Active Composite Layer**.

Visualisation of Draping Grid

The draping grid for individual lamina can be examined graphically as follows:

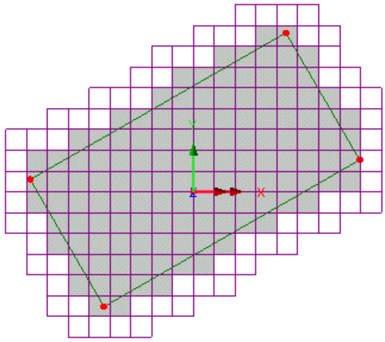
1. Right-click on the **Attributes** entry in the Treeview and select **Properties**.
2. On the Composite tab, click on **Settings** and select **Visualise ply directions**. Then select **Draping grid**.

If no mesh has been assigned to a model prior to selecting this option only the draping grid (and not the ply directions) will be visualised.

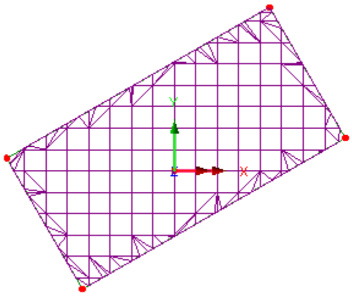
For details of how to choose a composite layer see **Setting The Active Composite Layer**.

Extending the draping grid

LUSAS Draped Solids and Shells grids are automatically trimmed at Surface boundaries. FiberSIM and Simulayt generated grids are not. If required, the draping grid can be extended by one grid row to ensure the edges of the component are fully enclosed by the draping grid. For FiberSIM and Simulayt grids this is specified at the file import stage (accessed via the Attributes > Composite menu item). For LUSAS Draped Solids and Shells grids, this is done via the Draping options on the **Model Properties** dialog.



Example of draping grid being extended by one row



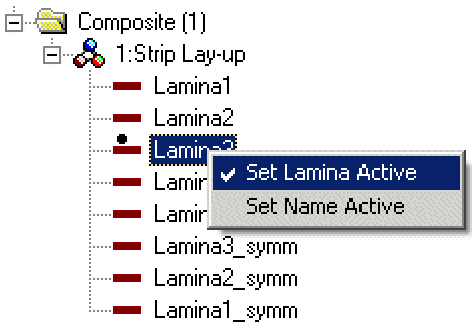
Example of draping grid being trimmed to a surface boundary

Visualisation of Other Composite Model Data

To view other composite model data such as lamina thickness, skew angles, offset layers and fibre volume fractions:

1. With a Contours layer in the Treeview right-click on **Contours**.
2. On the **Contour Results** tab select the **Composites (model)** entity and select the composite attribute name in the right-hand panel. The fibre volume fraction and skew angle (and any other composite modelling options relevant for the model) will appear for selection in the Component combo box.

Setting the Active Composite Layer



Composite shell and solid elements have multiple layers (laminae) of different materials though their thickness. The lamina or lamina name on which results or orientation axes are to be viewed is chosen by setting that lamina active. A lamina is set active by selecting the lamina with the right-hand mouse button from the Treeview and picking **Set Lamina Active** or **Set Name Active** from the context menu. A black dot next to a lamina indicates the active lamina..

If a lamina is set active only results or orientation axes for that lamina in that composite attribute will be displayed. If the lamina name is set active, results or orientation axes will be displayed for all laminae with that name across all composite attributes.

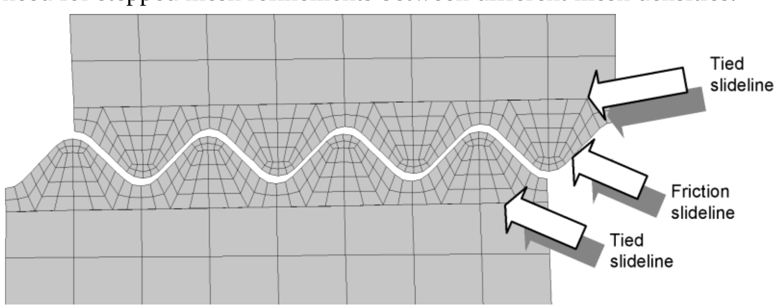
When viewing results the material transformation should be used to display stresses on or off axis.

Slidelines

Slidelines are attributes which can be used to model contact and impact problems, or to tie dissimilar meshes together. They can be used as an alternative to joint elements or constraint equations, and have advantages when there is no prior knowledge of the contact point. Slideline applications range from projectile impact, vehicle crash worthiness, the containment of failed components such as turbine blades, to interference fits, rock joints and bolt/plate connections. Slidelines can be defined to apply to all analyses that are present in the Analysis treeview, or be applied on a loadcase by loadcase basis for individual analyses.

Slideline properties are defined from the **Attributes**> **Slideline** menu item.

The properties of a slideline are used to model the contact interaction between master and slave features, such as the contact stiffness, friction coefficient, temperature dependency etc. The figure below shows a contact application in which a frictional slideline is defined between two bodies and where tied slidelines are used to join dissimilar meshes. The latter avoids the need for stepped mesh refinements between different mesh densities.



Slidelines are assigned to pairs or groups of features in a model, with one pair/group termed master and the second pair/group termed slave. An element face that lies on a slideline is called a slideline segment.

When one slideline surface is much stiffer than the other it can be defined as a rigid slideline surface. This approximation can improve the convergence rate and hence reduce the solution time. If the rigid surface is not part of the model, rigid elements should be assigned to the features.

Note. To use any type of slideline a nonlinear licence is required. However, tied-slidelines may be used in a linear loadcase.

Table of Elements for use with Slidelines

The following table gives a list of elements valid for use with slidelines:

Element type	LUSAS elements
Thick shells	TTS3, QTS4
Plane stress continuum	TPM3, TPM3E, TPK6, TPM6, QPM4, QPM4E, QPM4M, QPK8, QPM8
Plane strain continuum	TNK6, TPN3, TPN3E, TPN6, QNK8, QPN4, QPN4E, QPN4L, QPN4M, QPN8
Axisymmetric solid continuum	TAX3, TAX3E, TAX6, TXK6, QAX4, QAX4E, QAX4L, QAX4M, QAX8, QXK8,
Solid continuum	TH4, TH4E, TH10, TH10K, PN6, PN6E, PN6L, PN12, PN12L, PN15, PN15K, PN15L, HX8, HX8E, HX8L, HX8M, HX16, HX16L, HX20, HX20K, HX20L
Continuum two-phase	TH10P, TPN6P, PN12P, PN15P, HX16P, HX20P, QPN8P
2D rigid surface	R2D2
3D rigid surface	R3D3, R3D4

Slideline Types

There are several different types of slideline:

- **Null** The slideline attribute is ignored. Useful for performing a preliminary check on the model.
- **No Friction** Used to model contact without friction.
- **Friction** Used to model contact with friction.
- **Tied** Used to tie different meshes together.
- **Sliding** Used for problems where surfaces are kept in contact but which are free to slide relative to each other. The sliding behaviour is frictionless.

The friction/no-friction slideline types model the finite relative deformation of contacting bodies in two or three dimensions where the contact is stationary or sliding, constant or intermittent. The sliding only option is similar but does not permit intermittent contact, i.e. the surfaces are kept in contact, allowing frictionless sliding contact without lift-off to be modelled. The tied slideline option allows meshes of differing degrees of refinement to be connected without the need of a transition zone between the meshes.

Slideline Properties

- **Master/slave stiffness scale** Controls the amount of inter penetration between the two surfaces. Increasing the scale factor will decrease the amount of penetration but may cause **ill-conditioning**. Recommended values are:
  - Implicit/static solution **1.0**
  - Explicit solution **0.1**
  - Tied slidelines **100** to **1000**

  Slideline stiffnesses are automatically scaled at the beginning of an analysis if the average master/slave stiffnesses differ by a factor greater than 100. This is to account for contact between bodies that have significantly different material properties. This facility can be suppressed via File > Model properties > Attributes and selecting 'Suppress initial slide-surface stiffness check'.
- **Coulomb friction coefficient** Defines the coefficient of friction between contacting bodies for Coulomb's law. Only applicable for friction slidelines..
- **Zonal contact detection parameter** This defines the region around a node within which a search for contact is conducted. The size of the region is a factor of the size of the overall model – the model is projected onto the global x, y and z axes and the largest projection is used as a reference. For further information refer to the *Theory Manual*.
  - The default value of the zonal contact detection parameter is 0.01, i.e. 1% of the model size. A smaller value may result in undetected inter-penetration. The value should be set to 1.0 if the contact search should consider the entire model (though only points on the adjacent slideline surface will be considered valid contacts).
- **Slideline extension** A boundary of a slideline segment can be expanded by specifying a slideline extension. Points outside the segment but within the extended boundary are considered valid for contact. This is particularly useful near the edges of a slideline surface, where a node could be on a segment in one nonlinear iteration and off the segment in the next iteration – a form of chatter that can cause nonlinear convergence difficulties.
  - The extension parameter is an absolute number.
- **Close contact** This defines a region above a slideline surface within which a soft spring is applied, but with no force. The stiffness of this spring is applied to all nodes that are above a surface but within the close-contact region. This softens the transition between in-contact and out-of-contact states.

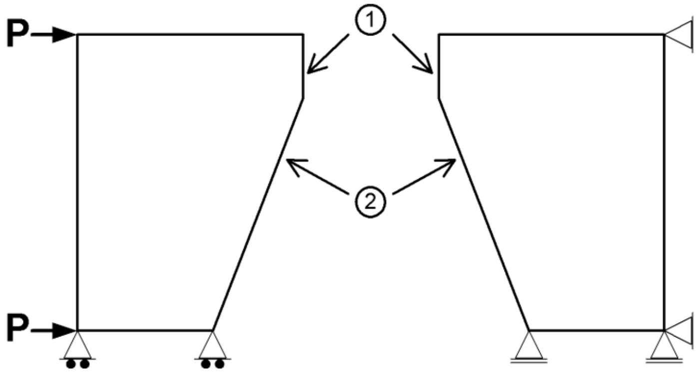
- The close contact facility helps stabilise solutions suffering from chatter in which nodes oscillate between in-contact and out-of-contact states. Chatter can cause a nonlinear analysis to experience convergence difficulties.
- The size of the close contact region is a factor of the segment size. The stiffness of the close contact spring is 10-3 that of the slideline stiffness. It's stiffness is controlled by the Solver system parameter SLSTCC.
- For analyses that continue to suffer from chatter, the size of the close contact region should be increased and the value for SLSTCC reduced accordingly. SLSTCC can be changed via File > Model properties > Solver system variables.
- The close contact facility is not available for explicit dynamics.

Temperature Dependency

Choosing the Temperature dependent check box allows different sets of slideline properties to be specified at different temperatures, thus providing temperature dependence. With temperature dependency, the stiffness scale factors and the coefficient of friction are linearly interpolated across the reference temperatures. All other properties remain unchanged.

Pre-contact

Pre-contact is used to overcome problems encountered when applying an initial load (other than **Prescribed Displacement**) to a discrete body that, without the slideline, would undergo unrestrained rigid body motion. This is particularly the case when an initial gap exists between the contacting surfaces and a load is applied to bring them into contact. Pre-contact is only applicable to static analyses.



Pre-contact brings two bodies into initial contact by using interface forces that act between the slideline surfaces in order to avoid unrestrained rigid body motion. These forces act in a direction normal to each surface. One of the surfaces must be free to move as a rigid body and the direction of movement is dictated by the interface forces, applied loading and support conditions. The facility allows a gap to exist between the surfaces. In the example above pre-contact is defined for slideline 1 but not for slideline 2.

Warning. Incorrect use of this procedure could lead to initial straining in the bodies or to an undesirable starting configuration. By selecting specific slidelines for the pre-contact process (i.e. slidelines where initial contact is expected) minimum initial straining will occur and more control over the direction of rigid body movement can be exercised.

Contact Cushioning

Contact cushioning can be used when convergence difficulties related to in-contact/out-of-contact chatter are experienced. The formulation applies a contact force and stiffness above a surface that increases exponentially as a node moves closer to the surface. This cushions the impact of a node with the surface and softens the transition between in-contact and out-of-contact states. Contact cushioning can therefore help improve nonlinear convergence when chatter is encountered and the set of active contact nodes is continually changing. See *Theory Manual* for details.

Initial slideline type

The slideline type at the start of the analysis (as described earlier)

Change slideline type during a single analysis

Select this option if the slideline type is to change with respect to a loadcase during a particular analysis. If the option is not specified, the slideline defined will apply to all analyses. Note that for the vast majority of analyses it will not be necessary to select this option.

The slideline type can be changed from one type to another at any loadcase stage within a particular analysis. For example, the slideline can be set to be a Friction slideline in loadcase 1, and will apply by default to loadcases 2 and 3, and then be stated to be a No Friction slideline in loadcase 4.

If a loadcase is specified at which a slideline type is to change, the slideline only affects the one analysis that contains that loadcase. If if the slideline type is also required in another analysis, a duplicate slideline attribute will need to be specified for that analysis and assigned to the same model geometry.

Type after change

The slideline type after change (e.g. Friction).

Changes at loadcase

The loadcase at which the slideline type should change from the initial setting to the changed setting. (e.g. Tied to Friction).

Rigid type

To model contact with rigid bodies, rigid slideline surfaces are available. Rigid surfaces can be assigned to valid structural elements as well as to special rigid surface elements R2D2, R3D3 and R3D4. The latter are recommended for modelling rigid bodies, since they remove the need for defining structural elements and hence speed up the solution. All nodes on a rigid surface need to be completely restrained. Since rigid surfaces cannot contact each other only one slideline surface can be defined as rigid – master or slave.

Number of passes

Slidelines involve a two pass procedure in general, in which contact on both slideline surfaces is processed. With rigid surfaces, however, a one pass procedure is available that only checks the penetration of the deformable surface into the rigid surface. If the one pass procedure is selected, it is recommended that the deformable body should have the finer mesh.

Geometric definition

Slideline surfaces can be modelled using linear/bi-linear segments, or as curved contact surfaces using quadratic patches.

- With quadratic patches the curved contact geometry is constructed from a patch of slideline segments. The contact forces are then distributed to the closest segment.
- The quadratic patches and the curved geometry are set-up automatically within LUSAS Solver and no additional specification is required. The standard patch configuration consists of two linear segments in 2D and four bi-linear segments (quadrilateral or triangular) in 3D. Where a patch definition is not possible the standard linear/bi-linear definition is used instead.
- The quadratic patch contact formulation has a non-symmetric tangent stiffness matrix. The non-symmetric solver is therefore set automatically.

Assigning Slidelines

Slideline surface pairs are created by assigning a slideline attribute to selected Lines or Surfaces (or to edges and faces in a mesh-only model).

To assign a slideline:

1. Select features that will form the master surface
2. Assign the slideline attribute to these features
3. In the Assign Slideline dialog that appears, specify Master and the set of features to which it applies. The orientation is computed automatically but needs to be specified for shells (Top or Bottom)
4. Select features that will form the slave surface
5. Assign the slideline attribute to these features

6. In the Assign Slideline dialog that appears, specify Slave and the set of features to which it applies. The orientation is computed automatically but needs to be specified for shells (Top or Bottom)

Slideline Modelling Considerations

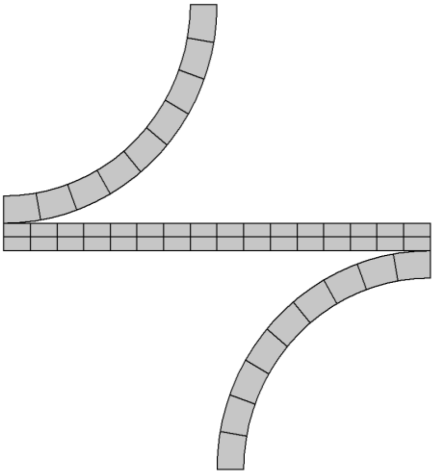
- Except for tied slidelines, the slideline contact facility is inherently nonlinear and must be used in a nonlinear analysis
- Only the expected region of contact should be defined as a slideline surface for tied slideline analyses.
- Coarse mesh discretisation in the region of contact should be avoided.
- Slidelines must be continuous and should not subtend an angle greater than 90 degrees. Sharp corners are best described by two separate slidelines.
- Large mesh bias should be avoided when using quadratic patches, to ensure a reasonable curved geometry is generated
- The stiffness scale factors should be increased for rigid wall contact
- The nodal constraint slideline (explicit tied slideline) treatment is more robust if the mesh with the greatest contact node density is designated the slave surface
- The use of tied slidelines to eliminate transition meshes is recommended for areas removed from the point of interest in the structure
- The use of a larger value for Young's modulus to simulate a rigid surface in a dynamic contact analysis is not advisable since this will increase the wave speed in that part of the model and give rise to a reduced time step. This practice significantly increases the computing time required.
- Slidelines may be utilised with higher order elements (quadratic variation of displacements) but it is necessary to constrain the displacements of the slideline nodes so that they behave in a linear manner (LUSAS Modeller will do this automatically). The deformation of the slideline surface will therefore be compatible with the slideline algorithm. This may, however, lead to a stiffer solution
- When defining slidelines for use in implicit dynamics or static analyses, low order continuum elements are recommended
- Explicit dynamics elements only may be utilised to define a slideline surface in an explicit dynamics analysis
- Do not converge on the residual norm with PDSP loading in a nonlinear analysis. This norm uses external forces to normalise which do not exist with PDSP loading.
- Slidelines may be used with automatic solution procedures (e.g. arc-length methods). The line search and the step reduction algorithms are also applicable.

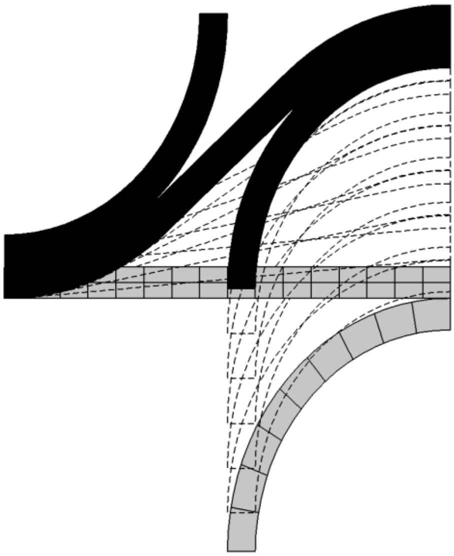
Slideline Options

Options relating to slidelines are set from the **Attributes** tab of the **Model Properties** dialog.

Slideline Example: Metal Forming Analysis

Initial configuration.	Deformed configuration.





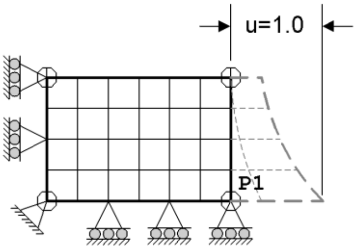
Constraint Equations

A constraint can be defined to constrain the movement of a geometric or nodal freedom. Constraint equations allow linear relationships between nodal freedoms to be set up. Constraint equations can be used to allow plane surfaces to remain plane while they may translate and/or rotate in space. Similarly straight lines can be constrained to remain straight, and different parts of a model can be connected so as to behave as if connected by rigid links. These geometric constraints are only valid for small displacements. Constraint equations can also be used to model cyclic symmetry, for example a single blade from a complete rotor may be modelled and then constrained to behave as if it were part of the complete model. As constraint equations refer to transformed nodal freedoms, any local coordinate assigned to the features are taken into account during **tabulation** when the constraint equations are assembled.

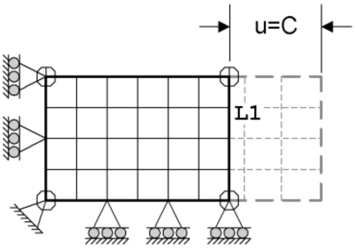
Several different types of Constraint Equations can be defined from the **Attributes> Constraint Equation** menu item. Constraints are grouped under the following types:

Displacement Control

- **Specified Variable** a nodal freedom takes a specified value across all the nodes in the assigned features, In this example, a specified variable constraint of Displacement in the X direction with value 1.0 is assigned to Point 1. The underlying node is then allowed to displace only by the specified distance in the specified X direction.

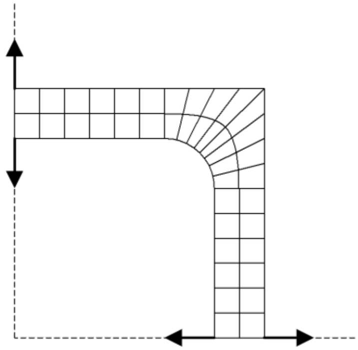


- **Constant Variable** used where a nodal freedom value is constant but unknown across all the nodes in the assigned features.
  In this example, a constant variable constraint of displacement in the X direction is assigned to Line 1. The underlying nodes move a constant amount in that direction.



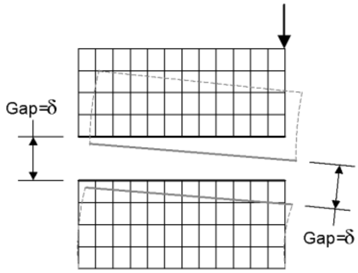
- **Vector Path** The nodes in the assigned features may be constrained to move along a specified vector defined by 2 Points or by 2 sets of X, Y and Z coordinates.

  In this example, vertical and horizontal vectors are used to restrict movement in those directions. Note that the vectors are used purely to define a direction. Nodes can travel along a vector in either direction.

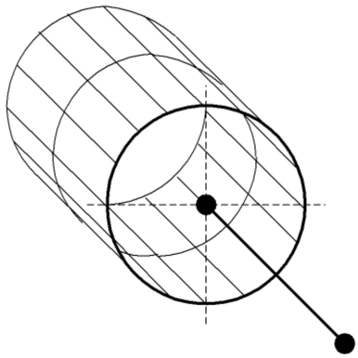


Geometric

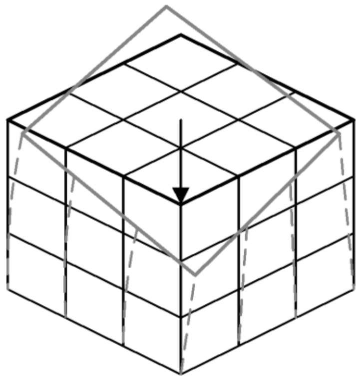
- **Rigid Displacements** The nodes in the assigned features may be constrained to be rigid, the group of nodes may translate and/or rotate but their positions relative to one another remain constant. Only translational displacements can be constrained using this type of constraint. This type of constraint is only valid for small displacements.
  Assigning a constraint of this type to Lines on either side of a gap, as in the example shown, maintains the underlying undeformed node positions relative to each other as if a rigid block were in place between the structures.



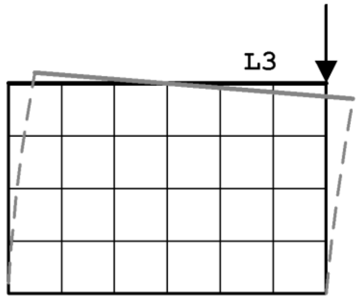
- **Rigid Links** Each rigid link attribute can be used at one location, to create a rigid fixity between features that it is assigned to. It is similar to the Rigid Displacements constraint type, except that rotational freedoms are also constrained to be rigid. In the example shown here, the end of a beam is rigidly linked to the shell edges around a cylinder. The plane containing beam and cylinder end will remain plane throughout the analysis.



- **Planar Surface** A surface may be constrained to remain plane, the surface may translate and/or rotate but remains plane. Nodal positions may vary relative to other nodes on the surface. This type of constraint is only valid for small displacements. In this example, a planar Surface constraint is assigned to the top Surface to force the underlying nodes to remain planar during loading. Constrained nodes may move relative to each other as long as they remain in plane.

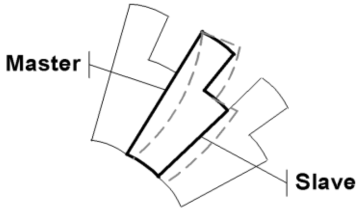


- **Straight Line** A straight line may be constrained to remain straight, the line may translate and/or rotate but will remain straight. Nodal positions may vary relative to other nodes along the line. This constraint type is only valid for small displacements. In the example shown, a straight Line constraint is assigned to Line 3 to force underlying nodes to remain in a straight line relative to each other during loading. Constrained nodes may move relative to each other as long as they remain in a straight line.

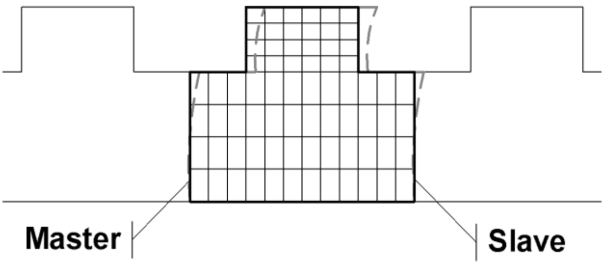


Cyclic

- **Cyclic Rotation** Cyclic rotational symmetry may be used to model a section from a continuous ring. The mesh on the two planes of symmetry may be different. In the example shown, the radial Lines are defined as a Master and Slave pair maintaining cyclic symmetry around the structure. Meshes on the Master and Slave Lines need not match.

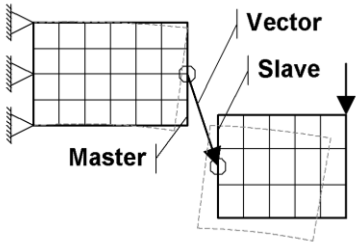


- **Cyclic Translation** Cyclic translational symmetry may be used to model a section from a continuous strip. The mesh on the two planes of symmetry may be different. In the example shown here, Master and Slave Surfaces define start and finish positions of repeating sections. Meshes on Master and Slave need not match.

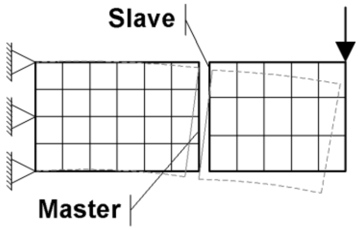


Tied Mesh

- **Specified Constraint** Tied meshes may be used to force two sets of assigned features to move together in a similar manner to tied slidelines. The meshes are tied along Master and Slave Lines to restrict relative movement. The mesh on the two sets of features need not match. A search direction vector is defined to limit the mesh to which it is tied. A vector defines the direction in which the constraint is applied.



- **Normal Constraint** Meshes tied along Master and Slave Lines to restrict relative movement. The underlying nodes maintain their original relative positions under loading. Meshes on Master/Slave need not match. This form of tied mesh constraint uses a search direction normal to the Master/Slave surfaces to detect the mesh to which it is tied.

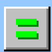


Constraint equation dependency across analyses / loadcases

Constraint equations defined for the first loadcase within a **base analysis** can be optionally inherited by all other analyses. Assigning a different constraint equation to selected features in a following analysis supercedes any inherited constraint equations but for only those assigned features.

Case Study. Using Constraint Equations

Differing meshes may be constrained to displace together in a similar way to a tied **slideline**.

1. Define two Surfaces separated by a small gap using **Geometry> Surface> Coordinates**.
2. Mesh the Surfaces with Linear Plane Strain elements using different mesh spacing on each Surface using the **Attributes> Mesh> Surface** menu item.
3. Define and assign a valid Material to the Surfaces and define and assign Supports and Load attributes so that the Surfaces are being forced towards each other.
4. Define a normal tied mesh Constraint using the **Attributes> Constraint Equation> Tied Mesh** menu item. Assign it to the Lines on either side of the gap. One Line must be selected as a master and the opposing Surface as a slave. If meshes on tied Lines have different spacing, choose the Line containing the finer mesh as the master.
5. Run Solver  and view the deformed mesh. The constraint equations will have prevented one surface from passing through the other.

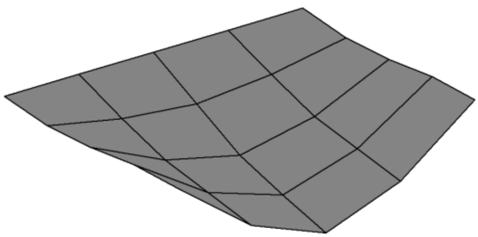
Retained Freedoms

Retained freedoms are used to manually define the master freedoms for use in the following analyses:

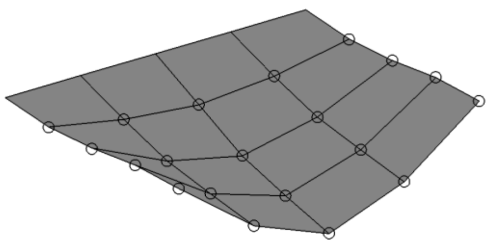
- **Guyan reduction eigenvalue analysis**
- Superelement analysis

Retained freedoms are defined from the **Attributes** menu. They contain the definition of the master (retained) and slave (condensed) degrees of freedom and are **assigned** to the features designated as the **master** nodes.

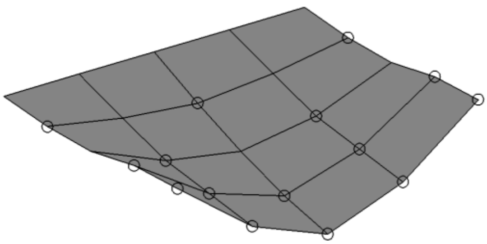
Full Subspace Iteration



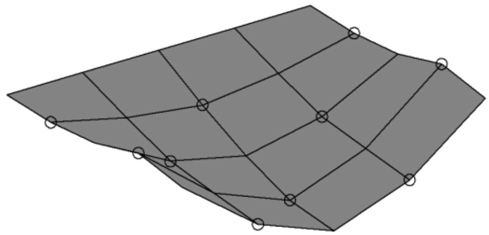
20 Masters



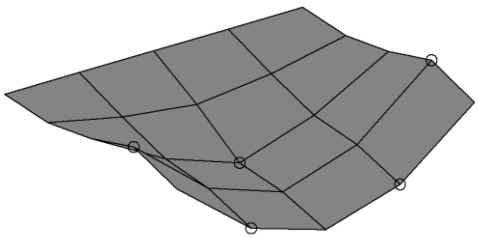
15 Masters



10 Masters



5 Masters



Damping

Damping is used to define the frequency dependent **Rayleigh damping parameters** for elements which contribute to the damping of the structure. Viscous (modal) and structural (hysteretic) damping can be specified. If no damping attributes are specified the properties are taken from the material properties (click on **dynamic properties** on the elastic page of the material attribute dialog).

Damping is usually specified when distributed viscous and/or structural damping factors are required for modal damping control. A modal damping analysis is performed as part of an eigenvalue analysis.

Defining Damping

Structural or viscous damping is defined from the **Attributes> Damping** menu item and **assigned** to features in the usual way. Mass and stiffness Rayleigh damping parameters are linked with the corresponding reference circular frequency value at which they apply in a damping attribute. If more than one set of damping values is defined linear interpolation is used to calculate damping values at intervening frequencies.

Birth and Death (Activation/Deactivation of Elements)

Birth and death enables the modelling of a staged construction or demolition process, whereby selected elements are activated and/or deactivated as the features to which they are assigned are activated and/or deactivated. Activate and deactivate attributes are defined from the **Attributes> Birth and Death** menu item and are assigned to features (or elements in a mesh only model) and manipulated in the same way as other attributes.

Activate

To model the creation or addition of a previously de-activated part of a model, an activate attribute is assigned to those features that are to be added.

- In a structural analysis, an unmodified stiffness matrix is computed for the underlying elements and these activated elements are introduced in a stress/strain free state (uninfluenced by previous stages), except for any initial stresses or strains that have been defined. Strains are incremented from the point of activation and the deformed mesh is used to define the activated element's initial position.
- In a thermal/field analysis activation works in the same manner as for a structural analysis, except that the quantities affected are the conductivity matrix (or other analogous quantity), the fluxes and the gradients.

Deactivate

To model the absence or removal of a part of a model, a deactivate attribute is assigned to those features that are to be removed. In a structural analysis, the underlying elements have their stiffness matrix reduced in magnitude, while for field analysis the conductivity matrix (or other analogous quantity) is reduced. This ensures the deactivated elements have a negligible effect on the behaviour of the remaining model. The element stresses and strains, fluxes and gradients and other analogous quantities are all set to zero.

Percentage to Redistribute

In a deconstruction situation a deactivate attribute controls the way in which internal forces in a loaded structure are processed by specifying how much of the internal forces should be redistributed to the remaining elements.

- ❑ **Zero Redistribution** 0% of the internal forces in a deactivated element are redistributed in the remaining elements (if this is prescribed in a static analysis, and the load remains constant, the stress, displacements etc. in the other elements will remain unchanged).
- ❑ **Full Redistribution** 100% of the internal forces in a deactivated element are redistributed in the remaining elements (this has the same effect as re-assigning very weak material properties to the element).
- ❑ **Fractional Redistribution** A percentage of the internal force to be redistributed is specified. Provides a solution which is part way between the two extremes.

Any remaining internal equilibrating force associated with a deactivated element is maintained in the system until the element is subsequently activated. When an element is activated it is assumed that the element has just been introduced to the model (although all elements must be defined at the outset). The current (deformed) geometry for that element is taken as the initial geometry and the element is assumed to be in a stress/strain free state (unless initial stresses or strains are defined). All internal forces that exist in the element are redistributed and the computed strains are incremented from the time at which the element becomes active.

Maintain relative position of inactive nodes

Specifying **Maintain relative position of internal nodes** ensures that all internal nodes within an inactive feature are constrained such that, for example, a straight line will remain straight and move as a rigid body. This is currently only for use with meshed line features.

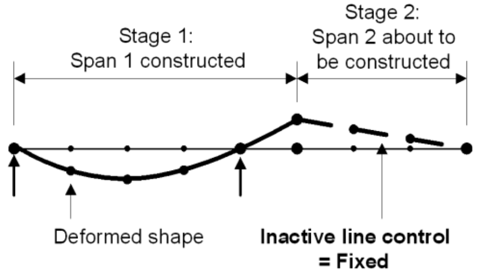
- ❑ **Inactive line control** This controls how the deactivated elements will be rotated or aligned with respect to existing or previously activated adjoining lines in a model. There are four options:

  - **Fixed** restrains the position of the inactive end(s) of an inactive member
  - **Horizontal** ensures the inactive member remains horizontal
  - **None** allows rotation between the active and inactive members. This would be used for a key section as, for example, when a section is positioned between two existing sections.
  - **Tangent** forces the rotation of the inactive members to match that of the active feature at their connection

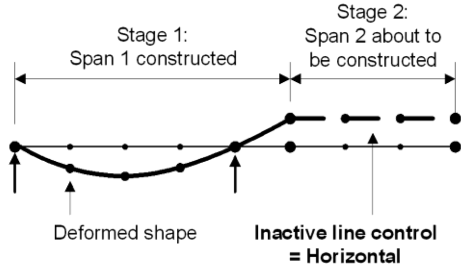
Notes

- The Horizontal, Tangential and Fixed options are only currently available for line features.
- Use of the tangent option may additionally require the use of a jacking load or a prescribed displacement if it is required to return the end of a beam to its original undeformed position. For more information see **Prescribed Structural Loads.**
- If a deactivation attribute is inadvertently assigned to a surface any inactive line control setting will be ignored.

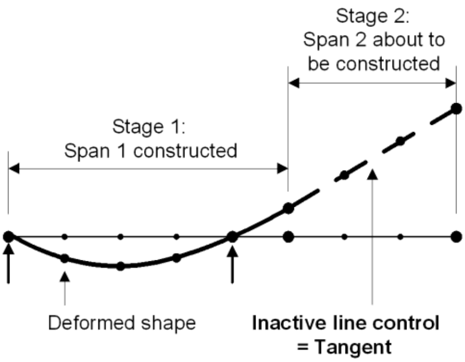
Inactive line control examples



Fixed line control



Horizontal line control



Tangential line control

Using Activate and Deactivate Attributes

All features (and hence elements) to be used in the model need to be defined at the start of the analysis. Activate and Deactivate attributes are assigned on a feature basis and control the history of the underlying elements throughout the analysis. An analysis and a loadcase are specified during assignment to indicate at what point the elements are added or removed.

Dependency across analyses / loadcases

Activate and deactivate attributes defined for one analysis or loadcase are not inherited by following analyses or loadcases, but they can be copied and pasted between analyses in the Analyses Treeview.

Notes

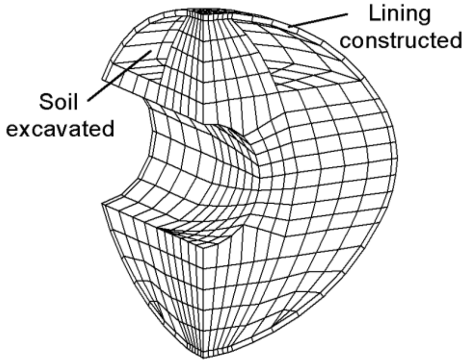
- Activation and deactivation can only be carried out within a nonlinear analysis.
- Deactivation and activation can take place over several increments if convergence difficulties are encountered.
- Elements cannot be activated and deactivated in the following circumstances:
  - Explicit dynamics analyses.
  - Fourier analyses.
  - When they are adjacent to slidelines.
- Deactivated elements remain in the solution but with a scaled down stiffness so that they have little effect on the residual structure. The stiffness is scaled down by a parameter which can be user-defined. In an implicit dynamic analysis the mass and damping matrices are also scaled down by the same factor.

- When an element is deactivated, by default, all loads associated with that element are removed from the system and will not be re-applied if an element is subsequently re-activated. This includes concentrated nodal loads unless the load is applied at a boundary with an active element. The only exception to this rule is a prescribed displacement which may be applied to a node on deactivated elements. Accelerations and velocities may also be prescribed in a dynamic analysis but this is not recommended. By setting LUSAS Solver option 385, the default behaviour can be overridden and loads applied to deactivated elements can be preserved to enable re-application if and when the elements are re-activated.
- If required, initial stresses/strains and residual stresses may be defined for an element at the re-activation stage.
- The activation of an element which is currently active results in an initialisation of stresses/strains to zero, an update of the initial geometry to the current geometry and the element is considered to have just become active. The internal equilibrating forces which currently exist in the element will immediately be redistributed throughout the mesh. This provides a simplified approach in some cases.
- The direction of local element axes can change during an analysis when elements are deactivated and reactivated. In particular, 3-noded beam elements that use the central node to define the local axes should be avoided as this can lead to confusion. For such elements the sign convention for bending moments for a particular element may change after re-activation (e.g. it is recommended that BSL4 should be preferred to BSL3 so that the 4th node is used to define the local axes and not the initial element curvature).
- Care should be taken when deactivating elements in a geometrically nonlinear analysis, especially if large displacements are present. It may be necessary to apply prescribed displacements to deactivated elements in order to attain a required configuration for reactivation.
- It should be noted that the internal forces in the elements will not balance the applied loading until all residual forces in activated/deactivated elements have been redistributed.
- For solving analyses involving constraint equations, such as those used by this facility, it is recommended that the Fast Parallel Iterative Solver solver is used.

General example

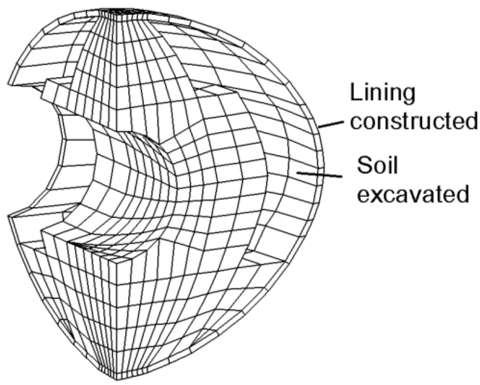
Excavation Stage 1

Top layer of soil deactivated and lining activated. Lining and soil elements duplicated in the model.



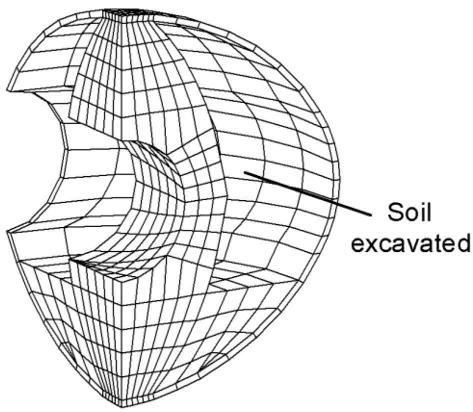
Tunnel Excavation Stage 2

Second layer deactivated as soil excavated. Surrounding lining elements activated



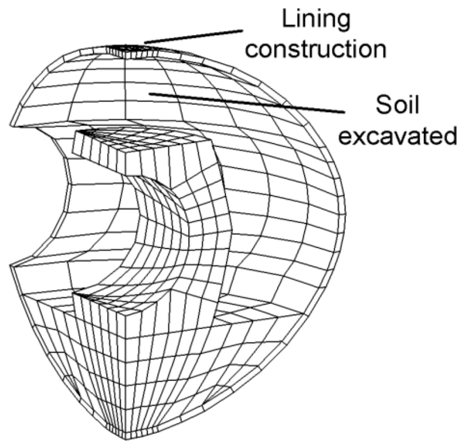
Tunnel Excavation Stage 3

Remaining second layer soil elements deactivated.



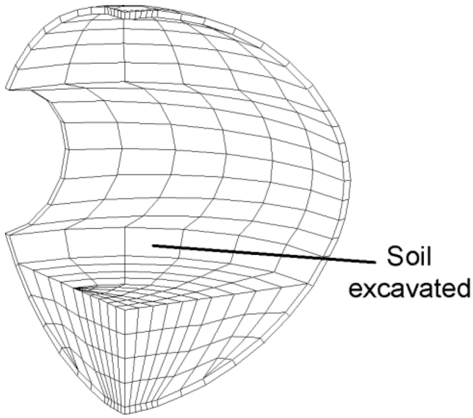
Tunnel Excavation Stage 4

Supporting soil pillar removed and top lining activated.



Tunnel Excavation Stage 5

Final central soil column removed.



Equivalencing

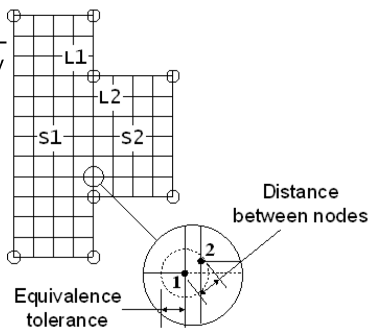
The equivalence facility is used to merge coincident nodes on otherwise unconnected features. If an equivalence attribute is assigned to any features the nodes will automatically be equivalenced after meshing has been carried out.

There are several ways equivalencing can be set up to work:

- ❑ By assigning equivalence tolerances to certain features - only these features will be equivalenced, all others are ignored.
- ❑ By switching on the automatic tolerancing, and accepting the default tolerance - all features are equivalenced according to the default tolerance.
- ❑ By switching on the automatic tolerancing, and assigning other equivalence tolerances to certain features - all features are equivalenced according to either an assigned tolerance or the default tolerance.

Example

In this example, Surfaces 1 and 2 do not share a common boundary Line, therefore the nodes created on their common boundaries will not be joined and must be equivalenced. Node 2 will merge with node 1 if it lies within the equivalence tolerance.



Using Equivalencing

Equivalence attributes are defined from the **Attributes** menu. They are defined as a tolerance, which is used to determine whether nodes are considered to be coincident.

The equivalence attribute is **assigned** to the features that are to be checked for coincident nodes. When an equivalence dataset is assigned to a lower order feature it will search through all higher order features for nodes to be checked. For example, in order to equivalence two Volumes at their boundaries, it is more efficient to assign the equivalence to the Surfaces on the boundaries, as a smaller number of nodes need to be checked.

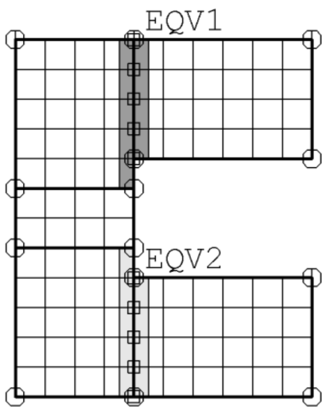
Automatic Equivalencing

Automatic equivalencing can be activated from the **Meshing** tab of the **Model Properties** dialog. This will equivalence all features in the model on meshing if they are within the default equivalence tolerance, or within an assigned tolerance. Note. Remeshing occurs each time a relevant command is issued, but a forced remesh is possible using the **Utilities> Mesh> Mesh Reset** menu item. Automatic equivalencing can be time consuming for models with a large number of nodes.

Visualising Equivalences

Displays the features which have a specified equivalence assigned to them in a chosen colour and line style.

In this example different equivalence tolerances are assigned to different parts of a model to merge more coarsely or finely as required. Using visualisation, the lines to which the equivalence is assigned can be highlighted. Equivalenced nodes can also be visualised as they are removed. In this diagram they are shown using the square symbol.



Summary

- More than one equivalence attribute may be defined in order to rationalise more than one section of the model independently.
- More than one equivalence attribute can be assigned to a feature to equivalence it within a different subset of the model.
- A check for unconnected elements and nodes can be performed using an outline mesh plot (Mesh layer properties), or by checking for duplicate node numbers using the

View > Browse selection menu item and box-selecting around selected points to see if more than one point appears in the list shown.

- The equivalence tolerance must be less than the smallest distance between two nodes on the same feature, otherwise the equivalencing operation will fail.
- Equivalencing may be used to position a point load or support at a node (which is not at a defining feature Point). A Point must be created, the load or support assigned, and the Point and meshed feature equivalenced.
- Equivalencing may be used to merge nodes on the constituent Lines of combined Lines i.e. the nodes on an entire combined Line may be equivalenced, including the Lines forming it.

Influence Attributes

Influence attributes define the result component and methodology that, when assigned to a part of the structure, and in combination with an **influence analysis,** will be used to create an influence surface. They can be defined for use with the following methods:

- ❑ **Reciprocal Theorem** - also known as the Muller-Breslau Theorem, or Maxwell's Theorem, is a means of calculating an influence by the use of an **auxillary structure**, to which a deformation corresponding to a load effect of interest is applied.
- ❑ **Direct Method** - is a more general and powerful way of calculating an influence where the effect of a specified point load is assessed at each node or grid location on a loadable area of a structure. The value of the load effect of interest at each specified location is then used to construct an influence line or surface for that location.

Both methods create influence attributes that need to be assigned to a model prior to an influence analysis being carried out. For details on how to define and assign influence attributes see **Influence Analysis.**

Reciprocal Theorem Influence Attributes

Reciprocal Theorem influence attributes are defined from the **Attributes > Influence> Reciprocal Theorem** menu item.

Specifying an influence attribute

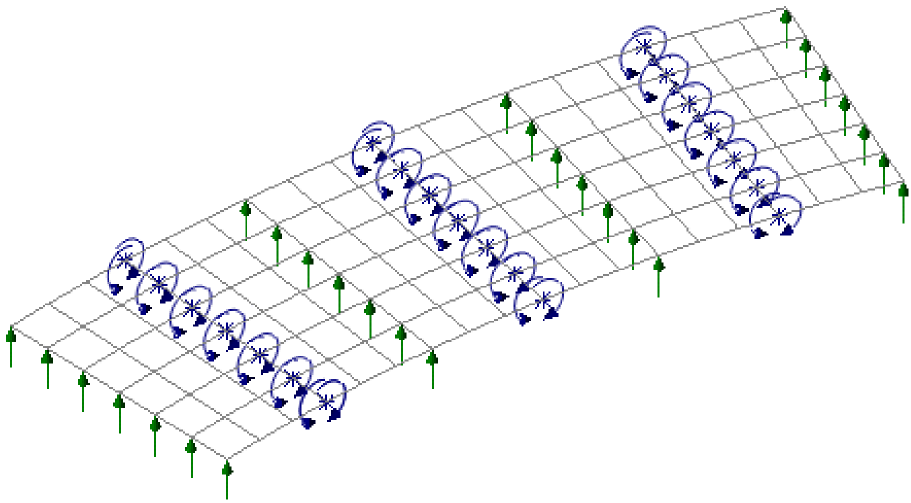
The influence attribute may be set by defining an **Influence type** of either Shear force, a Reaction, a Moment or a Displacement, an **Influence direction** of either Longitudinal or Transverse, and a **Displacement direction**of either Positive or Negative.

Note that a shear or moment influence type is mesh dependent. The Influence direction defines the axis about which the component of interest will be evaluated. Once created, an influence attribute is held in the Attributes Treeview for assignment to selected mesh nodes or Points on a model. A Direction definition object containing information relating to setting the vertical, longitudinal and transverse axes for a model is added to the Influence entry in the Attributes Treeview once the first influence attribute has been defined.

A short influence name is recommended since this is used to create filenames that also include coordinates of assigned influence attributes and element details, and if collectively this is too long, the Windows path limit of 260 characters may be exceeded. For more details see **File and folder naming in LUSAS**.

Assigning Reciprocal Theorem Influence Attributes

One or more nodes or Points on the model may be selected to make a reciprocal theorem influence attribute assignment. Assigned attributes are stored within an Reciprocal Influence analysis entry in the Analyses Treeview. Influence loadcase naming is based upon node coordinates. When assigned to the model LUSAS automatically determines the break-away elements required by the Reciprocal Theorem method in each case. Influence type symbols are drawn at each influence location to show the type of mesh break that is being used.



Visualisation of Assigned Influence Attributes

Reciprocal Influence surfaces can be visualised if attributes are assigned to a model and solved. The type of visualisation that takes place to help identify the influence type and the participating elements at each node or point of interest depends upon the type of influence and whether any manual selection of participating elements was done.

Visualisation settings are accessed using the context menu for an influence attribute. If influence assignments are visualised the whole model may be re-displayed after viewing selected influences by right-clicking in the View window and selecting the All Visible option.

Note that if a solve (to visualise influence surfaces) is not carried out prior to a Vehicle Load Optimisation analysis being run (with the generate Influence Surfaces check-box checked), influence loadcases are added to the Analysis Treeview but no visualisation of these influence shapes can be done.

Identifying Assigned Attributes

The location of an assigned influence attribute can be found on the model by using its context menu and choosing Select Assignments to list it in the **Selected items** window. Any objects listed in this window can be located on the model by using the Find menu item on the context menu for the object. The location of the assigned attribute will be identified by a temporary indicator.

Changes to Mesh after Attribute Assignment

If a model is re-meshed or has its geometry edited the influence points will remain visualised with those influence points still overlying a node or Point remaining marked with an appropriate 'break' symbol. Influence points that no longer lie on nodes or Points as a result of any modifications remain visualised on screen but with a 'not assigned' symbol alongside their name in the Analyses Treeview. Models can be solved with unassigned influences present in the Analyses Treeview.

Direct Method Influence Attributes

Direct Method influence attributes are defined from the **Attributes > Influence> Direct Method** menu item.

Specifying an influence attribute

The influence attribute may be set by defining an **Entity** of interest (such as a Reaction, or a Force/Moment, or a Stress for example), an influence **Direction** (such as a local axis of a member, or a path along a structure etc.) and a **Component** of interest (such as My etc).

The Entity drop-down list is populated according to the elements that are present in the model. The Direction drop-down list presented depends upon the type of entity selected. The Component drop-down list shows the influence components available for the chosen entity.

Using paths

Specification of an influence direction using the **Path** facility is recommended for grillage models, curved beam models, and for curved decks modelled with plates or shells where longitudinal and transverse effects are of interest. A default path of **longitudinal = global X** is provided to allow influences to be assigned to models where element axes lie in this direction without having to actually define a path. This default path does not appear in the Utilities treeview and is not editable. Otherwise for most models, the specification of a path will require a reference path to have been previously defined prior to selection. If a reference path has been defined incorporating a skew angle then selecting the **Skew** check-box will ensure that for grillage or beam models all elements lying at that skew angle will be considered to be the transverse elements. Otherwise, by default, transverse elements are assumed to be those elements perpendicular to the longitudinal elements. A path cannot be selected for 2D line beam elements; for these, all components are calculated using the element local x-axis.

The **Component** drop-down list shows the influence components available for the chosen entity. Note that when a Path has been specified, in addition to the standard choices of Component, an Entity selection of Force/Moment will show component values of:

- **FV (longitudinal members)** (Force in the Vertical direction for longitudinal members)
- **FV (transverse members)** (Force in the Vertical direction for transverse members)
- **MF (longitudinal members)** (Flexural Moment in longitudinal members)
- **MF (transverse members)** (Flexural Moment in transverse members)

Element Participation and Influences Generated

It is possible to control the number of influences created and the way in which values from participating elements are calculated or averaged for a node or point of interest when a solve is carried out. The influence type group consists of the following options:

- ❑ **Unaveraged influence for each connected element** - For this option, when assigning the attribute, a separate influence assignment is created for each connected element (if a node is selected) or connected geometry feature (if a point is selected). An assignment created this way provides un-averaged nodal results at the node of interest from the element for which the assignment was created. This is typically desirable in beam models where more than two elements meet, or when elements and their axes are not aligned, or when support conditions oppose the influence direction. This assignment type is drawn, by default, in orange and the contributing element is identified with a blue square box. Note that selecting adjacent elements to a node or point of interest prior to assignment of an influence attribute limits the operation to only those selected elements.
- ❑ **Average influence of all connected elements** (not available for displacement or reactions) - For this, when assigning the attribute, only one influence assignment is created at the node/point selected. An assignment created this way provides averaged nodal results at the node of interest from all connected elements at the time of solving. This is typically desirable for influences in shell models with a regular grid of nodes or points. Averaging of all connected element values is not generally appropriate where element axes are not aligned, or when support conditions oppose the influence direction, or for grillage models (see note). This type of influence assignment is referred as ‘averaged influence assignment’ and, by default, is drawn in blue. Note that selecting adjacent elements to a node or point of interest prior to assignment of an influence attribute limits the averaging to only those selected elements.
- ❑ **Automatically choose elements for averaging** (not for available plate and shell elements) - For this, when assigning the attribute, Modeller looks at the location of the assigned direct method influence attribute, checks for element type and a common alignment of element axes for the elements meeting at the point of interest (within a small tolerance), and if there are no supports opposing the influence, the 'Average influence of all connected elements' option is chosen (see note for grillage models). Otherwise, if the element axes are not aligned the 'Separate influence for each connected element' option is chosen instead.

- ❑ **Name** A short influence name is recommended since this is used to create filenames that also include coordinates of assigned influence attributes and element details, and if collectively this is too long, the Windows path limit of 260 characters may be exceeded. For more details see **File and folder naming in LUSAS**.

Once created, an influence attribute is held in the Attributes Treeview for assignment to selected mesh nodes or Points on a model - either directly or by the use of a loading grid. Subsequent assignment of the same attribute to other nodes or points of interest will use the same element participation settings.

Element participation notes

- Grillage models (whether constructed using 2D grillage elements or 3D beam elements) generally require moment results to be averaged over adjacent pairs of longitudinal members only, or adjacent transverse members only. This is automatically done by Modeller when an influence attribute has been defined with respect to a Path and with the 'Automatically choose elements for averaging' option selected.
- When assigning a shear influence defined with ‘Average influence of connected elements’ it will be necessary to additionally select elements to one side or other of the node or point at the support to clarify which elements should participate in the calculation.
- When only a point/node is selected at the time of assigning an ‘Average influence of all connected elements’ influence type, only a single assignment is made, but results will be presented for all elements that are connected to that node at the time of solution. So, if an additional member is added to the intersection after making the assignment, that additional member will also be considered when results are presented.
- When only a point/node is selected at the time of assigning a ‘Unaveraged influence for each connected element’ influence type, an assignment is made for each line/element connected at the time of assignment. If an additional member is added to the intersection after making the assignment, that additional member will not be considered when results are presented.
- It is not currently possible to edit element participation choices once an influence attribute has been assigned to a model. However, the de-assignment of the attribute from the model, or the deletion from the Analyses Treeview of a Direct Method Influence analysis that uses the influence attribute will allow participation settings to be changed.

The availability of **influence entity** options for different influence entities is summarised in the following table:

Influence entity	Separate influence for each connected element	Average influence of all connected elements	Automatically choose elements for averaging
Displacement or Reaction	Not available	Not available	Not available
Force or Moment	Available	Available	Available (for bar and beam elements only)
User defined results	Available	Not available	Not available

The availability of **element type** options for different element types is summarised in the following table:

Element type	Separate influence for each connected element	Average influence of all connected elements	Automatically choose elements for averaging
Bar and beam elements	Available	Available	Available
Plate, shell and membrane elements	Available	Available	Not available
Continuum elements	Available	Available	Not available
Joint elements	Available	Not available	Not available

Assigning Direct Method Influence Attributes

Influence attributes can be assigned to one or more selected nodes or points on the model using either a drag and drop technique or the context menu options for an attribute.Note that selecting adjacent elements to a selected node or lines adjacent to a selected point prior to the assignment of an influence attribute limits the averaging of values to only those selected items. Influence attributes are assigned in a Direct Method Influence analysis and if one has not been created at the time of assignment a prompt will be given to create one.

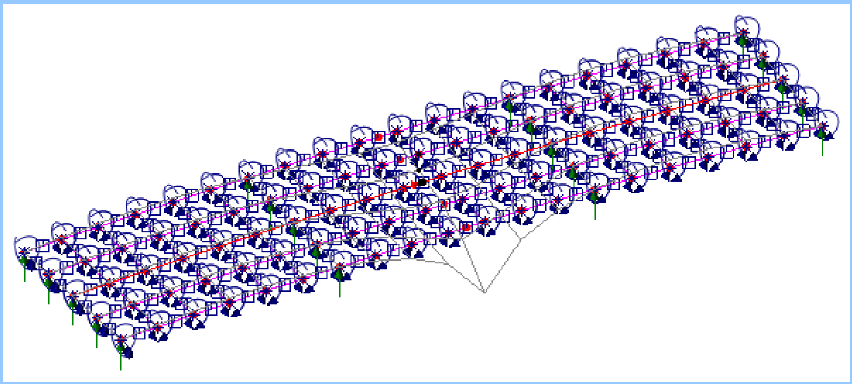
Assigning influence attributes to points as opposed to nodes of the mesh (as may be preferred, perhaps, for reaction influences) has the benefit that, if changes are made to the geometry and the location of the point is moved, the attribute assignment will continue to remain valid. An assignment to a node that has a coincident point will assign the attribute to that point instead. Attributes assigned to mesh nodes only, remain valid only if the influence location overlays a mesh node and the elements are compatible with the results entity specified in the Influence attribute. See **Changes to Mesh after Attribute Assignment.**

If a shear influence type is assigned to a node or point that is supported , a shear influence assignment is made to the end of each connected element or line at that assignment location.

Case Study: Defining and assigning Direct Method Influence attributes to a grillage model

The basic procedure to define and assign Direct Method influence attributes to a grillage defined using the LUSAS grillage wizard is as follows:

1. Define an influence attribute for Flexural Moment (Longitudinal members) using Automatically choose elements for averaging.
2. Using the point selection tool, select all points in the model.
3. In the Groups Treeview, right-click on the Longitudinal group name and choose Add to Group.
4. Assign the influence attribute to the selected points creating every possible longitudinal influence.
5. Define an influence attribute for Flexural Moment (Transverse members) using Automatically choose elements for averaging.
6. In the Groups Treeview, right-click on the Transverse group name and choose Add to Group.
7. Assign the influence attribute to the selected points creating every possible transverse influence.
8. Solve the model to generate the influences and view them by adding the Influence shape layer to the Layers Treeview.



For grillages created outside of the grillage wizard a similar process would apply using repeated box selections to select all longitudinal members (and their lower order points) prior to assignment of an influence attribute to create every possible longitudinal influence, followed by repeated box selections of all transverse members (and their lower order points) prior to assignment of an influence attribute to create every possible transverse influence.

Note also that a similar result could be obtained by using the “Automatically choose elements for averaging" influence type by selecting a suitable reference path and component, and assigning the influence attribute to all points in the model.

Using a Loading Grid

For certain types of Direct Method influence analysis (see Examples of Loading Grid use) it is beneficial to use a loading grid. Loading grids are predominantly used with line beam models where the geometric beam section represents a loadable top slab. The loading grid also provides the means to define a grid of points that can be used as an alternative to the nodes or points present in a shell or plate model that are used in an influence analysis, and depending upon grid settings used, can ensure that loadable grid locations do not occur at supported nodes - which, for particular components of interest, can be an **issue**. The use of a **reference path** defining the centreline of a loadable carriageway is recommended whenever a loading grid is used.

The use of a loading grid will require the following to be specified:

- ❑ **Centreline** - the centreline of the grid can be defined with reference to a previously defined **Path** or the **X**, **Y** or **Z** global axes. The use of a reference path is recommended for most situations.
- ❑ **Transverse width** - the width of the grid to be loaded (must be greater than 0.1).
- ❑ **Grid settings** control the density of the grid of points that will be loaded. Longitudinal and transverse values can be specified. If one of the global axes has been selected as the centreline of the grid, then the extents of the model will be calculated.
- ❑ **Loading settings** The magnitude of the influence load (in terms of a positive or negative direction) and its actual direction of loading (in terms of the X, Y, or Z global axes, or the Vertical axis of the model can be defined. A default load magnitude of 1e3 is used if the model units are metric, with an equivalent rounded value used if the model units are imperial. This default value is used in preference to a unit load so that any factoring or scaling of the loading (depending on the unit system in use) does not cause the order of magnitude of displacement to fall below tolerances used in vehicle load optimisation software.

Loading grid notes

- When using a loading grid to represent a generally loadable region of carriageway on a bridge deck, lines representing the extent of the carriageway will still need to be defined and selected prior to carrying out a **Vehicle Load Optimisation** analysis.
- For a model with a very fine mesh, loading a grid having fewer points will reduce the solution time. But note that coarse mesh refinement or coarse loading grid settings can lead to inaccurate influence shapes being created, especially near any cusps.

Loading Grid Visualisation

The visualisation of the loading grid is controlled for each direct method influence analysis entry in the Analyses Treeview by deselecting / selecting the **Show grid** context menu item. The default is to always show a loading grid. It is not possible to delete a loading grid.

Examples of Loading Grid Use

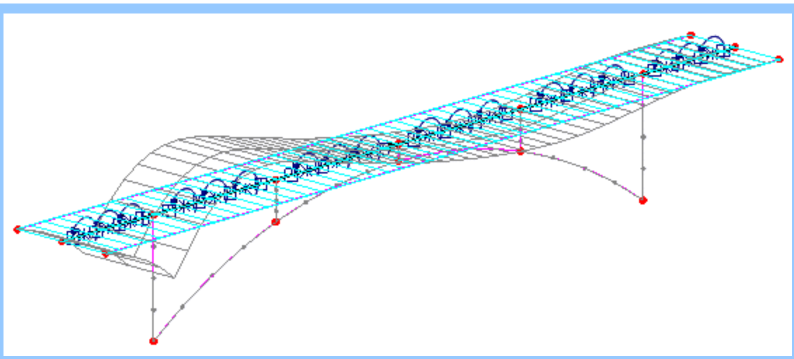
Loading grids are used in the following situations:

- When a general grid is required to encompass a whole model. (Note that this cannot be used for carriageways that divide into two or more branches).
- On a line beam model where the geometric section represents a beam with a loadable top slab.
- When a different set of loadable points is required in preference to nodes on a finely meshed shell and plate models.
- On a skewed grillage model where it is required to obtain results for longitudinal and transverse (skewed) members.

Case Study: Defining and assigning Direct Method influence attributes to a 3D line beam model using a loading grid

The basic procedure to carry out a Direct Method Influence analysis of a 3D line beam model where the geometric section represents a beam with a loadable top slab is as follows:

1. Define an influence attribute that uses a pre-defined reference path along the beams that represent the deck and loadable slab.
2. Using the node selection tool, select all the nodes in the beams along the path.
3. Assign the influence attribute to all the selected nodes using a loading grid with a centreline along the reference path and specify a width appropriate to the loadable carriageway.
4. Solve the model to generate the influences and view them by adding the Influence shape layer to the Layers Treeview.



Note that to carry out vehicle load optimisation analysis, lines defining the width of the carriageway must also be defined.

Naming and Visualisation of Assigned Influence Attributes

Assigned influence attributes create influence loadcases within a Direct Method Influence analysis entry in the Analyses Treeview. The naming of these influence loadcases and the type of visualisation that takes place (or not) to help identify the influence type and the participating elements at each node or point of interest depends upon three things: the type of influence Entity chosen; the element participation settings made when the attribute was first defined; and whether any manual selection of participating elements was done.

- Nodes assigned an influence attribute where a direction is associated with a component are denoted by an assignment symbol.
- Nodes assigned an influence attribute where a direction cannot be associated with a component are denoted by a blue symbol.
- If **Unaveraged influence for each connected element** was chosen when the attribute was first defined, the assigned attribute is drawn, by default, in orange and all elements participating in the calculation for that influence point are identified in the View window by a participating element symbol . In the Analyses Treeview a separate assignment is created for each node/point and participating element/line.
- If **Average influence of all connected elements** was chosen when the attribute was first defined, the assigned attribute is drawn, by default, in blue and no visualisation of participating elements take place. By implication all elements seen to be connected to the node or point of interest are being used in the calculation for that influence location. If any elements or features were explicitly defined as participating, these will be identified by a participating element symbol.
- Initially assigned influence attributes that no longer overlay a node (perhaps because of a revised mesh arrangement) are denoted by a red symbol. These influences are termed 'pending' and appear in red text in any list of influences to be analysed. Assigned influence attributes can also be denoted as 'pending' if some participating

features are not meshed at the time of assignment, or if those features are not compatible with the results entity defined for the influence attribute.

- Assigned moment or reaction influence types are denoted by arrows in the plane in which the calculation of averaging of values is taking place.

Additional visualisation settings are accessed using the context menu for an influence attribute.

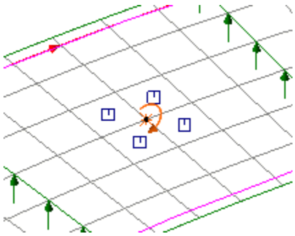
Examples of Element Participation and Visualisation Symbols

Showing symbols used to identify nodes, participating elements and components of interest.

Plate/shell model

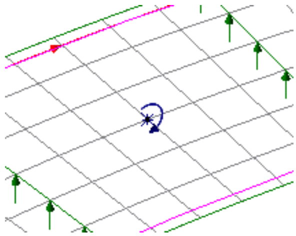
Symbols used for an influence attribute defined to investigate Flexural Moment in longitudinal members with reference to a path.

"Separate influence for each connected element"



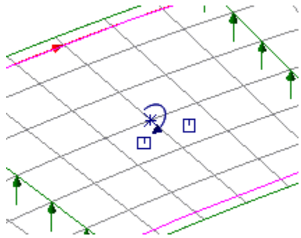
Elements identified by square symbols are participating at the node

"Average influence of all connected elements"



All connected elements to the node are being averaged

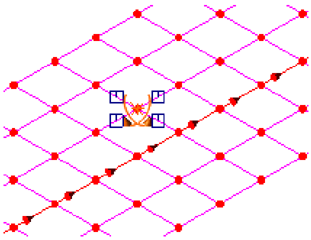
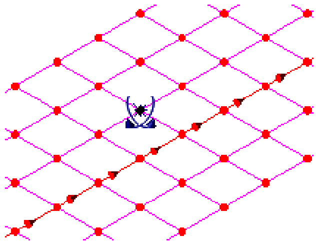
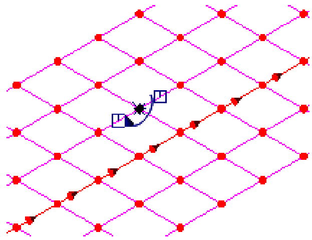
"Selected elements only"



Only those selected elements identified by square symbols are participating at the node

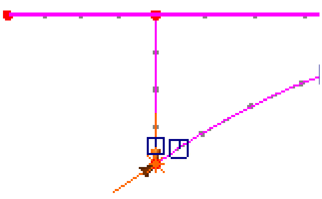
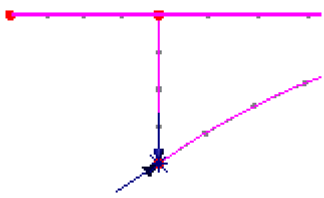
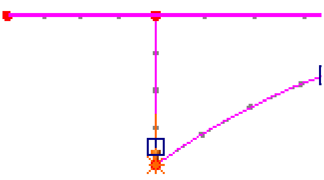
Grillage Model

Symbols used for an influence attribute defined to investigate Flexural Moment in longitudinal members with reference to a path.

"Separate influence for each connected element"	"Average influence of all connected elements"	"Automatically choose elements for averaging"
		
Elements identified by square symbols are participating at the node	**All connected elements to the node are being averaged. (Not normally applicable to grillages)**	**Only the longitudinal elements identified by square symbols are participating at the node**

2D/3D Frame Model

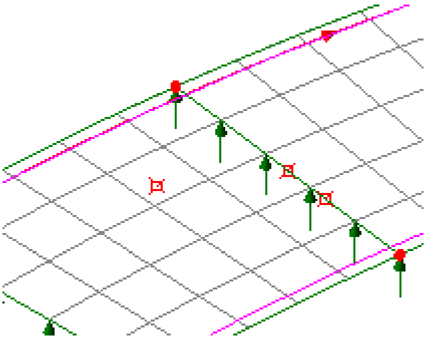
Symbols used for an influence attribute defined to investigate axial forces or moments in members of a 2D/3D frame with reference to a path.

"Separate influence for each connected element"	"Average influence of all connected elements"	"Selected elements only"
		
Elements identified by square symbols are participating at the node	**All connected elements to the node are being averaged.**	**Only the elements identified by square symbols are participating at the nodes.**

Identifying Influence Locations on the Model

The location on the model of an influence entry in the Analyses Treeview can be found by using the context menu for the influence entry and choosing the **Find** menu item. The location will be identified by a temporary indicator of animating concentric shrinking squares.

Changes to Mesh after Influence Attribute Assignment



If a model is re-meshed or has its geometry edited affecting the mesh, assigned influence attributes will remain visualised with those influence types still overlying a node remaining marked with an appropriate influence type symbol. These marked locations will be included in an influence analysis. Influence points that no longer lie on nodes as a result of any modifications remain visualised on screen and are denoted by a red symbol, but they will have a 'not valid' symbol alongside their name in the Analyses Treeview. These influences are effectively 'pending' (requiring correction or removal) and appear in red text in any list of influences to be analysed. These will not be included in an influence analysis. Note that models can be solved with influences marked as 'not valid' and with unassigned influences present in the Analyses Treeview.

Solving an Influence Analysis

See **Influence Analysis**

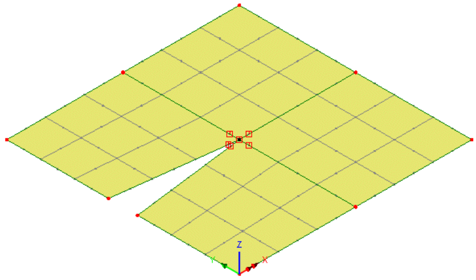
Age

Age attributes are defined from the **Attributes> Age** menu item. Age attributes define the time in **timescale units** between the creation and activation of features in the model and are used in conjunction with the **CEB-FIP Concrete Material Model**. When assigned to a feature all elements created by that feature are assigned the specified age.

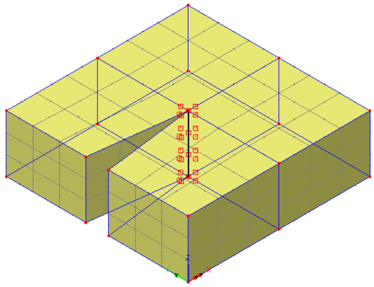
See *Solver Reference Manual* for further details.

Crack tip attributes

A crack tip attribute allows a crack tip location to be defined at a point in a surface model and at either a point or line in a volume model. Crack tip attributes are defined from the **Attributes > Crack tip** menu item and are only for use with 2D and 3D quadratic continuum elements. After assignment the mid-point nodes of elements adjacent to the crack tip assignment are automatically moved to the nearest quarter point position within the element and the continuum elements adjacent to the crack tip assignment are automatically replaced with an equivalent crack tip element. When assigned to a point, the crack tip always occurs at a corner node of an element. When assigned to a line (volume models only), the crack tip occurs all along the line. Assigned crack tip attributes can be visualised using symbols displayed on the nodes of the assigned feature and at the mid-side nodes of the adjacent elements that have been moved towards the assigned feature.



Crack tip attribute assigned to a point in a 2D model



Crack tip attribute assigned to a line in a 3D model

Thermal Surfaces and Heat Transfer

The thermal surface facility allows thermal gaps, contact and diffuse radiation to be modelled. Thermal surfaces are used to model the thermal interaction of two distinct bodies, or two different parts of the same body through a fluid medium.

- **Thermal Gaps** are used to model gaps between structures that are relatively close together.
- **Contact** is used in a thermo-mechanical coupled analysis where contact takes place and the contact pressure effects are then included in the analysis.
- **Diffuse Radiation** is the process of heat transfer from a radiation surface to the environment or to another thermal surface defining the same radiation surface. Radiation is modelled by specifying radiative properties for thermal surfaces.

Thermal surfaces

Thermal Surfaces are the thermal equivalent of structural slidelines. They are defined from the **Attributes> Thermal Surface** menu item and are assigned to features of the model and manipulated in the same way as all other attributes. A thermal surface must be defined before thermal gap or radiation properties can be specified.

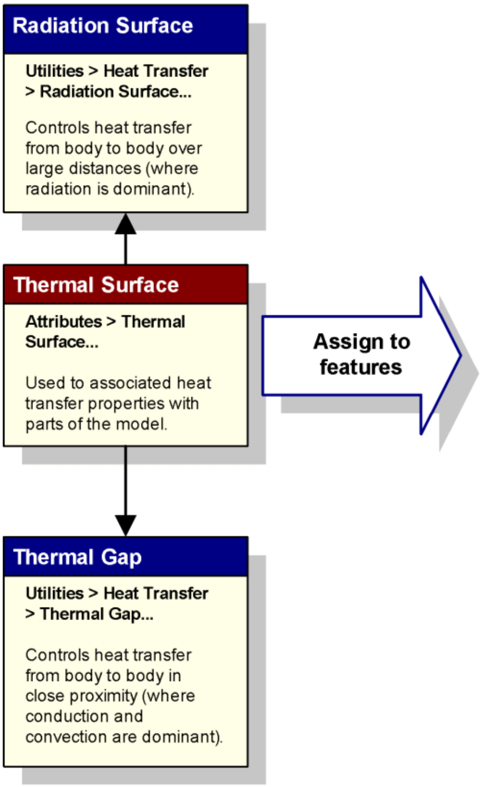
- **Radiation properties** are required when defining a radiation surface for heat transfer by radiation exchange.
- **Environment properties** are required when thermal environment properties exist. Used for heat transfer to the environment (convection and conduction).

Thermal Surfaces work in conjunction with **Thermal Gaps** and **RadiationSurfaces**. See below for details.

Heat Transfer

Thermal gap and radiation surface properties are used to dictate the type of heat transfer that can take place between Thermal Surfaces. They are defined from the **Utilities> Heat Transfer...** menu item. As utilities they cannot be assigned directly to features of the model as Thermal Surfaces can. A thermal surface must have been defined prior to specifying any thermal gap or radiation properties. The process of thermal surface / heat transfer definition is summarised in the following diagram.

- **Thermal Gaps** Thermal gaps are used to model heat transfer across a gap and heat transfer by contact when a gap is deemed to have closed. If these effects are required, the thermal surfaces defining the gap must be specified on the Thermal Gap properties dialog.
- **Radiation Surfaces** Diffuse radiation exchange may be modelled with a radiation surface that is defined by any number of thermal surfaces. Planes of symmetry that cut through the radiation enclosures may be defined so that it is not necessary to model the whole structure. Radiation surfaces allow for the calculation of diffuse view factors. These view factors may be output to a print file

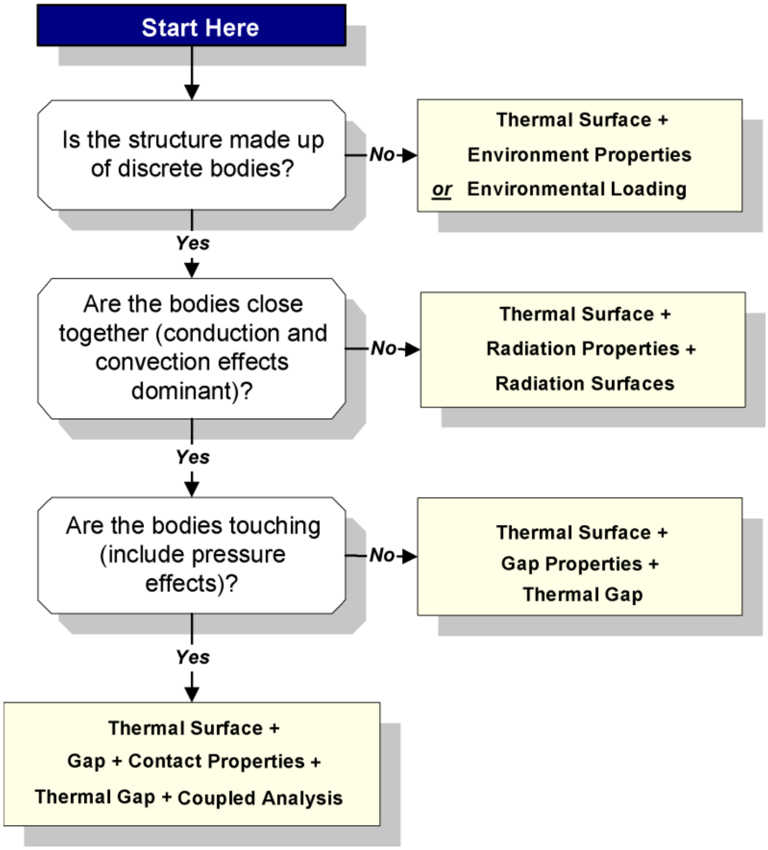


Specifying thermal surfaces defining a gap

Pre-defined thermal surfaces can be selected on the Thermal Gap properties dialog in order to define a gap. The gap can be defined as active or inactive initially and be set to change according to loadcase.

Choosing Thermal Properties

The following flowchart guides the decision making process for choosing thermal properties. The process is simplified if the analysis only considers a single body, when only environmental thermal properties are required. For analyses where discrete (multiple) bodies are considered, factors such as body proximity and whether the bodies are touching, or are likely to touch during the analysis, become important and the choice of thermal properties changes. Follow a route through the flowchart below and define your thermal surfaces using the properties given in the shaded box.



Environmental Nodes (LUSAS Solver data file only)

Environmental nodes may be used to represent the medium which separates the thermal surfaces between which heat is flowing. As the length of a link directly connecting two surfaces increases, the validity of the assumed flow becomes more tenuous. Alternatively, instead of forming a link, heat could flow directly to the surroundings, but in this case, the heat is lost from the solution. This, in some cases, is a poor approximation to reality, particularly when the thermal surfaces form an enclosure. In this instance an environmental node can be used to model the intervening medium, with all nodal areas which are not directly linked to other areas linked to the environmental node. The environmental node then re-distributes heat from the hotter surfaces of the enclosure to the cooler ones without defining the exact process of the transfer.

Note. Environmental nodes cannot be defined in LUSAS Modeller, and must be edited directly into the LUSAS Solver data file if required. See the *Solver Reference Manual* for further information.

Radiation Options

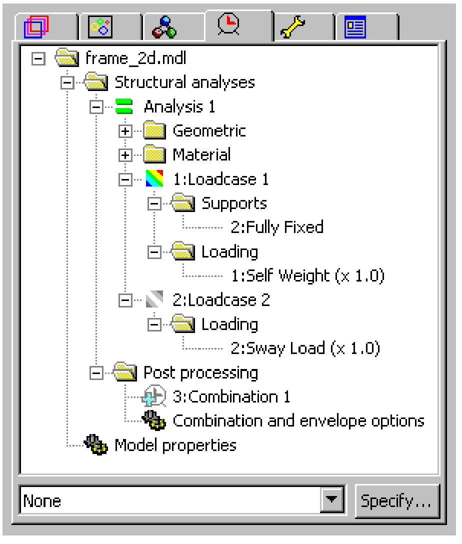
Radiation options are set from the Coupled analysis options object in the Analyses Treeview.

- **Suppress Recalculation of View Factors in Coupled Analysis** Turns on/off the view factor recalculation. The option should be turned on when the radiation surface geometry is unchanged by the structural analysis. This stops recalculation of the view factors. LUSAS Solver option 256.

Overriding of thermal surface attributes between analyses

The default state for a second or subsequent thermal analysis is to inherit the attribute assignments from the first one, but the additional analysis can 'override' some or all of those assignments. However the overriding of thermal surfaces relies on no overlaps between defining lines/surfaces so is only meaningful if they are assigned in exactly the same pattern. For example, in analysis 1, Thermal Surface 1 can be assigned to Surface 1 and 2, and Thermal Surface 2 can be assigned to Surface 3 and 4. In analysis 2 it is valid to assign Thermal Surface 3 to Surface 3 and 4, 'overriding' the assignment of the first analysis. But to assign Thermal Surface 3 to Surface 4 and 5 would cause an overlap because the state of Surface 4 is undefined - it cannot be a part of two different thermal surfaces. To overcome this the inheritance of the thermal surfaces would need to be switched off on the analysis definition so that the two analyses are independent.

Loadcases



A loadcase is a collection of loading types such as a point load, a body force, or a face load etc., that have been assigned to a particular analysis in a model using the same loadcase name. Each loadcase name and its loadcase dependent assignments (such as supports and loading) is held within an analysis entry the Analyses Treeview.

A typical model may have many loadcases assigned to potentially more than one analysis. When each analysis is run each loadcase is solved and the results for each loadcase are provided separately. Note that nonlinear analyses produce results loadcases for each iteration used to achieve a convergent solution. Combinations and envelopes of loadcase results can be created to look at combined or maximum / minimum effects.

Analysis control parameters for eigenvalue, fourier and nonlinear and transient analyses are defined as properties of a loadcase using its context menu. In a nonlinear analysis, LUSAS loadcases may be modelling individual analysis stages and within this type of analysis the loading or supports can be modified between loadcases.

Loadcase dependent attributes and results for a loadcase can only be visualised by setting that loadcase active. This is done by choosing the Set Active menu item from its context menu.

Loadcases can be copied, pasted, renamed, and deleted within the Analyses Treeview. Loadcases may be re-ordered using drag and drop within each analysis entry - but only when no results files are loaded.

Load curves

For analyses in which the load varies with time (or increment number) load curves may be used. When using load curves all loads must be assigned to a load curve instead of a loadcase. All other loadcase dependent attributes (support, slidelines etc.) and analysis control is assigned to loadcases in the usual way. The analysis control assigned to each loadcase determines the time over which the load is applied. The magnitude of the applied load is computed from the time dependent function defined within each load curve. Any number of loadcases and load curves may be specified within a single analysis. Each load curve is assumed to begin at the start of the analysis (t=0). If the input values stat from t=n the load curve is assumed to be zero when t<n. For more information see **Load Curves**.

Creating Loadcases / Loadcurves

New loadcases/loadcurves may be added to the Analyses Treeview in the following ways:

- By selecting the **Analyses> Loadcase** menu item.
- By right-clicking on the Structural analyses (or Thermal Analyses) folder or the analysis entry and selecting the **New** > **Loadcase...** menu item.
- By entering a loadcase name when a load attribute is assigned to a feature on a model.
- By copying and pasting existing loadcases in the Analyses Treeview.

Attribute dependency across loadcases

- Within a linear analysis, and with the exception of loading, all attributes assigned to the first loadcase of an analysis will apply to any additional loadcases that follow in that analysis.
- For nonlinear and transient analysis many attributes may be modified on a loadcase by loadcase basis as the analysis progresses. An attribute assigned in a loadcase will remain active until it is changed. This means a support assigned in the first loadcase will apply to all loadcases unless it is set free in a subsequent loadcase.

Loadcases and supports

- Supports defined for the first loadcase within a **base analysis** can be optionally inherited by all other analyses.
- For a linear analysis, supports assigned to the first loadcase of the analysis will be used by all following loadcases within that analysis. Supports cannot be changed for different loadcases within a linear analysis.
- For a nonlinear analysis, or a new time step of a transient problem, supports may only be reassigned on a new loadcase increment. All support assignments from previous increments or time steps which are not reassigned will remain unchanged.

Adding Gravity Loading to a Structural Loadcase

Gravity loading can be defined automatically either as a property of an analysis or loadcase, or manually by use of a menu item on the Bridge menu, or by specifying a constant body force load. The letter two require assignment of the gravity attribute to the model

- For an analysis entry in the Analyses Treeview select **Add Gravity** from its context menu. This effectively sets gravity loading to be 'on' for all structural loadcases within that Analysis regardless of whether they previously had gravity loading added or not. For the special case of loadcases having nonlinear controls, gravity loading is only added to those loadcases defined with Manual incrementation and not to loadcases defined with Automatic incrementation because the latter inherit the properties of the preceding defined Manual increment.
- By selecting the the **Automatically add gravity to this loadcase** option on the dialog that is displayed when defining a new loadcase or when editing the properties of an existing loadcase.
- By selecting the **Gravity** menu item from the context menu for an individual loadcase.

No visualisation of gravity loading on the model is provided for gravity defined as a property of an analysis or a loadcase. However, the general loadcase icon will change to include a loading arrow symbol to show that gravity is included for a particular loadcase.

Gravity loading, like other loadings, cannot be inherited from a base or other analysis.

Gravity loading is defined in accordance with the vertical axis direction that was specified either initially on the New Model dialog or subsequently on the Vertical Axis dialog accessed using the **Utilities** > **Vertical Axis** menu item.

Analysis Control Parameters

Analysis control parameters for certain analysis types are set as properties of a loadcase. If no control properties are defined, a linear elastic analysis is performed.

Analysis control parameters can be set to change throughout an analysis by specifying separate loadcases using different control parameters. Note that settings made on loadcases prior to solving will not necessarily correspond to post-processing load increments/time steps due to the use of automatic load incrementation. Analysis controls can be set for the following analysis types:

- **Nonlinear and Transient** Used to define control parameters for nonlinear, and/or transient (dynamic, thermal or consolidation) analyses. Settings must be specified on the first loadcase if they are to be respecified on a subsequent loadcase. It is not possible to change between thermal and dynamic during the course of an analysis.
- **Eigenvalue** Used to define natural frequency, eigenvalue buckling and stiffness analyses. Can be specified on any loadcase, but only once.
- **Fourier** Used to define Fourier analyses. Must be used if Fourier elements have been used. Can only be specified on loadcase 1.

Manipulating Loadcases

General loadcase editing commands are available from the context menu that is activated by right-clicking on a loadcase the Analyses Treeview. The following commands are available:

- **Set active** Sets a loadcase active for viewing attributes assignments and results for the current window.
- **Loadcases to solve** Select which loadcases to solve (if more than one are present)
- **Edit** View and edit the defined loadcase information
- **Copy** Copies the selected loadcase (including all defined loadcase controls for eigenvalue, fourier or nonlinear and transient analysis) in readiness for a paste.
- **Paste** Duplicates the copied loadcase and adds it to the bottom of the relevant section in the Analyses Treeview.
- **Rename** Allow the loadcase title to be modified. Note that loadcases are tabulated and solved in the order listed within each analysis entry in the Analyses Treeview.
- **Delete** Attribute assignments must be deassigned before a loadcase can be deleted. At least one loadcase will always exist in the Treeview.
- **Close Results File** closes the open results file for that loadcase.
- **Controls** Set analysis control parameters for eigenvalue, fourier, and nonlinear and transient analyses.
- **Deassign** Deassigns attributes from the loadcase, by choosing from a list of attribute types. Permits activation and deactivation of the loadcase.
- **Gravity** Apply gravity as a property of the loadcase. See Adding Gravity to a Structural Loadcase for more information.

Solving loadcases

Loadcases are solved in the order in which they are listed within each analysis entry in the Analyses Treeview.

Setting the Active Loadcase

An important concept is the active loadcase in the current window. The active loadcase is a window property, and is the loadcase that all results and visualisations will be generated from. This speeds up the process of comparing results and visualising loads and supports as a different windows can be used for each loadcase.

The active loadcase is set from the Analyses Treeview using the Loadcase context menu. A loadcase icon changes from being greyed-out to coloured when made the loadcase is set active. When modelling, the active loadcase is denoted by a coloured loading icon. When results are loaded the active loadcase is denoted by a coloured contoured results icon.

Viewing the Assignments in a Loadcase

Loadcase dependent attribute assignments such as supports or loading are displayed in the Analyses Treeview under the loadcase to which they have been assigned. The geometry to which an attribute has been assigned may be selected by picking **Select Assignments** from the context menu for each attribute in the Analyses Treeview. The location or extent of an assigned attribute assignment may be visualised for an active loadcase by picking the **Visualise Assignments** from the context menu for that attribute in the Analyses Treeview.

Load Combinations or Envelopes

Combinations and envelopes can be defined as part of the modelling process prior to carrying out an analysis, or as part of the post processing after carrying out an analysis. For more details see **Combinations and Envelopes** in the Viewing the Results section.

Viewing results for loadcases

Viewing results from the **active loadcase** for the current window. See **Visualising the Results** for more details. Loadcases results may be manipulated using **combinations and envelopes**, **fatigue loadcases** and **IMD loadcases** ggg

Load Curves

Load curves can be used to describe the variation of the loading in nonlinear, transient and Fourier analyses. For example, in a transient problem the loading changes with time, in a nonlinear problem the loading level varies with load increment and in a Fourier analysis the loading varies with angle.

Load curves are used to simplify the input of load data in situations where the variation of load is known with respect to a certain parameter. An example of this is the dynamic response of a pipe to an increase of pressure over a given period. The load curve factor would then consist of the variation of pressure with time.

Creating Load Curve Entries

New load curve entries may be added to the Analyses Treeview in the following ways:

- By creating a load curve from the **Analyses> Load Curve**menu item.
- By right-clicking on a Structural or Thermal analysis entry in the Analyses Treeview and selecting the **New Load Curve** menu item.
- By copying and pasting an existing load curve in the Analyses Treeview.

Defining Load Curves

A load curve is defined either using a user defined time vs factor curve, a standard sine, cosine or square wave curve, or a variation.

- ❑ **Time** versus **Factor** specify values for both in a table on the load curve dialog.
- ❑ **Sine, cosine, square wave** input values for amplitude, frequency and phase angle must be defined along with activation and termination points.
- ❑ **Variation** A line interpolation **variation** may be defined from the **Utilities> Variation> Line** menu item. The dependent variable in the variation will represent time (or increment number) depending on the type of analysis. The value of the variation will be the factor by which to scale the values in the assigned loading attribute.

Scaling in the form of activation and termination points for the curve, and a scale factor can also be specified.

Load curves scale all loads assigned to them. Therefore, if loads have a different variation of load factor with time, several load curves should be used.

Manipulating Load Curves

General load curve editing commands are available from the context menu that is activated by right-clicking on a load curve entry in the Analyses Treeview. The following commands are available:

- ❑ **Copy** Copies the selected load curve (including all defined load curve data) in readiness for a paste.
- ❑ **Paste** Duplicates the copied load curve and adds it beneath any current load curves in the Analyses Treeview.
- ❑ **Delete** Deletes a load curve. Attribute assignments must be deassigned before a loadcase can be deleted. At least one loadcase will always exist in the Treeview.
- ❑ **Rename** modifies the load curve name.
- ❑ **Edit** changes previously entered load curve data.

Notes

- Load curves are only applicable to nonlinear, transient and Fourier **analyses**.
- When defining load curves for transient (or nonlinear) analyses the time in all load curves must be defined from the start of the analysis.
- For **Fourier analysis** the load must only be applied over an angular range of 0 to 360 degrees.
- If the interpolation variable doesn't lie within that specified within the load curve a zero load factor will be applied.
- Only line variations with distance type **Actual** can be used for defining load curves.
- Time-based values are based upon currently set **timescale units**.

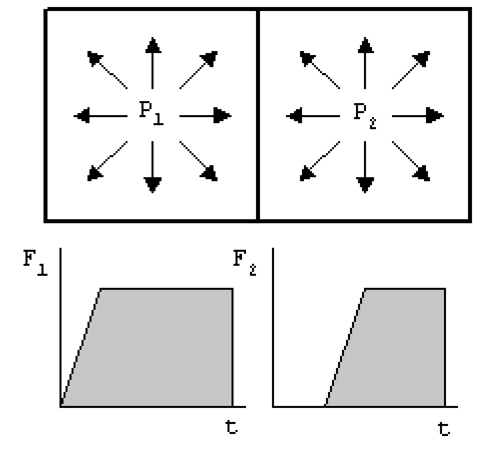
Evaluating a Load Curve

Load curves (and variations) may be viewed using the **Graph Wizard**.

Case Study. Pressurisation of Two Tanks with Multiple Load Curves

Two tanks are to be pressurised at different stages of a nonlinear dynamic analysis. This will be achieved using two different loadcases and two load curves to vary the loads individually. The following procedure outlines the steps required:

1. Use the **Utilities> Load Curve** menu item to define two user defined load curves which give the correct pressure variation with time. Note that the time is always from the start of the analysis.
2. Use the **Attributes> Loading> Structural** menu item to define a face load attribute containing a unit pressure load. Note that the pressure value in the load definition will be multiplied by the load factor used on the load curve associated with it.
3. Assign the face load to the features in the model, selecting the appropriate load curves for each tank. The accompanying diagram shows a schematic of the tanks under internal pressure with their corresponding force versus time graphs.



4. To set the size and number of time steps right click on **Loadcase 1** and choose

Controls> from the context menu. Pick **Nonlinear and Transient** and on the dialog switch on the **Time Domain** option and choose **Implicit Dynamics** from the combo. Set the initial time step and response time as required.

Notes

Supports should be assigned to the loadcase. If the support conditions are to be modified part way through the analysis the response time in the Nonlinear and Transient control should be set to terminate at the time the supports are to be modified and a second loadcase should be created to which the modified supports are assigned. A Nonlinear and Transient control should be then set on this loadcase which terminates at the end of the analysis or when a future support is to be modified.

Chapter 6 : Utilities

Model Utilities

Model utilities differ from model attributes in that they are not intended for assignment to the model geometry or mesh objects. A utility, however, may be used in the definition of model geometry or attributes, or to control an analysis, or to provide a particular functionality, such as to define a load combination or produce a report for example.

Details of the features and use of many of the utilities listed below are provided in other relevant chapters of the manual. Annotation utilities, for example, are described in the Chapter 2: Using Modeller in the Annotating the Model section. The remainder are described in this chapter.

Some Utilities create entries in the Utilities Treeview.

The complete list of LUSAS utilities (accessed from the **Utilities** menu) is shown below.

- **Mesh** - mesh node measurement, the controlling of automatic remeshing. and saving a deformed mesh for re-use in a new analysis
- **Annotation** - adding of text, line, bitmap and border annotation to the view window
- **Transformation** - moving, mirroring and copying of geometry
- **Heat transfer** - specification of thermal gap properties and radiation surfaces
- **Variation**- ways of varying attributes over features
- **Reference Path** - defines a route through the model that provides a concept of distance for positioning multiple varying sections or applying vehicle load patterns.
- Objects to drape - for use with Composites or High Precision Moulding
- **IMDPlus** - the investigation of various dynamic responses using the results from an eigenvalue analysis
- **DesignFactors** - assess the reserve strength capacity of a component or structure
- **Set Fourier Angle** - specify the angle around the circumference at which Fourier results are required
- **Background Grids** - grade the mesh pattern locally when irregular surface meshing
- **Graph Wizard** - plot results on x,y graphs
- **Animation Wizard** - animate the mode shapes or load history of a structure
- **Section Through 3D** - slice a solid model and plot results on the section defined
- **Slice Resultants Beams/Shells** - etc

- **Graph Through 2D** - slice a model and plot a graph based upon the intersection of the elements sliced.
- **Print Results Wizard** - printing of selected results to a grid or a file.
- **User Defined Results** - create results components from user-defined expressions
- **Vertical - Axis** - sets the model X, Y,or Z direction to be the vertical axis.
- **Direction Definition** - sets the vertical, longitudinal and transverse axes for a model to assist with model orientation and calculation of particular effects
- **Renumber All** - renumbering of Nodes and Elements, and Point, Line, Surface and Volume geometry features for all of a model.
- **Renumber Selection** - renumbering of Nodes and Elements, and Point, Line, Surface and Volume geometry features for selected parts of a model.
- **Library Management** - specify library locations and add and delete basic section data from a library
- **Section Property Calculation** - calculate cross sectional geometric properties for a range of sections
- **Report Generation** - build reports containing model and results data and images from your model

Some utilities menu items, for example Heat transfer, are only listed if the appropriate user interface is in use.

Utilities can be transferred between models. See **Transferring Data Between Models.**

Variations

Variations allow parameters in attributes to be varied over the assigned geometry by defining the manner in which the parameter will vary. If a variation is not specified, the parameters within an attribute will be constant over the geometry to which the attribute is assigned.

Note that geometric section property variations along a beam are best defined using tapering or multiple varying section facilities. See **Geometric Properties** and **Multiple Varying Sections - Types and methods of assignment** for more details.

The different types of variation are available are:

- **Field** allowing variations to be defined in terms of the global Cartesian coordinate system variables. This form of variation can be used for hydrostatic and wind loading and is applicable to **all feature types except Points**. Variations on volumes are limited to field variations.
- **Interpolation** variations may be applied to **Lines** and **Surfaces**. The variation is defined by interpolating between values at specified local distances. The order of the interpolation may be specified as constant, linear, quadratic and smoothed (cubic) in either actual (local) or parametric distance.
- **Function** variations are expressed as symbolic functions in terms of the parametric coordinates of a feature. They can be applied to **Lines** and **Surfaces**. For Lines, the parametric distance is the distance along the Line (u), and for a Surface the distances are the local parametric u and v coordinates.

- **Boundary** defines values by specifying variations around the Surface boundary Lines.
- **Grid** defines a grid of values in Surface local x and y directions.

Defining and Using Variations

Variations are defined from the **Utilities** menu and are presented in the Utilities Treeview. Once defined, a variation is used by clicking on the additional input button in the appropriate edit box on the attribute dialogs. This allows each parameter within a single attribute to be varied independently.

Variations can also be defined from inside dialogs that support the use of the variations facility by selecting the **New...** entry on the drop-down list displayed for the Select a Variation Attribute dialog.

Notes

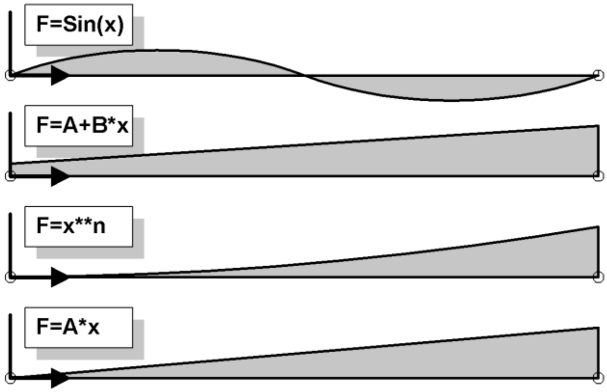
- It is possible to vary all load types except General Point and Patch loads and Internal Beam Point and Internal Beam Distributed loads, which incorporate variable loading implicitly in their definition. Values of loads which are applied to elements will be evaluated at the element centroid.
- Geometric attributes containing a variation are tabulated as multiple geometric properties. An additional parameter is added to the assignment to relate to the original defining attribute number for use in post-processing.
- To vary geometric properties along bar or beam elements use the geometric beam tapering facility.
- Variations in materials are limited to elastic material values and certain joint properties. Attributes containing a variation are tabulated as multiple material properties containing the material value calculated at the element centroid. An additional parameter is tabulated to the assignment data chapter in the data file to relate to the original defining attribute number for use in post-processing.
- When defining supports the spring stiffness values can be varied but the spring stiffness values are not scaled when drawn in post-processing .
- Checking of the assigned variations can be carried out by contouring the assigned data using an unsmoothed contour display.
- Variations of the Rayleigh parameters cannot be contoured as they are calculated at element centroid positions.

Field Variations

Field variations allow a variation according to a mathematical expression in terms of coordinate variables in either the global Cartesian or a specified **local coordinate**. Coordinates may be Cartesian, cylindrical or spherical. The expression may be cutoff if desired.

Field variations are applicable to all Lines, Surfaces and Volumes. The value of the variation at any position on the structure will be calculated by substituting the values of the coordinate variables at that position.

A field variation is defined by specifying a field expression and an optional local coordinate which will be used to specify a coordinate system other than the global Cartesian set.



These examples show field variations expressed in terms of the global X coordinate displayed along a Line parallel to the global X axis. The typical field expressions used are shown in the boxes next to each diagram.

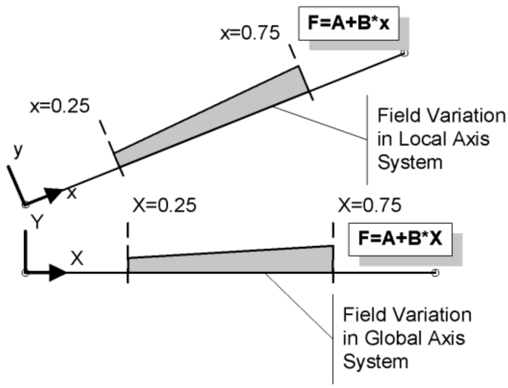
For example, a field expression in Cartesian coordinates would typically be:

```
-9.81*y
```

and in cylindrical coordinates:

```
10+r*tan(thetaz)
```

Coordinate Systems in Field Variations



The functions available in the definition of field expressions are listed below. The variables used in field expressions are limited to those used in the LPI language plus the Cartesian, cylindrical and spherical coordinate variable names. The coordinate variable names that should be used in a field expression are dependent on the type of coordinate systems in use. Definitions are given in the tables below.

In this example, a field expression referring to the global axis coordinates (XY), is also used with a local coordinate axis set (indicated by xy) to create a variation relative to a rotated system. Cylindrical and spherical axis sets can also be used.

Variables

Cartesian (global/local)		Z Cylindrical (local)		Z Spherical (local)	
x	X ordinate	r	Radial distance	r	Radial distance
y	Y ordinate	thetaz	Angle about axis of cylinder	thetaz	Angle about z axis
z	Z ordinate	z	Distance along cylinder longitudinal axis	thetac	Second angle

Operators

+ - * / ^

Functions

Trigonometric functions	Radians	Degrees
Sine of angle	sin(angle)	sind(angle)
Cosine of angle	cos(angle)	cosd(angle)
Tangent of angle	tan(angle)	tand(angle)
Arcsine of a	asin(a)	asind(a)
Arccosine of a	acos(a)	acosd(a)
Arctangent of a	atan(a)	atand(a)
Arctangent of the specified x- and y-coordinates.	atan2(x,y)	
hyperbolic sine of a	sinh(a)	
hyperbolic cosine of a	cosh(a)	
hyperbolic tangent of a	tanh(a)	

Function return value	
e raised to the power of a. The constant e equals 2.71828182845904, the base of the natural logarithm.	exp(a)
natural logarithm of a	log(a)
logarithm of a to base 10	log10(a)
square root of a	sqrt(a)
a rounded up, away from zero, to the nearest integer	ceil(a)
a rounded down, towards zero, to the nearest integer	floor(a)
absolute value of a	abs(a)
maximum value of a and b	max(a,b)
minimum value of a and b	min(a,b)

Function	return value
x to power y	pow(x,y)
remainder of a/b	mod(a,b)

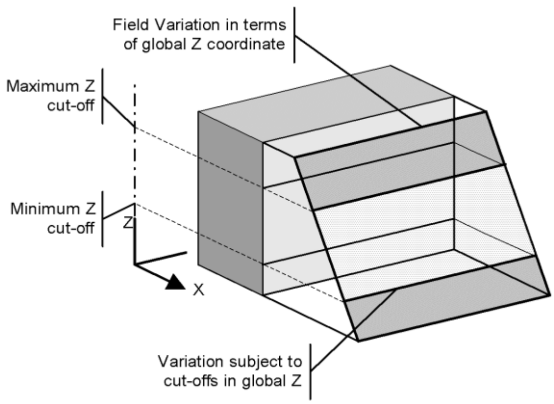
Cylindrical and spherical field variation expressions can use radians (default) or degrees to specify angles. If trigonometric functions are used in a field expression, they will dictate what angular measure is used. For example, a function will use degrees if degree-based trigonometric functions, such as **sind**, **cosd** and **tand** are used.

Notes

- An expression may not mix radian and degree functions.
- Any angle cut-off values will use the same units as the expression.

Maximum and Minimum Cut-Off Values

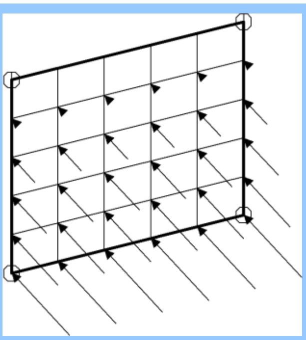
Maximum and minimum cut-off values may be specified for the chosen coordinate system. This allows the range of application of load to be limited, such as would be necessary to model a structure not wholly submerged in water. These examples (right) show field variations in terms of the global X ordinate displayed along a Line parallel to the global X axis. The typical field expressions used are shown in the boxes next to each diagram. All expressions are subject to a cut-off in minimum and maximum X at parametric distances of 0.25 and 0.75 respectively.



The example above shows a variation in terms of the global Z axis ordinate with minimum and maximum cut-offs at specified Z ordinate values.

Case Study. Applying Hydrostatic Loading

A hydrostatic loading may be modelled using a combination of a field variation and a Structural Face Loading. The loading can be considered to be dependent on the depth varying as: water density*g*(h-y) where g is the acceleration due to gravity, h is the height of the water above the structure origin and y is the height of the structure. Use the following procedure:



1. Define a simple 100 unit square Surface using the **Geometry> Surface> By Coordinates** menu item and entering the following coordinates (0,0,0), (100,0,0), (100,100,0) and (0,100,0).

2. Define a simple thin shell mesh using the **Attributes> Mesh> Surface** menu item and **Assign** the mesh to the Surface.

3. Define a field variation using the **Utilities> Variation> Field** menu item and specify a function of density*g*(h-y), where density is the water density (1000), g is acceleration due to gravity (9.81), h is the maximum height of the water above the structure origin (80) and y is the global Cartesian y ordinate. This will apply a hydrostatic loading down the depth of the Surface (global y axis). Enter **1000*9.81*(80-y)** on the dialog.

4. To model a water depth of 80 (and to avoid negative loading above the surface of the water), select a Cut-off in Maximum y at 80. Click on the **Advanced** button and set the maximum second coordinate to **80**. Click the **OK** button.

5. Name the variation **Hydrostatic variation**. Click the **OK** button.

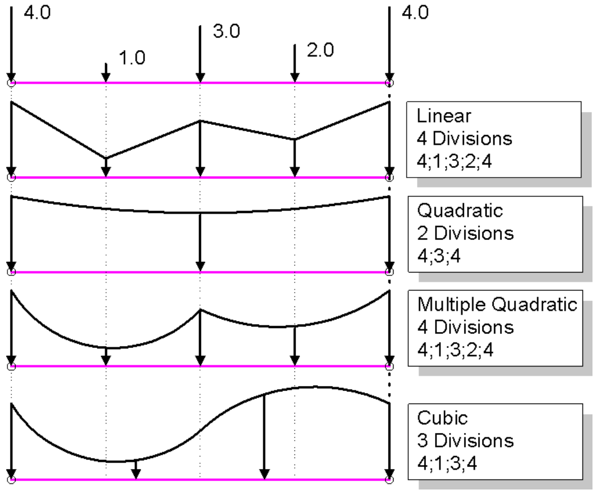
6. Using **Attributes> Loading> Structural** menu item, define a **Local Distributed** load entering the Z component as **1**, notice that in doing so the additional input button appears. Click on the button and select the variation **Hydrostatic variation**. This will factor a negative unit load using the variation defined in 1. Type **Water load** as the attribute title. Click the **OK** button. Assign the loading to the Surface.

7. The applied loading with the variation is visualised as arrows on the model. Use dynamic rotate to get a 3D view of the surface. If the load is not visualised, select the load attribute in the Treeview, right-click and choose Visualise from the context menu. Note. Visualising attribute assignment requires that the model is meshed.

Line Interpolation Variations

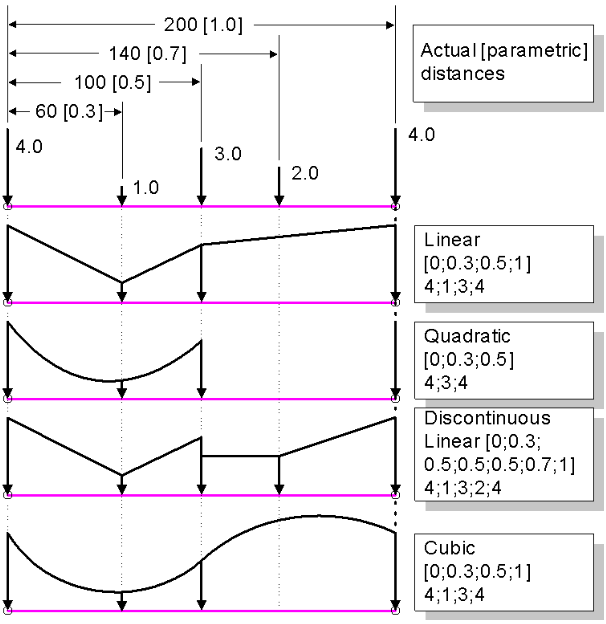
Line interpolation variations allow values to be varied along a line at any number of distances. The distances may be equally or unequally spaced and the interpolation order my be constant, line, quadratic or smoothed (cubic). Line interpolation variations are defined from the **Utilities> Variations> Line** menu item and selecting Type **Interpolation**.

- **By Equal Distances** defines values at equal distances along a Line. The actual value used will be interpolated at the appropriate distance between these values using the interpolation method specified.



- **By Unequal Distances** defines values at specified distances along a Line. The distances can be entered as actual or parametric values. The actual value used will be interpolated at the appropriate distance between these values using the interpolation method specified.

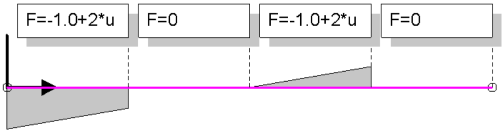
  The unequal distance examples below show user distances specified by actual or parametric values (indicated in square brackets) with a corresponding interpolation value at each position. Repeating a distance and specifying an additional associated interpolation value will allow a discontinuity in the variation to be defined.



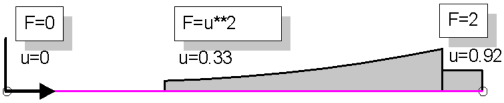
Linear variations require a minimum of two values. Quadratic variations require a minimum of three values and Smoothed (cubic) variations require a minimum of four values to be specified. Where more values are specified multiple interpolation functions are used. i.e. if three values are specified for a linear variation, two straight line interpolations are used.

Line Function Variations

A Line function variation defines a variation by a series of functions specified at distances along a Line. The function is specified in terms of the parametric or actual coordinate along the Line. The interpolated value of the variation at any position along the Line is calculated by finding the interval in which the position occurs and then substituting the parametric or local distance into the function. Line function variations are defined from the **Utilities> Variations> Line** menu item and selecting Type **Function**.



- **By Equal Distances [in u]** defines functions in terms of u, the parametric distance along the Line. In this example, the Line is split into a specified number of distances, each with an associated function.



- **By Unequal Distances [in u]** defines a series of parametric or actual distances, and a set of functions. The distance specified is the starting position for the function associated with it. Each distance must have an associated function specified. To enter a maximum cut-off position, associate a zero function with it. In this example, a parametric distance of 0 is associated with the value 0.0, a parametric distance of 0.33 is associated with u**2 and a parametric distance of 0.92 is associated with the value 2.0

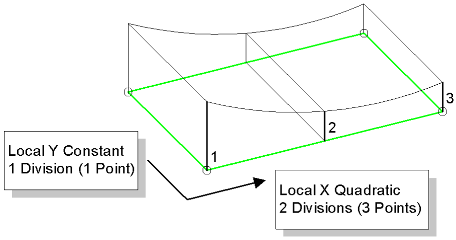
Surface Variations

On Surfaces, interpolation may be defined using a grid of values or a set of line variations applied to the boundary Lines. For interpolation by grid the interpolation order may be constant, linear, quadratic or smoothed (cubic).

- **Surface By Grid** defines a grid of values in Surface local x and y directions. Surface grid interpolation can only be used on 3 and 4 sided Surfaces.

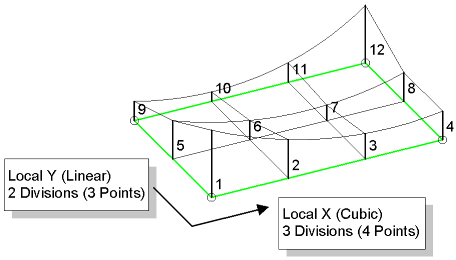
- **Quadratic vs Constant Surface variation**

  The quadratic variation in the local x direction is specified with three interpolation points. The constant variation in the local y direction requires no additional points. A total of three values are required.



- **Smoothed (Cubic) vs. Linear Surface grid variation**

  The local x direction takes a smoothed (cubic) variation defined with four interpolation points and the local y direction takes a linear variation using three interpolation points. A total of twelve values are required.

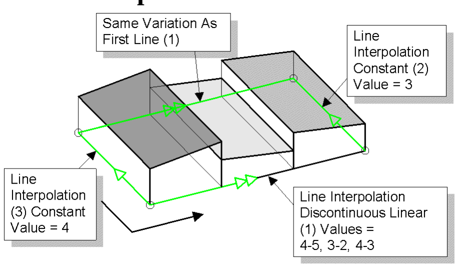


- **Surface By Boundary** defines values by specifying variations around the Surface boundary Lines. A variation must be specified for each Line in the Surface definition. If no variation is required along a Line, a constant order variation must be specified. Care must be taken to ensure that values at common points are common to both variations meeting at that point otherwise an error will occur. Variations are defined in the same direction on opposite sides of the Surface (see the following example) and use the Line order in the Surface definition on which to base the variation direction. Individual Line directions have no effect on variation directions.

- **Surface boundary interpolation using three Line interpolation variations.**

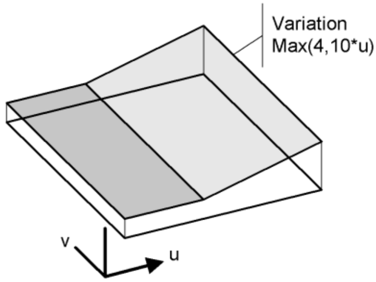
  A discontinuous Line interpolation (1) is specified for first and third Lines using a Line by unequal distance variation.

  Note that the Line axes drawn here dictate the variation directions and the line directions on opposite sides of a surface must match as shown. Variations are applied in a positive surface normal direction.



  The second and fourth Lines in the Surface definition use constant interpolation variations. The variation sense is denoted by double and single arrows shown on boundary Lines. The variation along the local x axis (signified by the double arrow) is specified first. The Surface variation in this case is 1;3;1;2.

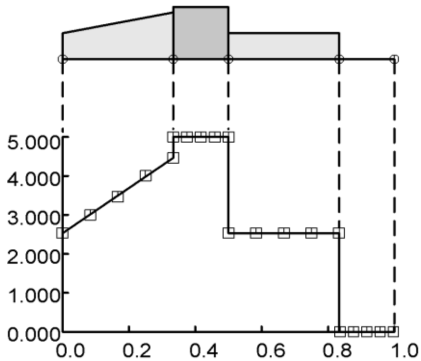
Surface Function Variations



A Surface function variation consists of a single function in terms of the parametric coordinates of the Surface u and v. The value of the variation at any point on the Surface is given by finding the parametric coordinates of the point within the Surface and substituting them into the specified function. Surface function variations are only allowed for 3 and 4 sided Surfaces.

The example shown here defines a variation using the function **max(4,10*u)** in terms of the local Surface x direction parametric distance. The **max** function takes two arguments and returns the maximum of both arguments. In this case, 4 is the maximum value until u exceeds 0.4.

Plotting Graphs of Line & Field Variations



Line and Field variations can be evaluated along a specified Line and displayed using the **Graph Wizard** from the **Utilities** menu and Line variations can be evaluated alone. The number of points at which to sample the variation can be specified. A factor may be applied to the variation values before the ordinates are calculated. This example demonstrates the graphical visualisation of a discontinuous Line interpolation variation.

Reference paths

A reference path defines the direction of a single line, or route that a series of lines take in a model, that can be used to provide a concept of distance to other points in a model. Reference paths are currently used for the following aspects of modelling, analysis and results processing:

- **Modelling of multiple varying sections** For line beam models, and grid or grillage models where longitudinal and transverse beams are modelled with individual grillage or line beam elements a reference path can be used in the definition of a varying section, such that when the section is assigned to a set or series of lines, the path is used to interpret which part of the defined varying section is appropriate to each line.
- **Direct Method Influence Analysis** Reference paths can be optionally used in the definition of a direct method influence attribute (where for grillages they aid in the calculation of influences for longitudinal and transverse members), and also in the definition of a loading grid that can be used as an alternative to using the nodes or points present in the model for an influence analysis.

- **Results Transformation** Specification of a reference path provides an easy way of transforming results relative to the angle or route of the defined path.

Any number of reference paths can be created within a model. Once created, the data that defines each path can be viewed in the Utilities Treeview. Like other utilities, paths are not directly assignable to geometry and can only be edited by editing their properties via their context menu.

Defining paths

Paths are created by using the **Utilities> Reference Path** menu item. They can be defined by

- By defining and selecting lines, arcs and splines in LUSAS Modeller (the most commonly used method).
- Specifying the coordinates of each point defining a line/arc in a table
- Importing geometry and line segment information from a spreadsheet.

A reference path can be defined as a line between two points (if a straight path is to be considered), an arc between three points (if a simple curved path is to be considered), or be created from as many consecutive lines and/or arcs as necessary. Paths can be created either separate from model geometry, or make use of it in the definition by selecting the line (or lines) defining a path prior to selecting the menu item. It is important to remember that whilst model geometry may be used to arrive at the points required to generate a reference path, no connection between the model geometry and reference path data exists.

Reference paths are usually defined to be along and coincident with beam lines. For clarity it is also possible to define reference paths away from beam lines but if the beam lines are not straight (perhaps they curve on plan) the path should be defined above the beam lines rather than be defined in the same horizontal plane.

The way that two adjacent and intersecting reference path lines will be shaped can be controlled by smoothing which involves adding a radius transition between two lines inside of their defined intersection point or adding a radius transition between two lines through their defined intersection point.

Transverse direction settings control how line attributes containing multiple varying sections are assigned to more than one set of lines when using the same common reference path. One example of use is for straight or skewed grid or grillage line beam models. For this type of modelling a single reference path can be used in conjunction with a transition setting to offset multiple tapering section line attributes appropriately for each longitudinal beam member.

A 'Value of distance at start of path' can be used specify the local x value at which the path should begin. For bridge engineering this would equate to specifying a chainage value for a known point at the 'start' of the structure. This value is added to the distance value that can be displayed for each of the points defining the path.

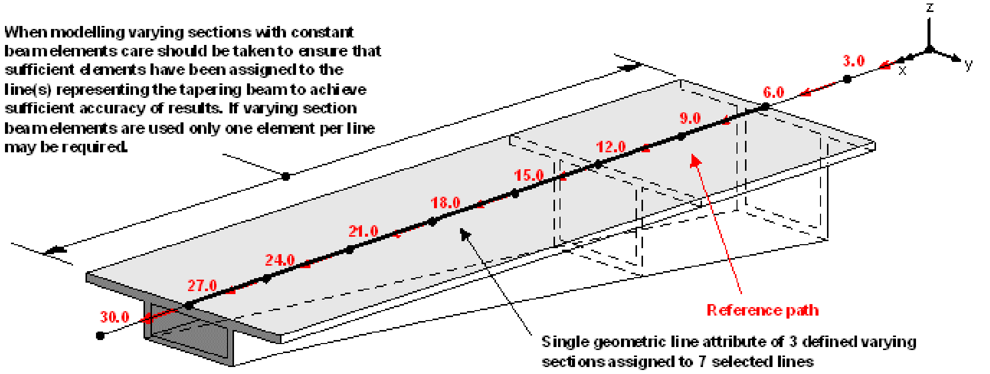
Deleting paths

Defined reference paths can be viewed in the Utilities Treeview. A reference path can be deleted from this location by using the Delete option on its context menu, but only if the path is not used in the definition of any other modelling attributes that are present in the Attributes Treeview, or in the definition of a **Direct Method Influence Analysis**.

Reference paths for use in modelling multiple varying sections

3D line beam models

For 3D line beam models consisting of multiple longitudinal lines a reference path can be used. This allows one multiple varying section geometric line attribute to be assigned to multiple lines. The number of line beams required to model the changing section depends upon whether a staged construction analysis will be carried out. For the creation of simple models it is possible to assign a multiple varying section line attribute to a single line beam without the use of a reference path but for staged construction analysis (where individual lines need to be activated and deactivated) the multiple varying section line attribute can be assigned to multiple lines with reference to an associated path. See **Distance types and methods of assignment** for more information.

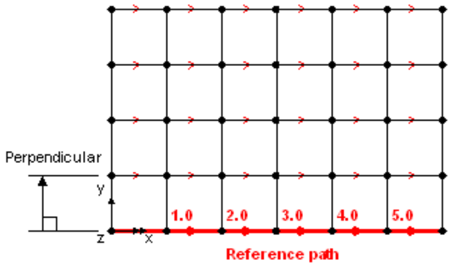


Reference path showing multiple line beams assigned a single multiple varying section (for clarity beam lines have been visualised at top of section)

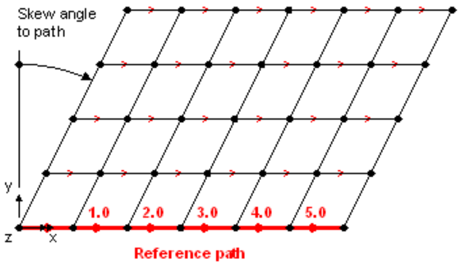
3D grid/grillage-type models

For grid/grillage models, longitudinal beams are comprised of separate line beams often grouped together (for ease of manipulation and assignment of properties etc). Because the actual profile of the set of grouped members as a whole may vary along the longitudinal beam's length a reference path is used to control the assignment of a multiple varying section line attribute to the set of lines. When assigning a geometric line attribute the following transverse direction settings are available:

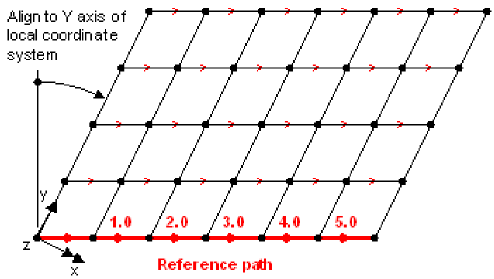
- **Perpendicular to path** - the plane of constant distance is normal to the path tangent in both local y and z directions
- **Skew angle** - is defined as the horizontal angle between the orthogonal plane and the plane of equal distance.
- **Local coordinate** - as skew angle, but where the skew angle is read from an existing local coordinate system.



Perpendicular



Skew angle to path



Local coordinate

Reference paths for Direct Method Influence Analysis

See **Direct Method Influence Attributes**

Reference paths for Results Transformation

See **Results Transformation**

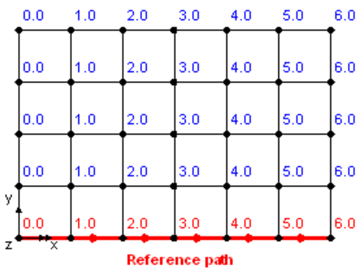
Visualisation of reference paths

By default, reference paths are drawn in red and direction arrows at mid-points along each line segment, show the direction of the path. Each path entry in the Utilities Treeview has a context menu enabling the following selections:

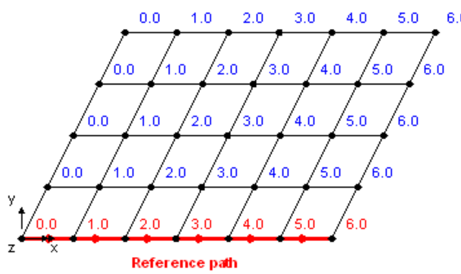
- **Rename** changes the path name
- **Delete** removes the path entry from the Utilities treeview but the menu item is only available if a path has not been associated with a geometric line assignment.
- **Edit Path** displays the path definition dialog to enable changes to be made
- **Create geometry** converts the path into points, lines, arcs and splines in the Geometry layer.
- **Visible** turns the display of the reference path on, if off.
- **Invisible** turns the display of the reference path off, if on.
- **Visualise at points** labels the points defining the path with absolute distances along the path. This also labels other lines in the model to show what would happen if the path were associated with those lines. See below for examples.

Visualisation of reference path at points on the model

The reference path context menu option **Visualise at points** shows the value of the reference path distance for other points in the model. It helps to show the validity of using the reference path for other lines in the model and in cases where path labels drawn on these other points did not match those of the reference path it would, for some situations, draw attention to an invalid transverse direction settings being used.



Perpendicular (orthogonal) grid/grillage



Skewed grid/grillage

Examples of valid reference path points visualised on the model

Vertical axis

The vertical axis dialog is accessed from the **Utilities> Vertical Axis** menu item. The vertical axis setting specifies whether the model X, Y,or Z direction is to be used as the vertical axis. It:

- Defines the initial vertical axis and orientation of element types and section library items as visualised on the Geometric Line dialog prior to them being added to a model.
- Defines the model orientation that is viewed when using the isometric, dimetric and trimetric views.
- Determines the direction that gravity loading will be applied if it has been specified as a property of a loadcase.

- Determines the direction that gravity loading will be applied if added using the **Bridge > Bridge Loading > Gravity** or the**Civil > Gravity** menu item.

Note that setting the vertical axis on the Vertical axis dialog will supercede any vertical axis setting defined on the Direction definition dialog.

Direction Definition

Specifying a direction definition sets the vertical, longitudinal and transverse axes for a model to assist with model orientation and the calculation of particular effects. The Direction definition dialog is accessed from the **Utilities> Direction Definition** menu item. The options are:

Vertical

- **Global axis** This determines the direction that gravity loading will be applied if added using the **Bridge >** or **Civil > Gravity** menu item. It also defines the initial vertical axis and orientation of element types and library items as displayed on the Geometric Line dialog prior to them being added to a model. It also defines the model orientation viewed when using the isometric, dimetric and trimetric views.

Note that setting the vertical axis on the Direction definition dialog will supercede any vertical axis setting defined on the Vertical Axis dialog.

Longitudinal

- **Global axis** For the majority of models this will simply be the global X-axis. However LUSAS allows generic input of any direction which may even be a complex path through or along a structure, such as that defined by a set of lines forming a continuous path.
- **Local axis** Use a local coordinate set to define the direction. An example of use is for aligning influence attributes along a singly curved bridge deck.
- **Follow line path** If a path of lines is to be used, selecting the lines to be used prior to selecting **Utilities > Direction definition** will cause the correct line path definition to be automatically inserted into the line path field. An example of use is for ensuring correct alignment of influence attributes along all spans of a bridge deck when those spans are formed of multiple straight lines, arcs or any combined sequence of these two feature types in order to describe the carriageway shape.

Note that a longitudinal direction definition must always be correctly defined for an **influence analysis**.

Transverse

This is assumed to be orthogonal to both longitudinal and vertical directions. This is currently only used for an **influence analysis**.

Renumbering

By using **Utilities > Renumber All...** or **Utilities > Renumber Selection...** renumbering of Nodes and Elements, and Point, Line, Surface and Volume geometry features (referred to collectively as 'objects' in this topic) can be carried out for all, or selected parts of a model. Renumbering can be done by specifying numerical increments with respect to either global axis directions or **reference paths**.

To visualise node, element and geometry numbers/names add the Labels layer to the Layers Treeview, choosing the object names to be displayed.

Renumbering options

When Renumber All is chosen the following two options are seen:

- **Auto renumber using these settings when re-meshing** If checked, renumbering of all nodes and/or elements in a model will take place according to the settings made on the dialog after remeshing is requested. If not checked, renumbering of all nodes and/or elements in a model will take place according to the default Global direction settings after remeshing. Autorenumbering is always checked 'on' by default when a new model is created. This status is shown by a tick on the Renumber All menu item.
- **Renumber now** applies the renumbering straightway once the Apply or OK buttons are pressed.

The ability to check (turn-on), or not check (turn-off) either of these two options results in either one, both or neither being applied. When both are checked both actions that are described above take place. When both a re unchecked, auto-renumbering is turned-off, as subsequently shown by a greyed-out tick on Renumber All menu item.

When Renumber Selection is chosen only the following option is seen:

- **Preserve numbering in other parts of the model** (for Renumber Selection menu item) If checked, renumbering of only selected objects will take place according to the settings chosen on the Renumber Selection dialog with all other objects retaining their current number. If not checked, numbers used in the re-numbering of the selected objects may use numbers from other parts of the model, and these in turn will then also be re-numbered with previously unused numbers.

Global direction method

Renumbering of objects using the global direction method is carried out with respect to the three model view axes. Positive and negative numbering directions can be stated. The incremental value defines the interval between the numbers assigned to each object within each specified direction.

To help understand the effect of choosing certain renumbering settings note how Modeller evaluates a model prior to carrying out a renumbering operation for selected objects.

1. For the first global axis direction specified, (a positive Z-axis by default), Modeller sweeps through the model in that direction locating all and any objects of equal distance along that axis.
2. For those objects that it finds, Modeller then sweeps along the second global axis specified, (a positive Y-axis by default) locating all and any objects of equal distance along that axis.
3. For the objects that it finds Modeller then sweeps along the third global axis specified, (a positive X-axis by default) locating all and any objects of equal distance along that axis.

Once a set of objects has been assembled, renumbering of objects then takes place using the starting values and specified increments for each axis, but in the reverse order to the axis order chosen. So, for the default settings stated above, renumbering would start and increment along the X axis first, then along the Y axis, and then along the Z axis.

Reference path method

Renumbering of objects using reference paths uses longitudinal, transverse and vertical reference axes in place of the named model axes of the Global direction method. Positive and negative renumbering directions can be stated for each direction considered. The incremental value defines the interval between the numbers assigned to each object within each specified direction.

The reference path method requires specifying longitudinal, transverse and vertical directions and requires specifying positive and negative renumbering directions that may not be initially obvious:

1. For a defined and selected reference path, the positive longitudinal direction is measured along the path, in the direction that the path was defined.
2. The transverse direction is measured orthogonally to the longitudinal direction and the vertical direction.
3. The vertical direction is orthogonal to the longitudinal direction and the plane described by the longitudinal direction and the global vertical direction.

Once a set of objects has been assembled renumbering of objects then takes place using the starting values and specified increments for each axis, but in the reverse order to the axis order chosen.

Notes

- Points and Nodes are located by their coordinates
- 2D Elements and Lines are located by the coordinates of their mid-points.
- 2D Surfaces, 3D Elements and Volumes are located by the coordinates of their centroids.

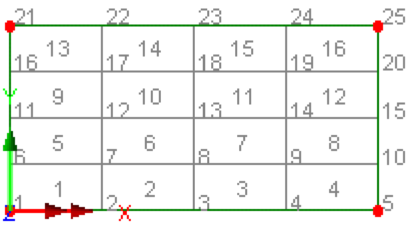
- The renumbering facility can renumber all or just selected objects but the renumbering strategy defined is applied separately to each object type. For instance, renumbering a selection of nodes and elements with a starting value of 1 and an increment of 2 will result in a renumbering sequence of node 1, element 1, node 3, element 3 etc. If different renumbering patterns are required these should be carried out by restricting the type of object selected and carrying out successive individual renumbering passes through the model.
- Auto renumbering is only applied to nodes and elements. If the model has no mesh, the auto renumbering option is disabled.
- Use of the **Utilities> Mesh Reset** menu item and other remeshing events will cause any nodes and elements to revert to their initial numbering, or to a re-numbering scheme as specified on the Renumber dialog if the 'Auto renumber using these settings when re-meshing' check box has been checked.
- For models created prior to Version 15 auto renumbering is turned off by default.

Renumbering examples

In these examples note that for 2D models created in the XY plane, the Z axis direction is redundant and no incrementation settings in the Z axis are applied. The same would be true for models drawn in other planes with other redundant axes. Similarly, when selecting a single row of objects for renumbering (as in the case of the Simple line beam bridge example that is shown) the second and third global directions are effectively redundant.

Simple node and element renumbering

Nodes and element numbers in a 2D surface mesh are renumbered using
Utilities> Renumber All



Before: All nodes and elements numbered using global directions.

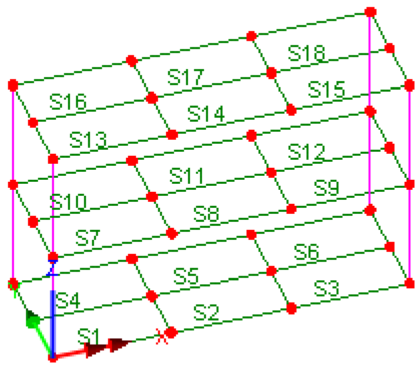
Start value=1, Positive Z-axis, increment 1, Positive Y-axis, increment 1, Positive X axis, increment 1 (Default numbering method)

After: All nodes and elements renumbered using global directions.

Start value=1, Positive Y-axis, increment 10, Positive X axis, increment 1, Positive Z axis, increment 1 - is not used.

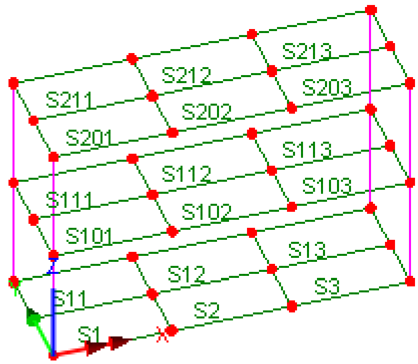
Simple multi-storey building frame

Surfaces are re-numbered in a 3D model using **Utilities> Renumber All**



Before: All surfaces numbered using global directions.

Start value=1, First: Positive Z-axis, increment 1, Second: Positive Y-axis, increment 1, Third: Positive X axis, increment 1 (Default numbering method)

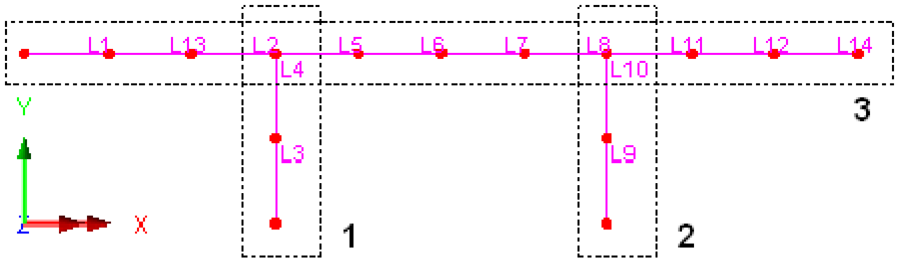


After: All surfaces renumbered using global directions.

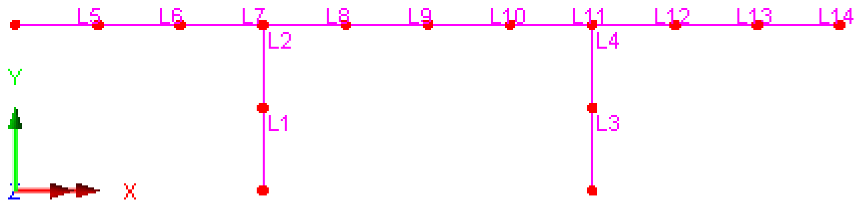
Start value=1 , First: Positive Z-axis, increment 100, Second: Positive Y-axis, increment 10, Third: Positive X axis, increment 1

Simple line beam bridge

Lines are renumbered in a 2D line model using **Utilities> Renumber Selection**.



Before: All lines numbered as drawn. Three separate renumbering selections shown.



After: Renumbered using three steps as follows:

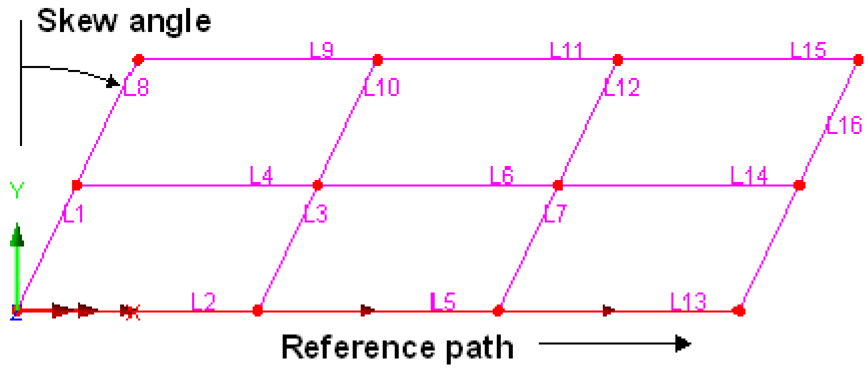
Step 1: Lines 3 and 4 selected (on Before model) and renumbered using Global direction, Start value=1 , Positive Y-axis.

Step 2: Line 9 and 10 selected (on Before model) and renumbered Global direction, Start value=3 , Positive Y-axis, preserving numbering in other parts of model.

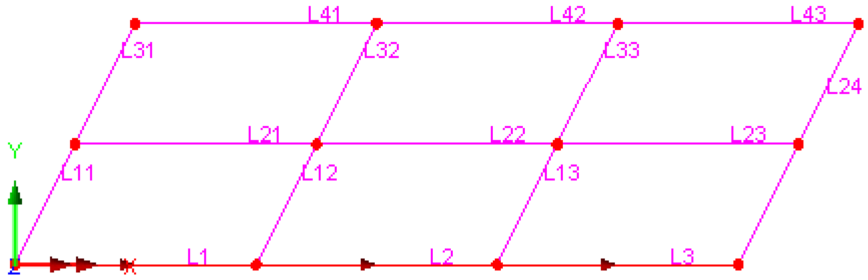
Step 3: Remaining lines (Lines 1 to 14 on Before model) selected and renumbered using Global direction, Start value=5 , Positive X-axis, preserving numbering in other parts of model.

Skewed slab

Lines are renumbered in a 2D line model with respect to a reference path and a skew angle using **Utilities> Renumber All**.



Before: All lines numbered as drawn. Reference path defined along lines L2, L5 and L13

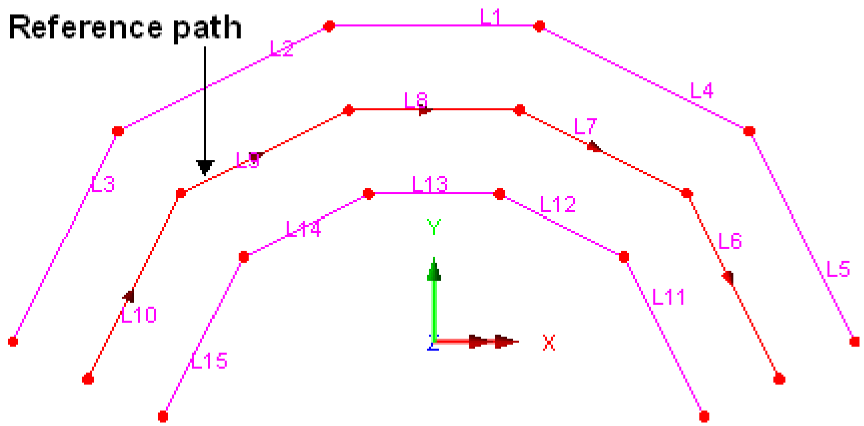


After: All lines renumbered along and transverse to reference path.

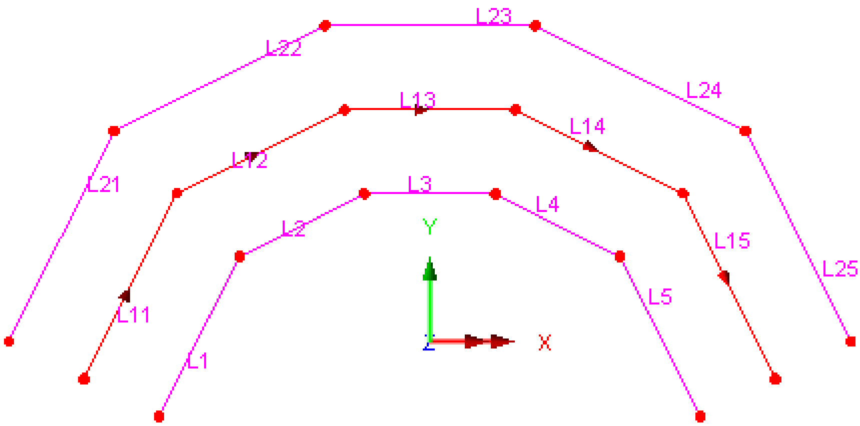
Start value=1 , First: Positive vertical axis, increment 1, Second: Positive transverse axis, increment 10, Third: Positive longitudinal axis, increment 1

Curved path

Lines are renumbered in a 2D line model with respect to a reference path using **Utilities> Renumber All**.



Before: All lines numbered as drawn. Reference path defined along lines L10, L5, L8, L7 and L6



After: All lines renumbered along and transverse to reference path.

Start value=1 , First: Positive transverse axis, increment 10, Second: Positive longitudinal axis, increment 1, Third: Positive vertical axis, increment 1 - is not used.

Section Property Calculation

Cross-sectional geometric properties (for use with line beam models) can be calculated for:

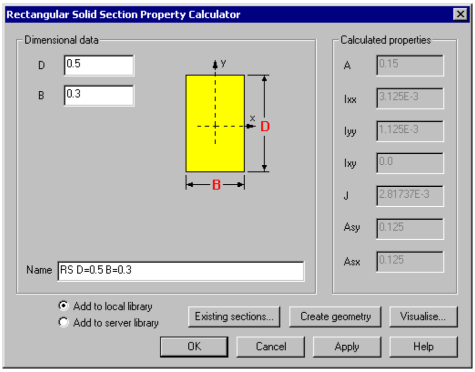
- **Standard sections** - a range of commonly used section shapes
- **Arbitrary sections** - any user-defined cross-sections, with and without internal voids, that are drawn in LUSAS Modeller
- **Precast beam sections** with and without a concrete slab.
- **Box sections** for both simple and complex box sections, with and without an internal void.

Standard Section Property Calculator

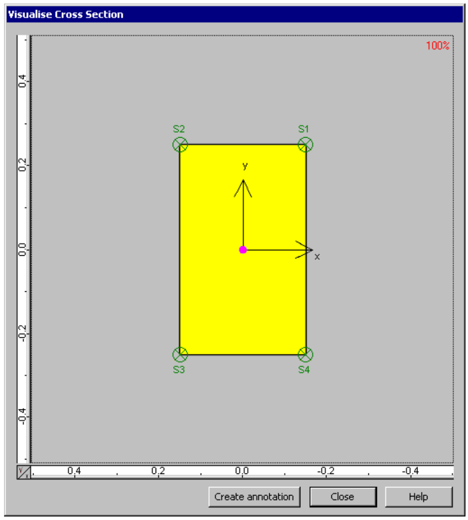
Standard section property calculators are accessed from the **Utilities> Section Property Calculator** menu item. The following sections are supported:

- **Rectangular solid section** - equal and unequal thickness
- **Rectangular hollow section** - equal and unequal flange / web thicknesses
- **Circular solid section**
- **Circular hollow section**
- **I section** - equal and unequal flanges, haunch section
- **T section**
- **L section** - single and double (back to back)

- **C section** - lipped, unlipped, double (back to back), double (face to face), top hat
- **Z section** - lipped right-angle, lipped inclined, unlipped



Typical standard section property calculator dialog



Section visualisation showing fibre locations

Section properties for standard cross-sections are computed instantaneously once valid user-defined dimensional data has been entered. The resulting section can be optionally visualised to verify the section shape created, and to view the automatically defined fibre locations (used when plotting stresses on fleshed beams). The section can also be optionally converted into model geometry and used as a base for further modification in some way inside LUSAS Modeller before re-calculating the new section properties of the edited section using an Arbitrary Section Property Calculator. Sections can be added to a local library (model folder specific) or a server library (typically computer specific) for use on the current project, or for re-use across other projects respectively.

Note that standard section property calculators for standard sections (such as rectangular, circular, L-sections, T-sections etc) and for box sections (both simple and complex), do not currently calculate the additional section properties required for use with design calculations. Instead, after defining a section shape, the **Create geometry** button should be used to draw the section prior to using the arbitrary section property calculator to calculate the full set of section properties for the shape. Note that the additional properties required for design calculations are not required for a general analysis to be carried out.

Using standard sections

To use the computed section properties in a model the section must be saved to a local or server library. To add a library item to the Attributes Treeview select the **Attributes> Geometric> Section Library** menu item, then select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available. The section can then be **assigned** to the required Line(s) in the model.

Arbitrary Section Property Calculator

The arbitrary section property calculator is accessed from the **Utilities> Section Property Calculator> Arbitrary Section** menu item. It computes section properties (area, moments of inertia and torsion constant etc.) of any open or closed section of similar (homogenous) material, or of a compound section made of two or more materials, and calculates extreme fibre positions for use when plotting stresses on fleshed beams.

The arbitrary section property calculator should generally be used in preference to manually defining geometric line property data since cross-sectional shapes, as used by the fleshing facility, and fibre locations are automatically calculated and stored as part of the section property calculation.

Defining cross-sections

Cross-sections must be defined in the View window in the XY plane. Whilst they will normally be created initially using points and lines, the cross-section must ultimately contain surfaces that define a single continuous shape. Individual surfaces separated by gaps do not form a valid cross-sectional shape for section property calculation purposes. See **Creating Holes in Surfaces** for details of how to create sections with voids.

Compound sections constructed of two or more materials can be accommodated using the modular ratio approach. For this an isotropic field material property attribute is created with the thermal conductivity set to the Ei / Ebase value of the material and then assigned to surfaces made from that material. E_i is the Youngs Modulus of the material the surface defines and E_{base} is the Youngs Modulus of the material to be used in the structural analysis. See **Case study: Arbitrary Section Property Calculation of a Compound Section** for a basic procedure.

Options/Settings

On the Arbitrary Section Property Calculator dialog:

- **Annotate properties** displays the calculated section properties and section location data to the Annotation layer.
- **Automatic meshing** lets LUSAS create the model mesh.
- **Max elts/line** specifies the maximum elements to be assigned to any one line, when automatically meshing.
- **Recompute section properties** Recalculates properties if mesh setting or geometry has been modified.
- **Add to local library** Adds the calculated values to the local library when the Apply button is selected
- **Add to server library** Adds the calculated values to the server library when the Apply button is selected.
- **Name** used to identify the section when saved as a library item.

The mesh used to compute arbitrary section properties determines the accuracy of the section properties but also affects the computation time. It has been found that a reasonable result is achieved if at least two elements are used through thin sections of the model. If a finer mesh is required themaximum number of elements per line should be increased. If the mesh still requires adjustment it is recommended that the problem is initially set up using the default mesh and then the **Automatic Meshing** option is switched off to allow the mesh to be manually adjusted.

Calculation of arbitrary section properties is done using thermal/field equations and, as a result, a Thermal Analysis entry will appear in the Analyses Treeview. This is not intended for use or for manipulation by users.

Using arbitrary sections

To use the computed section properties in a model the section must be saved to a local or server library. To add a library item to the Attributes treeview select the **Attributes> Geometric> Section Library** menu item, then select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available. The section can then be **assigned** to the required Line(s) in the model.

The use of the arbitrary section property calculator is described further in the worked example; Arbitrary Section Property Calculation and Use. See *LUSAS Examples Manual.*

Precast Beam Section Generator

The **Precast Beam Section Generator** (for beams with a top slab) is available for Bridge and Civil & Structural software products only. See *Application Manual (Bridge, Civil & Structural)* for details.

Box Section Property Calculator

The **Box Section Property Calculator** is available for Bridge and Civil & Structural software products only. See *Application Manual (Bridge, Civil & Structural)* for details.

Section properties for unspecified cross-sections

Section properties of an unspecified cross-sectional shape may be defined using the **Attributes> Geometric> Line** menu item to enter section properties directly prior to saving in a user or server library. See **Library Locations** for more details.

Case Study: Arbitrary Section Property Calculation of a Compound Section

The basic procedure to carry out section property calculation of a compound section such as that typically encountered with a precast beam and a cast insitu slab, is as follows:

1. Draw the section, assign the beam material to the surfaces or lines representing the beam and the in situ slab initially, run the Arbitrary Section Property Calculator (ASPC) to calculate homogeneous section properties, and save the section to a user library with the name homogenous_section.
2. Use the Attributes > Material > Isotropic menu item to define a new material, setting the Thermal Conductivity, K, to be G1/G2 where G1 is the Shear Modulus of the material <u>to be used</u>, and G2 is the Shear Modulus of the material <u>to be represented</u>, and setting the Specific Heat, C, to be E1/E2, where E1 is the Young's Modulus of the material <u>to be used</u>, and E2 is the Young's Modulus of the material <u>to be represented</u>, These calculated properties will take into account the modular ratios. Note that for isotropic elastic materials, the shear modulus, G=E[2(1+v)].
3. Assign the newly defined properties to the surfaces or lines representing the in situ slab.
4. Re-run the ASPC to calculate the compound section properties, and save the section to a user library with the name compound_section.

Library Files

Library files are used to store geometric section property, material, and composite data for use when building a model. Some library files contain data supplied by LUSAS, and are read-only, others contain data that can be saved by users. The full range of library files are summarised here.

Section property libraries

- **Section library** is a LUSAS supplied library of a range of steel section sizes for various countries.
- **User (local) library** stores user-created section properties for use inside the current project only.
- **User (global) library** stores user-created section properties for use across all projects.

Material property libraries

- **Material library** is a LUSAS supplied library of commonly used structural materials.
- **Composite library** is a LUSAS supplied library of materials for use in the composites industry.

Library Management

The location of the section and material libraries is defined from the **Utilities> Library Management> Library Locations** menu item. The local section library is always located in the current working (project) directory while the server library may be located anywhere on a computer network. The server library allows common sections to be stored and made available across an organisation.

Add Section to Library

Basic geometric section properties may be manually added to either the local or server section library from the **Utilities> Library Management> Section Library> Add Properties**menu item. This facility is intended primarily for entering section data of an unspecified cross-sectional shape.

Geometric section properties for standard cross sections or arbitrary cross sections can be added to either the local or server section library by choosing the appropriate button on the dialog where they are defined.

Delete Section from Library

Section properties may be deleted from either the local or server section library from the **Utilities> Library Management> Section Librray> Delete Properties** menu item.

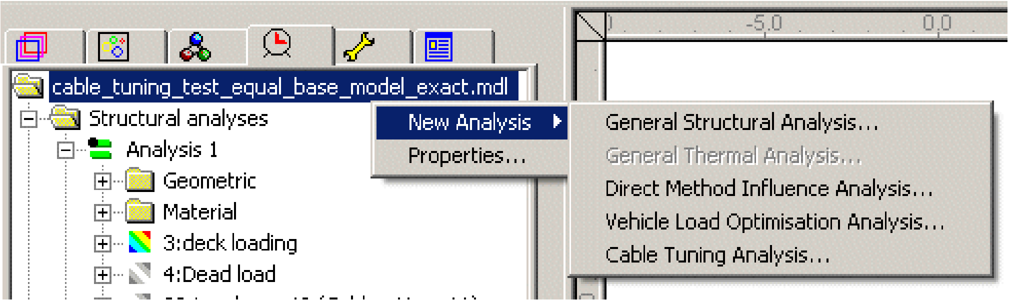
Chapter 7 : Analyses

The Analyses Treeview

The Analyses Treeview provides access to all analysis, loadcase / load curve and post-processing information related to a model. From this panel new analyses can be defined, new loadcases added, solving of selected analyses carried out, and results loadcases can be set active for viewing results.

By default, a base analysis entry is automatically created in the Analyses Treeview by LUSAS Modeller when a new model is defined, or when a model is read from a previous version.

Additional analysis entries can be added to the Analyses Treeview from the using the **Analysis**> menu items. New analysis entries, copies of analysis entries, and other entries can also be added to the Analyses Treeview using context menus.



Analysis entries in the Treeview are listed alphabetically for ease of location but the order has no relevance to the order in which solving of the various analyses takes place. Any analyses that are dependent upon others are automatically solved after those analyses. Analyses entries can be copied and pasted within the Analyses Treeview. Analysis entries cannot be re-ordered.

An analysis entry cannot itself be made active in order to display loading arrangements or to view results. Instead, individual loadcases or load curves within an analysis are set active for a given view window. To identify which analysis contains an active loadcase a black dot is added to the analysis icon. See **Visualising the Results** for more details.

Analysis entries

Analysis entries are held within folders according to analysis type. Folder names are Structural Analysis or Thermal Analysis.

The following analysis entries (showing their default names) may appear in the Structural analysis folder:

- **Analysis 1, 2, 3 etc**
- **Direct Method Influence**
- **Reciprocal Influence Analysis**
- **VLO Analysis 1, 2, 3 etc**
- **Cable Tuning Analysis**

Note that an ASPC analysis entry will also appear when the arbitrary section property calculator is used

Main folder

Context menus for the main model folder provide the means to view model properties and define new analysis types such as General Structural Analysis, General Thermal Analysis, Direct Method Influence Analysis, Vehicle Load Analysis and Cable Tuning Analysis.

Analysis folders

Context menus for the Structural analysis or Thermal analysis folders in the Analyses Treeview provide the means to define new loadcase related entries such as envelopes, basic and smart combinations, and fatigue, IMD and target values loadcases as well as new analysis types. Close All Results files does exactly what it says. Sort by Name lists loadcase and loadcurve names aphabetically for all analyses entries.

Analysis entries

Within each analysis folder the context menu for each analysis entry provides general editing facilities allowing analysis entries to be copied and pasted within the Analyses Treeview, renamed or deleted. Other entries permit selection of which loadcases to solve, the means to add new loadcases, the adding of gravity as a property of an analysis, and various Solver related options.

Copying and pasting analysis and attribute data

Copying an analysis entry in the Analyses Treeview will copy Support and Loading data into the newly created analysis entry. Note that Geometric and Material attribute data, if defined for an analysis, is also copied, otherwise this data is inherited from the Base analysis.

Geometric, Material, Support and Loadcase data can also be copied and pasted individually between analysis entries by the use of their own context menus.

Loadcase entries

Loadcases (a collection of attributes that have been assigned to a particular analysis in a model) and loadcase dependent assignments (such as loading and supports) are displayed within each analysis entry. When solved the results for each loadcase are provided separately and can be accessed by using context menus on each loadcase. See **Loadcases** for more details.

Post-processing folder

The Post processing folder (which only appears when one of the entries on its context menu is created) contains entries that may reference loadcase results data from more than one Analysis. Its context menu provides the means to create envelopes, basic and smart combinations, and fatique, IMD and target values loadcases.

The Combination and envelope results options object allows selected envelope and combination components to be automatically calculated at the end of a LUSAS Solver analysis.

Analyses Treeview Icons

Refer to **Appendix F** for an explanation and details of the status of any Analyses Treeview icons that are seen.

The Analyses Menu

The Analyses menu entries listed here all add entries to the Analyses Treeview by using a similarly named **Analyses>** menu item.

Analysis types

- **General Structural Analysis** - used to carry out all kinds of linear static, eigenvalue, fourier, nonlinear, or transient dynamic analysis.
- **General Thermal Analysis** - used to assess conduction, convection or radiation effects.
- **Direct Method Influence Analysis** - used to carry out influence analysis where the effect of a specified point load is assessed at each node or grid location on a loadable area of a structure. The value of the load effect of interest at each specified location is then used to construct an influence line or surface for that location.

- **Vehicle Load Optimisation** - used to identify critical vehicle loading patterns on bridges and apply these loading patterns to LUSAS analysis models.
- **Cable Tuning Analysis** - calculates load factors for selected lines in a model that represent cables in order to achieve defined target values set for various components or features.

The use of some analyses menu items, such as Vehicle Load Optimisation, or Cable Tuning Analysis is dependent upon the software licence key in use. Note also that some analysis menu permutations are not allowed - for example Cable Tuning analysis and General Thermal Analysis.

Coupled analysis

Coupled Analysis - performs a thermal and structural analysis simultaneously, or one after the other, by transferring data between them via an additional data transfer file. Whilst not accessed via the Analyses menu, a Coupled Analysis can be created by choosing analysis type 'Coupled' when a new model is created, or by adding a structural analysis entry to an Analyses Treeview containing a single Thermal analysis entry or vice-versa.

Load cases and load types

- **Loadcase** adds a new empty loadcase to the Analyses Treeview. Control settings for eigenvalue, fourier and nonlinear and transient analyses can be set using the loadcase context menu.
- **Load Curve** is used to describe the variation of the loading in nonlinear, transient and Fourier analyses.

These facilities can also be accessed using the context menu of an Structural or Thermal analysis entry.

Other entries

- **Envelope** - create an envelope of maximum and minimum effects.
- **Basic Combination** - combine results from different loadcases applying load factors to get max and min effects.
- **Smart Combination** - combine results from different loadcases taking into account beneficial and adverse load factors to get max and min effects.
- **Fatigue** - calculate fatigue life and number of cycles to failure.
- **IMD Loadcase** - create a loadcase for use with interactive modal dynamics.
- **Target Values** - specify a target values loadset of chosen components for selected features in the model.

Each of these facilities can also be accessed using the context menu of the Post processing entry (once it has been created by defining one of the above)

Base Analyses

A base analysis is used a reference for other analyses defined in a single model file and it is identified by a green solve icon alongside its Analysis name in the Analyses Treeview. For models containing both structural and thermal analyses, there are two base analyses – one of each analysis type. If a number of analysis types (that may include a nonlinear analysis) are to be added to the Analyses Treeview it is sensible for the base analysis to be a simple linear static analysis. Other analyses present in the Analyses Treeview that inherit assignments, options and settings from the base analysis are identified by a cyan solve icon.

Creating a base analysis

By default, a base analysis entry is automatically created by LUSAS Modeller when a new model is defined, or when a model is read from a previous version. When a new general structural or general thermal model is created a base analysis will be the only analysis listed in the Analyses Treeview. A Coupled analysis creates two base analysis entries - one for each analysis type.

When additional analyses are defined in the same model by default they will inherit most of the attribute assignments, options and settings etc. of the base analysis, but it is possible to override this default and individually select which attribute assignments, options and settings should be inherited from the base analysis and, by their non-selection, any which should not.

Attribute dependency across analyses / loadcases

The following table summarises how attributes assigned in a base analysis are inherited in other dependent analyses.

Attribute type	Behaviour
Mesh, Local coordinates, Equivalence, Search Area, Crack Tip, Retained Freedom	Assignments apply to all analyses
Constraints, Damping, Geometric, Material (Composites, Age, Damping), Radiation Surface, Thermal Surface, User Attributes	Assigned per-analysis. Optionally inherited from base analysis. Appear in their own folder in the Analyses Treeview. Note that Material entry also controls whether Composite, Age and Damping attributes are inherited or not. Note that radiation surfaces and thermal surfaces, unlike slidelines, cannot change type during an analysis. The User setting only appears if a User attribute has been defined (advanced use only)
Supports	Optionally inherited from first loadcase of base analysis. Supports can be assigned to any loadcase (in a nonlinear model) but note that only supports from the first loadcase can be inherited between analyses. Supports in other loadcases are not inherited.
Loading, Activate, Deactivate	Not inherited between analyses.

Attribute type	Behaviour
Transformation	Not an attribute that is used on a per-analysis basis.
Thermal gap	Not an attribute that is used on a per-analysis basis, but note that thermal gap definitions contain a ‘change’ loadset. The definition dialog checks that all referenced thermal surfaces are assigned in the same analysis, and that this is the same analysis as the change loadset. An error message is generated if this is not the case.
Reciprocal Influence	Assigning a reciprocal influence creates an analysis to contain it.
Direct Method Influence	Can only be assigned with a Direct Method Analysis

Making changes to analyses referencing a base analysis

If any changes are made to attribute assignments, options and settings in an analysis that is linked to a base analysis those specific changes will then override any behaviour it was previously inheriting from the base analysis. For example, consider a model having two analyses with Analysis 1 defined as the base analysis with a "steel material" assigned to all surfaces. By default, Analysis 2 will inherit those material assignments, and "steel material" would appear to be assigned in Analysis 2 as well. But if a different material, say "aluminium", were to be assigned to a single surface in Analysis 2 that single assignment would override the previous "steel material" assignment, but only for that assigned surface. When solving the model all surfaces would be solved with "steel material" properties in both analyses with the exception of the surface that had "aluminium" assigned to it in Analysis 2. This surface would be solved using "steel material" in Analysis 1 and "aluminium" in Analysis 2.

Setting an existing analysis to be a base analysis

Only an existing general structural or thermal analysis entry (as marked by a cyan solve icon in the Analyses Treeview) can be set to be a base analysis. This is done by selecting its context menu and choosing **Edit...** to access the Analysis definition dialog. Note that setting a different analysis as a base analysis can have a significant impact on all other analyses because all inherited attribute assignments will be affected.

Multiple Analyses within a Single Model

Within a single model file multiple analyses may be defined. Each analysis will inherit most of the assignments, options and settings of the **base analysis** for the model. Analyses that inherit from a base analysis are identified by a cyan coloured 'Analysis' icon.

Use of the multiple analyses facility does 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. It also overcomes the need to load results files from a variety of different models on top of one model in order to carry out combinations of loadcases from those different models.

Examples of use

There are many situations where multiple analyses within the one model file should be employed. A few examples are stated here:

- Where static analysis and dynamic analyses are both of significance as, for example, in footbridge design. Multiple analyses would be stored in the same model with typically the same attribute assignments but with different analysis controls and loadcases appropriate to the analysis in question.
- Where static analysis and influence analysis are both of significance, as in bridge design.
- Where it is required to combine load effects derived from models that are identical in terms of mesh, but have some attributes changed, such as the use of long term concrete modulus for some loadcases and short term concrete modulus for others.
- Where superposition of effects with different material properties needs to be considered as, for example, with linear analyses of any beam and slab deck structure where effects arising over long term and short term need to adopt appropriate Young's Modulii of concrete.
- For buckling analysis of symmetrical structures, where the unstressed deformed mesh derived from an eigenvalue buckling analysis (with a magnitude multiplier) is typically required as the starting point for a subsequent nonlinear analysis.
- For sensitivity analyses, such as soil-structure interaction analyses where a range of analyses could represent clone models with differing material properties, support stiffnesses etc. Results from those analyses could be enveloped to ensure that design makes adequate allowance for uncertainties.
- For pushover analysis (a static-nonlinear analysis method where a structure is subjected to gravity loading and a monotonic displacement-controlled lateral load pattern which continuously increases through elastic and inelastic behavior until an ultimate condition is reached) may require several analyses to achieve the analysis goal.

General Structural / Thermal Analysis Types

LUSAS may be used to numerically model a wide range of engineering problems. The following section briefly explains the analysis types available.

Default analysis

- ❑ **Linear Analysis** is the most common analysis carried out by engineers and unless specified otherwise, LUSAS will perform a linear elastic, static analysis or a **steady state thermal/field** analysis (according to the user interface in use). In these types of analysis multiple loadcases can be accommodated but the model geometry and other boundary conditions cannot be altered. Linear analysis assumes that:

- The loads are applied instantaneously and transient effects are ignored.
- The loaded body instantaneously develops a state of internal stress so as to equilibrate the total applied loads.
- The structural response is linear, i.e. both the geometric and material response are assumed to be linear.

Analyses defined as a property of a loadcase

The solving of analyses other than a linear analysis or a steady state thermal/field analysis requires an analysis control to be specified. Analysis controls are specified as a property of a loadcase. Loadcases can be seen and are accessed within each analysis entry in the Analyses Treeview.

The control parameters for the following analysis types are specified as properties of a loadcase.

- **Nonlinear Analysis** is used to model significant changes in geometry, material or boundary conditions. Significant geometry deformation may occur due to the applied loading. Changes in material may occur due to material yield. Changes in boundary conditions may occur due to the lift-off of supports or from changes in contact or frictional behaviour. Examples of nonlinear analyses include **Creep Analysis** and **Impact Dynamics**.
- **Transient analysis** is used to carried out analyses over a period of time and is progressed in a step-by-step manner, giving results at each time-step. Both Transient Dynamic Analysis and **Transient Thermal Analysis** are available.
- **Eigenvalue Analysis** is available to compute the **Natural Frequencies** of a structure or to carry out an Eigenvalue Buckling Analysis in order to estimate the maximum load that can be supported by a stiff structure prior to structural instability. **Eigenvalue Stiffness** may also be performed on the stiffness matrix at a selected stage of an analysis. This facility can be used in conjunction with a nonlinear analysis to predict structural instability or bifurcation points during a geometrically nonlinear analysis.
- **Fourier Analysis** provides an extended form of axisymmetric analysis where applied loading can be considered to be non-axisymmetric when applied using a Fourier distribution around the circumference.
- **Thermo-Mechanical Coupled Analysis** either performs the thermal and structural analyses simultaneously or one after the other with transfer of data between them via an additional data transfer file.

Solver-based analyses

The following analysis types are also possible in Solver but the tabulation of the harmonic analysis control is not fully supported by LUSAS Modeller:

- **Harmonic Response Analysis** The behaviour of a structure subjected to vibrating loads can be analysed without the need for a full dynamic step-by-step analysis. See also **Modal Response analysis**.
- **Temperature dependent materials** The definition of temperature dependent materials in a tabular form are supported by LUSAS Solver. See *Solver Reference Manual* for details.

Specifying initial deformations for an analysis

When defining an analysis, an undeformed or deformed mesh of a chosen loadcase or load increment, time step, or eigenvalue from one analysis can be used as the starting point for the 'undeformed' mesh of another general or thermal analysis. The coordinates of all nodes in the mesh can be adjusted based upon a specified deformed mesh factor. Initial deformations are specified as a property of an analysis.

Nonlinear Analysis

Nonlinear analysis controls are defined as properties of a loadcase. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item.

The Nonlinear and Transient object created provides the means to modify the control settings. Additional Nonlinear analysis control options are accessed via the Nonlinear analysis options control object in the Analyses Treeview. Nonlinear analysis options can be specified separately for each analysis present.

What is Nonlinear Analysis?

Linear finite element analysis assumes that all materials are linear elastic in behaviour and that deformations are small enough to not significantly affect the overall behaviour of the structure. Obviously, this description applies to very few situations in the real world, but with a few restrictions and assumptions linear analysis will suffice for the majority of engineering applications.

The following indicate that a nonlinear finite element analysis is required:

- Gross changes in geometry
- Permanent deformations
- Structural cracks
- Buckling
- Stresses greater than the yield stress
- Contact between component parts

Good practice when carrying out nonlinear analysis

It is good practice to perform a linear static analysis prior to carrying out a nonlinear analysis. This eliminates the added complexities of the nonlinear variables and will enable a check on the basic performance of the structure for the applied loadings. Any warning or error messages in the LUSAS output file should be investigated.

It is advisable to add in nonlinear behaviour in stages. For example in a material and geometrically nonlinear run containing slidelines it would probably be advisable to start with only slidelines, then add the geometric nonlinearity and finally add the nonlinear material effects. By proceeding in this manner will ensure that each nonlinear procedure is stable before progressing to the next.

Nonlinear Analysis Types

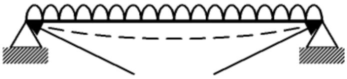
Three types of nonlinear analysis may be modelled using LUSAS:

- ❑ **Geometric Nonlinearity** e.g. large deflection or rotation, large strain, non-conservative loading.
- ❑ **Boundary Nonlinearity** e.g. lift-off supports, general contact, compressional load transfer, dynamic impact.
- ❑ **Material Nonlinearity** e.g. plasticity, fracture/cracking, damage, creep, volumetric crushing, rubber material.

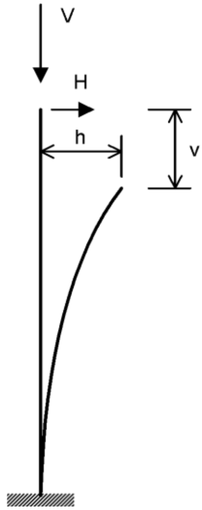
The LUSAS analysis types within which nonlinear geometric and material effects may be incorporated are shown in the following table:

Analysis Type	Geometric Nonlinearity	Material Nonlinearity
Static	yes	yes
Dynamic	yes	yes
Thermo mechanical	yes	yes
Creep	yes	yes
Natural Frequency	yes	yes
Eigenvalue Buckling	-	-
Spectral Response	-	-
Harmonic Response	-	-
Fourier Analysis	-	-
Field or Thermal	-	yes

Geometrically Nonlinear Analysis



Geometric nonlinearities arise from **significant changes in the structural configuration** during loading. Common examples of geometric nonlinearity are plate structures which develop membrane behaviour, or the geometric **bifurcation** of truss or shell structures. The changing application of loads or **boundary conditions** are also geometrically nonlinear effects. The figure below shows two simple structural examples which serve as good illustrations of geometrically nonlinear behaviour.



For the simply supported beam (top) the linear solution would predict the familiar simply supported bending moment and zero axial force. In reality as the beam deforms its length increases and an **axial component** of force is introduced.

For the loaded strut (bottom) the linear solution would fail to consider the **progressive eccentricity** of the vertical load on the bending moment diagram.

In both these cases depending on how large the deflections were, serious errors could be introduced if the effects of nonlinear geometry were neglected.

In LUSAS geometric nonlinearity is accounted for using four basic formulations:

- ❑ Total Lagrangian
- ❑ Updated Lagrangian
- ❑ Eulerian
- ❑ Co-rotational

These are defined from the **Model Properties> Solution** - Nonlinear Options tab. All four formulations are valid for arbitrary large deformations. In general, if rotational degrees of freedom are present, rotations must be small for Total Lagrangian. An exception to this rule is the Total Lagrangian formulation for thick shell elements where large rotations may be applied. Large rotations are allowed for Updated Lagrangian (provided that they are small within each load increment) or Eulerian. The co-rotational formulation is unconditionally valid for large rotations and results are generally independent of load step size.

All formulations are valid for small strains. For some elements the Updated Lagrangian formulation is valid for moderately large strains. The Eulerian formulation is also generally valid for moderate strains. In general, the Total Lagrangian is a more robust formulation, which is usually able to cope with substantial load increments. The Updated Lagrangian, and particularly Eulerian, formulations generally require smaller load increments in order to avoid a divergent solution.

Standard geometrically nonlinear formulations account for the change in position of the loading, but not the change in direction relative to the deformed configuration. Loading is always **conservative** for the Total Lagrangian geometrically nonlinear formulations (that is, the load is always applied in the same direction as was initially prescribed). Using an Updated Lagrangian formulation, the geometry is updated at the end of each increment and the applied loads may maintain the same relative orientation as to the original surface (depending on element and load types). Therefore **non-conservative** loading can be increment size dependent. True non-conservative loading may only be achieved by using the Eulerian and co-rotational formulations.

The choice of particular formulations is both problem and element dependent (the element formulation determining which strain formulations are available). The availability of each formulation is given for each element in the *Element Reference Manual*. For further details regarding the geometrically nonlinear formulations refer to the *Theory Manual*.

Nonlinear Boundary Conditions

Deformation dependent boundary condition models account for the modifications to the external restraints resulting from support lift-off, or smooth or frictional contact within an analysis. Within LUSAS node on node contact may be accounted for using **joint** elements and arbitrary contact may be accounted for using **slidelines**.

Consider the simple example shown in the figure right in which the structure and its supporting surface can resist being pushed together, but not being pulled apart. The required contact condition may be imposed by using joint elements to connect between the structure and the rigid support, and specifying a nonlinear contact joint model incorporating large and zero local stiffness in compression and tension respectively.

Materially Nonlinear Analysis

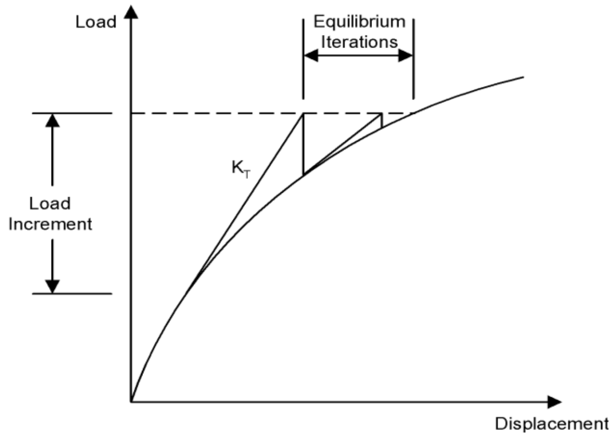
Materially nonlinear effects arise from a **nonlinear constitutive model** (that is, progressively disproportionate stresses and strains). Common examples of nonlinear material behaviour are the plastic yielding of metals, the ductile fracture of granular composites such as concrete, or time-dependent behaviour such as creep.

LUSAS incorporates a variety of nonlinear constitutive models, covering the behaviour of the more common engineering materials. Details of these material models and their applicability to each LUSAS element are described in **About Material Properties** which should be read in conjunction with the *Element Reference Manual*.

Nonlinear Solution Procedures

For nonlinear analysis, since it is no longer possible to directly obtain a stress distribution which equilibrates a given set of external loads, a solution procedure is usually adopted in which the total required load is applied in a number of **increments**.

Within each increment a linear prediction of the nonlinear response is made, and subsequent iterative corrections are performed in order to restore equilibrium by the elimination of the residual or out of balance forces.



The iterative corrections are referred to some form of **convergence** criteria which indicates to what extent an equilibrium state has been achieved. Such a solution procedure is therefore commonly referred to as an **incremental-iterative** (or **predictor-corrector**) method shown in the figure above. In LUSAS, the nonlinear solution is based on the Newton-Raphson procedure. The details of the solution procedure are controlled using the nonlinear control properties assigned to loadcase.

For the analysis of nonlinear problems, the solution procedure adopted may be of significance to the results obtained. In order to reduce this dependence, wherever possible, nonlinear control properties incorporate a series of generally applicable default settings, and automatically activated facilities.

Iterative Procedures

In LUSAS the incremental-iterative solution is based on Newton-Raphson iterations. In the Newton-Raphson procedure an initial prediction of the incremental solution is based on the **tangent stiffness** from which incremental displacements, and their iterative corrections may be derived.

Standard Newton-Raphson Procedure

In the standard Newton-Raphson procedure each iterative calculation is always based upon the **current tangent stiffness**. For finite element analysis, this involves the formation (and factorisation) of the tangent stiffness matrix at the start of each iteration.

Although the standard Newton-Raphson method generally converges rapidly, the continual manipulation of the stiffness matrix is often expensive. The need for a robust yet inexpensive procedure therefore leads to the development of the family of modified Newton-Raphson methods.

Iterative Acceleration (Line Searches)

A slow convergence rate may be significantly improved by employing an **iterative acceleration** technique. In cases of severe and often localised nonlinearity, encountered typically in materially nonlinear or contact problems, some form of acceleration may be a prerequisite to convergence.

In LUSAS, iterative acceleration may be performed by applying **line searches**. In essence, the line search procedure involves extra optimisation iterations in which the potential energy associated with the residual forces at each iterative step are minimised. Line search application is controlled via parameters on the Iteration section of the Nonlinear Control properties.

The selection of line search parameters is problem dependent and largely a matter of experience. However, a maximum of 3 to 5 line search iterations with a tolerance of 0.3 to 0.8 is usually sufficient (the closer the tolerance is to unity, the more slack the minimum energy requirement).

Separate Iterative Loops

In problems where both material and contact nonlinearities are present, convergence difficulties can arise when evaluating material nonlinearities in configurations where the contact conditions are invalid because the solution is not in equilibrium. To avoid this situation contact equilibrium can be established using elastic properties from the previous load increment before the material nonlinearity is resolved. The option to define separate iterative loops is defined on **advanced solution strategy** dialog which can be found on the nonlinear control dialog.

See the *Theory* Manual for further details.

Incremental Procedures

For the Newton-Raphson solution procedures it is assumed that a displacement solution may be found for a given load increment and that, within each load increment, the load level remains constant. Such methods are therefore often referred to as **constant load level** incrementation procedures.

However, where **limit points** in the structural response are encountered (for example in the geometrically nonlinear case of **snap-through** failure) constant load level methods will, at best, fail to identify the load shedding portion of the curve and, at worst, fail to converge at all past the limit point. The solution of limit point problems therefore leads to the development of alternative methods, including displacement incrementation and constrained solution methods.

Constrained Solution Methods (Arc-Length)

Constrained methods differ from constant level methods in that the load level is not required to be constant within an increment. In fact the load and displacement levels are constrained to follow some pre-defined path.

In LUSAS two forms of arc-length method have been implemented:

- ❑ **Crisfields** modified arc-length procedure in which the solution is constrained to lie on a spherical surface defined in displacement space. For the one degree of freedom case this becomes a circular arc.
- ❑ **Rheinboldts** arc-length algorithm which constrains the largest displacement increment (as defined by the predictor) to remain constant for that particular increment.

The use of the arc-length method has the following advantages over constant load level methods

- Improved convergence characteristics
- Ability to detect and negotiate limit points

In LUSAS, control of arc-length solution procedures is via the Incrementation section of the Nonlinear Control properties. If required, the solution may be started under constant load control, and automatically switched to arc-length control based on a specified value of the current stiffness parameter (defined as the scaled inner product of displacements and loads). The required stiffness parameter for automatic conversion to arc-length control is input in the Incrementation section of Nonlinear Control properties.

Where limit points are encountered LUSAS will automatically determine the sign of the next load increment by the sign of the **determinant of the stiffness matrix**. This is a reliable method in most cases, however, it will often fail in the vicinity of bifurcation points when negative eigenvalues may cause premature unloading. In such cases the load reversal criteria may be optionally changed to be dependent on the sign of the **current stiffness parameter**. This method is better at coping with bifurcation points, but will always fail when a **snap-back** situation is encountered.

Note. In certain circumstances, notably in the presence of strain-softening, the arc-length method may converge on alternative, unstable equilibrium paths.

Bracketing Critical Points And Branch Switching

Bracketing can be used to locate a limit or bifurcation point during a geometrically nonlinear analysis. The nonlinear analysis is executed and one of three methods is used to isolate the first critical point.

- ❑ **Bi-section**
- ❑ **Interpolation**
- ❑ **Riks semi-direct approach**

Two further options for the bracketing procedure exist depending on whether the material response is elastic (reversible) or plastic/path dependent (irreversible). Only the first critical point can be processed and a subsequent eigenvalue analysis must be invoked to determine whether the critical point encountered is a limit or bifurcation point. A limit point may be defined as the point at which load starts to decrease as displacements increase (e.g. the snap through of a shallow arch). In this instance the structure may not have failed completely and could subsequently still be capable of carrying more load. A bifurcation point indicates that the solution of the nonlinear differential equations has encountered an alternative unstable solution path or paths which may be followed instead of the stable equilibrium path. The branch switching procedure must then be undertaken if an unstable equilibrium path is to be followed.

The branch switching procedure should only be carried out within a restart analysis after bracketing has been successfully completed. Two options exist for guiding the solution onto a secondary path.

- **Eigenmode injection**
- **Artificial force and Rheinboldts arc**

Incremental Loading

Incrementation for nonlinear problems may be specified in four ways:

- **Manual Incrementation** where the loading data in each load increment is specified separately.
- **Automatic Incrementation** where a specified loadcase is factored using fixed or variable increments.
- **Mixed Incrementation** Mixed manual and automatic incrementation.
- **Load Curves** where the variation of one or more sets of loading data is specified as a graph of load factor vs. load increment or time step.

The choice and level of incrementation will depend on the problem to be solved.

Automatic Load Incrementation

Two methods of automatic incrementation are available:

- **Uniform Incrementation** By default, uniform incrementation will be applied. That is, for each increment the current load factor will be multiplied by the specified load components to generate the applied load.
- **Variable Incrementation** Alternatively, variable incrementation may be requested. In this case the current load factor will be automatically varied according to the iterative performance of the solution. The variation is a function of the required number of iterations and a specified desired iterative performance. Thus, where the number of iterations taken is less than the desired value the incremented load factor will subsequently be increased, and conversely, if the number of iterations is greater than the desired value, it will be decreased. Variable incrementation may be used in conjunction with either constant load level or arc-length solution methods and is an effective way of automatically adapting the performance of the solution procedure to

the degree of nonlinearity encountered. The overall effect is therefore to increase and decrease the numerical effort in the areas of most and least nonlinearity respectively.

Mixing Manual And Automatic Incrementation

If required, manual and automatic incrementation procedures may be mixed freely. When mixing manual and automatic incrementation the following rules apply:

- Loadcases may be respecified as often as required.
- If the automatic procedure is specified, it will continue until one of the termination criteria is satisfied.
- In switching from manual to automatic control, any loading input under the manual control is remembered and held constant, while the automatic procedure is operating.
- In switching from automatic back to manual control, any loading accumulated under automatic control is forgotten and must be input as a manual loadcase if required.
- If prescribed displacements are being used, then in any switching from one type of control to another, the effect of prescribed displacements will be remembered and will not need to be input again.

Automatic Increment Reduction

Where an increment has failed to converge within the specified maximum number of iterations it will be automatically **reduced and re-applied**. This will be repeated according to values specified in the step reduction section (Advanced Nonlinear Incrementation Parameters dialog) until the maximum number of reductions has been tried. In a final attempt to achieve a solution the load increment is then increased to try and step over a difficult point in the analysis. If after this the solution has still failed to converge the solution terminated.

Solution Termination

When using manual incrementation, the solution will automatically terminate following execution of one increment. With automatic incrementation, the solution progresses one Nonlinear Control chapter at a time. The finish of each Nonlinear Control chapter is controlled by its Termination parameters.

Termination may be specified in 3 ways:

- ❑ **Limiting the maximum applied load factor.**
- ❑ **Limiting the maximum number of applied increments.**
- ❑ **Limiting the maximum value of a named freedom.**

Where more than one criteria is specified, termination will occur on the first criteria to be satisfied.

Failure to converge within the specified maximum number of iterations will either result in a diagnostic message and termination of the solution or, if automatic incrementation is being used, a reduction of the applied load increment. If required, the solution may be continued from an unconverged increment (Option 16, 17), although the consequences of such an action should be appreciated.

In addition, the solution will be terminated if, at the beginning of an increment, more than two **negative pivots** are encountered during the frontal elimination phase.

Use of Load Curves

Load curves are used to simplify the input of load data in situations where the variation of load is known with respect to a certain parameter. An example of this could be the dynamic response of a pipe to an increase of pressure over a given period. The load curve would consist of the definition of the load and its variation with time.

Nonlinear Solution Convergence Criteria

The convergence criteria specifies to what extent the numerical iterative procedure has reached the true equilibrium state. The specification of convergence therefore involves two considerations:

- Type of convergence criterion.
- Convergence tolerance.

The types of convergence criteria incorporated in LUSAS are as follows:

- **Absolute residual norm**
- **Root mean square residual norm**
- **Displacement norm**
- **Residual force norm**
- **External work norm**
- **Incremental displacement norm**

The convergence tolerance for each criteria is specified in the Solution parameters and advanced solution parameters section of the Nonlinear Control properties. The selection of a convergence criteria, and the associated tolerance, is problem dependent. However, the following points should be considered:

Clearly, the convergence criteria must not be too slack so as to yield an inaccurate solution, nor too tight so as to waste computer time performing unnecessary iterations. In general, sensitive geometrically nonlinear problems require a tight convergence criteria, whereas with predominantly materially nonlinear problems, larger local residuals may be tolerated.

Where more than one criteria has been specified, convergence will be assumed only on the satisfaction of **all** specified tolerances.

The following considerations apply to individual convergence parameters:

- **Absolute Residual Norm** is of limited use owing to its dependence upon the units being used. It is a strict criteria and for some problems, especially those involving plasticity, it may be very difficult to reduce locally large residuals. However, in sensitive geometrically nonlinear problems near bifurcation points, it can sometimes be necessary to ensure that large **residuals** are completely eliminated.

- **Root Mean Square Residual Norm** is the square root of the average of the squares of the residual forces and is generally more applicable than the above, but is still dependent upon the units being used.
- **Displacement Norm** is the sum of the squares of all the iterative displacements as a percentage of the sum of the squares of the total displacements and is a useful measure of how much the structure has moved during an iteration. Being a scaled norm it is not affected by units but convergence is not guaranteed. Typical values of slack and tight norms are (5.0 - 1.0) and (0.1 - 0.001) respectively.
- **Residual Force Norm** is the sum of the squares of all the residual forces as a percentage of the sum of the squares of all the external forces. This is the most versatile of the five criteria. Typical slack and tight values are (10.0 - 5.0) and (0.1 - 0.00001) respectively.
- **External Work Norm** is the work done by all the residuals acting through the iterative displacements, as a percentage of the work done by the loads on iteration zero of the increment. Since all freedoms are considered it is very versatile (the default displacement and force norms consider only the translational freedoms). However, it should be noted that a minimum detected potential energy need not necessarily coincide with the equilibrate state. Typical values of slack and tight norms are (0.1 - 0.001) and (10-E6 - 10-E9) respectively.
- **Incremental Displacement Norm** is the sum of the squares of all the iterative displacements as a percentage of the sum of the squares of the total displacements for the increment. This norm is an incremental form of the total **displacement norm** previously described and the same comments regarding usage apply.

Nonlinear Output Control

Nonlinear analyses may generate a vast amount of output. In addition to the normal nodal and element output controls, the frequency of nonlinear solution output may be restricted via the Output section in the Nonlinear Control properties.

The restart output facility enables failed or terminated analyses to be restarted from the last saved restart output dump. This is particularly useful where the termination of the analysis was due to a failure of the solution process rather than that of the structure. In this way, the solution may be restarted from the last converged increment with a different or modified solution strategy. For example, a failed increment may be restarted under either constant load or arc-length control. Restarts are not supported by LUSAS Modeller and hence must be defined directly in a LUSAS Solver data file.

The Nonlinear Logfile

During the course of a nonlinear analysis, various information is output to the screen or logfile, so that you may assess the performance of the solution. For more information refer to the *Solver Reference Manual.*

Creep/Viscoelastic Analysis

Nonlinear viscous behaviour occurs when the relationship between stress and strain is time dependent. The viscous response is usually a function of the material properties, stress, strain and temperature history. Unlike time independent plasticity where a limited set of yield criteria may be applied to many materials, the viscous response differs greatly for different materials.

A creep/viscoelastic analysis may be carried out using a linear or nonlinear material model within a nonlinear, transient dynamic or thermo-mechanically coupled analysis. When carried out in a nonlinear analysis inertia effects are neglected and the time component is introduced using viscous control. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object created provides the means to modify the control settings.

When using viscous control, automatic time step calculations are only available when creep is included in the analysis.

- **Creep material properties** are defined using the **Attributes> Material> Isotropic / Orthotropic** menu item.
- **Viscoelastic material properties** are defined using the **Attributes> Material> Isotropic / Orthotropic** menu item.

Consolidation Analysis

Consolidation is the process in which reduction in soil volume takes place by expulsion of water under long term static loads. It is a typical diffusion process in which the field changes rapidly at the start of the process before settling down to a steady state condition a considerable time after the initial load is applied. As a result, time step parameters must be defined, and these are based upon currently set **timescale units**.

A consolidation analysis is controlled using nonlinear and transient loadcase control properties which are defined as properties of a loadcase. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object created provides the means to modify the control settings. Nonlinear analysis options can be specified separately for each analysis present.

Specifying an automatic time stepping procedure enables the time step to be modified so that the small time steps required at the start of the problem can be increased as the analysis progresses. In some cases the overall response time can be orders of magnitude larger than the initial time step. The value of the initial time step is important since the early variations in pore pressure must be accurately accounted for. Vermeer and Verruijt suggest the following criteria for determining the initial time step:

$$\Delta t \geq \frac{\gamma_\omega}{6Ek}(\Delta h)^2$$

where 'delta h' is the minimum distance between mesh nodes, which can be determined by selecting the two nodes and picking the Utilities>Mesh>Distance between Nodes menu entry.

Additionally, for a consolidation analysis, both the elastic and two-phase soil properties need to be defined.

See the *Application Manual (Bridge, Civil & Structural) 2D Consolidation under a Strip Footing* worked example for more details.

Geostatic control

Most geotechnical problems begin from a geostatic state. This represents the initial stress distribution where the undisturbed soil or rock body remains in equilibrium with the prescribed boundary conditions and geostatic loads, including self-weight, or overburden, producing zero deformations. This stress state is then used as the initial stress field in a subsequent static or coupled pore fluid diffusion/stress analysis.

Geostatic control can be set as part of the Nonlinear and Transient control properties of a loadcase, or by accessing the Nonlinear and Transient analysis control object in the Analyses Treeview.

Eigenvalue Analysis

Eigenvalue analysis controls are defined as properties of a loadcase. To add an eigenvalue analysis control to a loadcase, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. Eigenvalue analysis options can be specified separately for each analysis present. In a linear analysis, eigenvalue control should be specified on loadcase 1.

Applications

- **Eigenvalue** (also known as a frequency analysis) extracts the natural modes of vibration of a structure.
- **Eigenvalue (complex)** is used when joint or material damping has been specified for a model. For this, resultant displacement results of the real and imaginary components are displayed within Modeller.
- **Buckling load analysis** A linear analysis which may be applied to relatively ‘stiff’ structures to estimate the maximum load that can be supported prior to structural instability or collapse.
- **Stiffness analysis** Used to perform an eigenvalue analysis of the stiffness matrix at a selected stage of an analysis. This facility may be used in conjunction with a **nonlinear analysis** to predict structural instability or bifurcation points during a geometrically nonlinear analysis.
- By including **Modal Damping**, the overall damping factors for each mode can also be printed as a table in the LUSAS Solver output file. These values may then be used in a modal dynamic or spectral (harmonic/IMD) analysis if desired.

Good practice when carrying out eigenvalue analysis

It is good practice to perform a linear static analysis prior to carrying out an eigenvalue analysis. This eliminates the added complexities of the dynamic variables and will enable a check on the basic stiffness matrix for the structure. Any warning or error messages in the LUSAS output file (such as zero, negative or small pivots) should be investigated. In the event of problems occurring after completing the linear static and the eigenvalue analysis refer to **When An Eigenvalue Analysis Goes Wrong**

Solving an Eigenvalue Problem

Solving an Eigenvalue problem requires setting the Eigenvalue control properties for a particular loadcase.

In LUSAS, the following methods for eigenvalue extraction are available (described below):

- ❑ **Subspace Iteration** (Jacobi and QL solvers) The objective of the subspace iteration algorithm is to solve for a specified number of the lowest or highest eigenvalues and corresponding eigenvectors.
- ❑ A **Guyan Reduction** eigenvalue analysis may also be performed in conjunction with the subspace iteration method.
- ❑ **Inverse Iteration** (only available if a range of eigenvalues is specified) Computes the eigenvalues and corresponding eigenvectors within the range of interest.
- ❑ **Lanczos** Derived from the same principles as the subspace iteration method, but significantly faster, although convergence is not guaranteed. As well as calculating an eigenvalue range, as with the inverse iteration method, it is also able to calculate the minimal and maximal eigenvalues.

Having calculated all the required eigenpairs, the solution is completed by calculating error estimates on the precision with which the eigenvalues and eigenvectors have been evaluated, and normalising the eigenvectors according to a user-specified criterion.

For further details regarding the operation of the eigenvalue extraction facility, refer to the *Theory Manual*.

Subspace Iteration

The first step in the subspace iteration procedure is to establish the number of starting iteration vectors. This should be greater than the number of required eigenvalues to increase the rate of convergence. It is important to remember that the number of starting iteration vectors cannot exceed the number of degrees of freedom of the system. The default is the lowest of:

- 2 * number of Eigenvalues
- 8 + number of Eigenvalues
- The number of structural degrees of freedom

Note that entering zero (0) will force LUSAS to use the default value.

Occasionally insufficient eigensolutions are computed in the initial eigenvalue analysis. The number of eigensolutions can be increased by specifying an eigenvalue range.

Convergence Of Subspace Iteration

As the procedure iterates it is necessary to refer the numerical solution to a criterion with which to measure its convergence. It is assumed that the eigensolution has converged on iteration k when:

$$\frac{\lambda_i^k - \lambda_i^{k-1}}{\lambda_i^k} \leq \mathrm{rtol}$$

for all eigenvalues λ_i.

Using Eigenvalue Shifts

An important procedure that may be used in eigenvalue extraction is shifting. If rigid body modes are present in the system, the stiffness matrix will be **singular**, hence causing numerical problems in the subspace iteration and the Guyan reduction algorithms. To overcome this a shift may be applied to form a modified stiffness matrix, of which the associated eigenvalues will all be positive. To obtain the actual eigenvalues the shift is automatically subtracted from the calculated eigenvalues. The eigenvectors for both systems are the same.

The frequency shift enables the eigenvalues of unrestrained structures to be computed by removing the zero diagonal terms from the stiffness matrix. The convergence rate of the iterative eigenvalue solution procedure will increase with a smaller shift provided the shift is large enough to avoid numerical problems.

A frequency shift value can be estimated between 10% to 50% of the lowest expected fundamental eigenvalue.

Guyan-Reduced Eigenvalue Extraction

Good finite element approximations to low frequency natural vibrations may often be obtained by considering only those freedoms whose contribution is of most significance to the oscillatory structural behaviour. This characteristic may be utilised in the condensation of the full discrete model to a reduced system, in which the remaining equations adequately encompass the required vibration modes. Such a procedure is often termed **Guyan reduction**, and may be used to significantly reduce the overall problem size.

In a Guyan-reduced eigenvalue extraction, the stiffness contribution of those freedoms whose inertia effect is considered insignificant (designated the slave freedoms), are condensed from the system. The reduced equation system is therefore dependent on those freedoms remaining (designated the master freedoms). The resulting eigenvectors of the reduced problem are linear approximations to the true eigenvectors.

Guyan-reduced eigenvalue extraction is specified from the advanced dialog of the eigenvalue control properties.

Selecting The Master Freedoms for Guyan-Reduction

Master freedoms may be specified in one of three ways:

- **Manually** Using the attribute **Retained Freedoms**.
- **Automatically** Alternatively, a specified number of master freedoms may be automatically generated by setting the Eigenvalue properties (Advanced button). The generated master freedoms will be automatically selected such that the highest stiffness to mass ratios of the associated structural freedoms are used.
- **Mixed manual and automatic** Where manual and automatic master selection is combined, the specified number of automatic masters will be automatically selected from the available free equations.

The effective selection of the master freedoms is central to the accuracy of the simulated structural response. In the selection of the master freedoms, the following points should be considered:

- The master freedoms must accurately represent all the significant modes of vibration.
- Master freedoms should exhibit high mass to stiffness ratios. Hence rotational freedoms are usually inappropriate masters.
- Master freedoms should, where appropriate, be as evenly spaced throughout the structure as is appropriate.
- The ratio of master to slave freedoms should generally be within the range 1:2 to 1:10.
- Poor selection of the master freedoms will have a detrimental effect on the accuracy of the Guyan reduction solution especially at higher frequencies.

Sturm Sequence Check

When extracting eigenvalues it is important to verify that the computed eigenvalues constitute a continuous set, and that intermediate eigenvalues are not missed. To do this the Sturm sequence check is invoked; this may be switched off by setting the appropriate parameter on the eigenvalue control properties. All eigensolutions present are searched for, unless you request a smaller number of solutions by specifying the number of eigenvalues (note that in this case, the eigenvalues returned will not necessarily be the lowest in the range, unless the Fast Lanczos solver is used). A number of shift points are set up from which the eigensolutions are computed. These are determined by the maximum number of eigensolutions (system parameter MEIGSH) that can be located from each shift point. Shift points may be altered automatically in order to improve the rate of convergence. Eigensolutions are computed from each shift point in turn until all eigensolutions have been located.

Modal damping

By including Modal Damping, the overall damping factors at the eigenmodes can also be output as a table in the LUSAS Solver output file. These values may then be used in an

Interactive Modal Dynamic analysis if desired. Modal damping is only applicable to a Frequency analysis.

Inverse Iteration With Shifts

Eigenvalue extraction by inverse iteration may be utilised when calculating an eigenvalue range or frequency range.

This method uses a series of **shift** points from which to extract the eigensolutions using the inverse iteration method. The convergence to each eigensolution is governed by the closeness of the eigenvalue to the shift point and the method is thus efficient for locating the eigensolutions within narrow bands.

Convergence Of Inverse Iteration

As the procedure iterates it is necessary to refer the numerical solution to a criterion with which to measure its convergence. For inverse iteration it is important that the eigenvectors as well as the eigenvalues are computed to some degree of accuracy. The convergence criteria for the inverse iteration scheme is therefore based upon the mass orthogonality tolerance:

$$_{i\neq j}\Phi_i^T M \Phi_j < E_{i\neq j}$$

for all eigenvectors Φi and global mass matrix M.

Lanczos

When convergence is achieved, the Lanczos eigenvalue solver is usually faster than the subspace or inverse iteration solvers, and can use significantly less physical memory and hard disk than subspace methods. For these reasons it is ideal for large numbers of requested eigenvalues, and for large problems, although convergence cannot be guaranteed.

Fast Lanczos

The Fast Lanczos solver is both faster and much more robust than the original Lanczos solver, and is the recommended solver of choice for all eigenvalue analyses. When a software licence includes a fast Solver option this is used by default when the Default solver option is selected.

Centripetal Stiffening Effects

In rotating machinery, load correction terms that arise from the effects of rotation may significantly influence the natural frequencies of vibration. Within LUSAS the load correction terms due to centripetal acceleration can be considered.

The load correction terms, due to Coriolis forces and angular acceleration, are currently ignored because they result in non-symmetric damping and stiffness matrices respectively.

An eigenvalue analysis in LUSAS will include the gyroscopic effects in the stiffness matrix (for certain elements; see *Element Reference Manual*) if you use a CBF load to simulate the angular velocities of the shafts (note that this requires a nonlinear analysis). Such a natural frequency analysis would give the frequencies of the **lateral** modes of vibration. The physical effect modelled by the centripetal stiffening facility for eigenvalue analyses is the stiffening that a rotating be as a result of radial expansion and the corresponding increase in hoop stresses. These stresses effectively stiffen the structure and can significantly increase the eigenvalues. See the *Theory Manual* for further information.

Notes

- The relationship between the eigenvalue, λ, and the angular frequency, ω, is:

$$\lambda = \omega^2 \qquad \lambda = (2\pi f)^2 \qquad f = \frac{\sqrt{\lambda}}{2\pi}$$

- An eigenvalue analysis of the stiffness matrix has no physical meaning except that a zero magnitude implies a critical point of some description.
- Ensure that mass normalisation is chosen for the eigenvalue analysis if it is to be followed by a spectral or IMD analysis.
- It is possible to use constraint equations in both an eigenvalue and a harmonic response analysis in LUSAS. However, the Sturm sequence check may prove unreliable, unless the fast Lanczos solver is used.
- Non-zero rigid body eigenvalues may be experienced when using QSI4 elements. This is due to the method used to obtain the lumped mass matrix for this element (a consistent mass matrix not being available). QSL8 and QTS4 elements give correct eigenvalues for both lumped and consistent mass matrices, forming the mass matrix using a shape function array. QSI4, however, forms the rotation terms explicitly without the use of these functions. Small inaccuracies in the lumping of the mass to the rotational degrees of freedom may thus be possible for certain mesh definitions. If these eigenvalues are significant, the analysis should be continued using another shell element type, such as QSL8 or QTS4 elemnts.
- The error norm for a given mode provides a relative measure of the accuracy of the computed modes. A high error norm will provoke a warning message, and signifies inaccuracy in either the eigenvalue or the eigenvector, or both. Warnings are not issued for computed modes which are close to zero, since they may approximate rigid body modes which are exactly zero, and are thus prone to incurring a large relative error.
- For eigenvectors which are normalised to unity, the largest translational component will be set to one. Thus analyses containing rotational degrees of freedom, for example, may have eigenvectors normalised to unity that contain rotational components greater than one in magnitude.
- If it is required to use the **1/1-buckling load** option to try and obtain all positive eigenvalues the specified load must be close to the collapse load in order to obtain an

accurate load factor. It should be noted that this procedure is not without its problems. Depending on the structure and the load level considered the eigenvalues can be very closely spaced, causing convergence problems in the iterative solution.

- When specifying the range within which Eigen solutions will be located Sturm sequence checks are carried out on the range limits in order to determine the number of eigen solutions that exist within the range. All solutions are then searched for.

Eigenvalue Buckling Analysis

A linear buckling analysis is a useful technique that can be applied to relatively stiff structures to estimate the maximum load that can be supported prior to structural instability or collapse. The assumptions used in linear buckling analysis are that the stiffness matrix does not change prior to buckling, and that the stress stiffness matrix is simply a multiple of its initial value. Accordingly, the technique can only be used to predict the load level at which a structure becomes unstable if the pre-buckling displacements and their effects are negligible. As this procedure involves assembly of the stress stiffness matrix, only elements with a geometric nonlinear capability can be used in a linear buckling analysis.

The main objective of an eigenvalue buckling analysis is to obtain the critical buckling load, which is achieved by solving the associated eigenvalue problem.

For buckling analyses involving constraint equations, the Fast Lanczos solver will only find eigenvalues either side of zero, i.e. in the range $(-\infty, 0)$ or $(0, \infty)$. If a range of eigenvalues is required in an interval which contains zero, two separate analyses must be carried out, where the interval is divided into two sub-intervals either side of zero.

Alternative Eigenvalue Buckling

Occasionally the initial stress stiffness matrix may not be positive-definite, causing the eigensolution method to fail. To overcome this problem the original buckling problem may be recast into a form where all eigenvalues are positive except when the buckling load factor is less than unity. When using this technique the load level must be adjusted to ensure that all the load factors are greater than unity. In other words, the load applied should be below the lowest expected buckling mode of the structure. An accurate load factor will however only be obtained if the specified load is close to the collapse load.

It should be noted that this procedure is not without its problems. Depending on the structure and the load level considered the eigenvalues can be very closely spaced, causing convergence problems in the iterative solution.

Output From Buckling Analyses

For a linear eigenvalue buckling analysis, the buckling load is obtained from the print results wizard. This buckling load is directly related to the eigenvalues extracted and will be in the following format

MODE	EIGENVALUE	LOAD FACTOR	ERROR NORM
1	33.0456	33.0456	0.190087E-10
2	64.3432	64.3432	0.179595E-07
3	130.903	130.903	0.202128E-11

The buckling load for a mode is obtained by multiplying the actual magnitude of the applied loading by the load factor (33.0456 in the case of the 1st mode).

Absolute displacement output is not available from any eigenvalue analysis. It is available, however in a normalised state. For buckling analyses the eigenvectors (mode shapes) are normalised to unity, where the maximum translational degree of freedom is set to one (mass normalisation is not applicable to buckling analyses).The mode shapes are, therefore, accurate representations of the buckling deformation but do not quantitatively define the displacements of the structure at the buckling load.

Reactions, stresses and strains represent the distribution at the buckling load, again their magnitude is not quantitative.

Spectral Response Analysis

To study the effects of ground motion excitation on structures it is necessary to input the intensity of the motion. One practical measure can be obtained from a knowledge of the response spectra. Spectral response analysis seeks to determine the response of a structure subjected to a specified support excitation using modal superposition. This can be achieved without recourse to direct integration of the model over the complete duration of an event.

A spectral response analysis is available using the **IMD** loadcase.

Starting Procedure

Before specifying the spectral response data the eigenvalues and eigenvectors of the system are computed using an eigenvalue extraction analysis (note that the computed eigenvectors must have been normalised to the global mass).

Spectral Response Data Input

The spectral curve, spectral curve type and percentage damping are specified in the spectral curve which is part of the IMD loadcase definition.

The spectral curve may be defined as:

- **Frequency or period vs displacement**
- **Frequency or period vs velocity**
- **Frequency or period vs acceleration**

To compute the participation factors it is necessary to specify the direction of excitation. The excitation may be specified simultaneously in three directions. The factor specified in each direction is used to scale the spectrum intensity in that direction.

For each mode the spectral displacement is determined from the frequency and this is multiplied by the participation factor and the excitation vector to determine the response.

Damping may be specified for each mode of the structure or at known frequencies of vibration. If damping is only specified for the first Eigen mode this value is applied to all modes. When the percentage damping specified on the spectral curve differs from that specified in the viscous damping a correction is made to the spectral displacement based on the formula chosen when defining the spectral combination. Damping may also be described in terms of the Rayleigh damping parameters and transferred from LUSAS Solver.

Normally the number of modes included should ensure the sum of the mass participation is not less than 90% in all significant excitation directions.

To obtain the design values some form of combination may be used. Within LUSAS the following methods of combination are available:

- **Square root of the sum of the squares (SRSS)**
- **Complete quadratic combination (CQC)**
- **Absolute Sum**

Note: When zero damping is specified the CQC gives exactly the same results as the SRSS technique.

Transient Dynamic Analysis

A dynamic analysis is controlled using nonlinear and transient loadcase control properties which are defined as properties of a loadcase. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object created provides the means to modify the control settings. Additional Nonlinear analysis control options are accessed via the Nonlinear analysis options control object in the Analyses Treeview. Nonlinear analysis options can be specified separately for each analysis present. Time-based values are based upon currently set **timescale units**.

Application

A transient dynamic analysis (sometimes referred to as **step-by-step**) will be required when loading may not reasonably be considered to be instantaneous, or where **inertia** or **damping** forces are to be considered.

Solution methods

Dynamic solution methods generally numerically integrate in the time domain. The solution is progressed through time in a step-by-step manner by assuming some variation of the displacements and velocities over small intervals of time. Within each time step the solution yields the displacements at the discrete time points representing the end of the current time step. For known initial conditions, successive application of this procedure furnishes the dynamic response of the structure.

The following numerical integration schemes are available:

- Central Difference
- Hilber-Hughes-Taylor

Implicit and Explicit Dynamics

Dynamic analysis may be performed using two methods:

- **Implicit Dynamics** Implicit methods require the inversion of the stiffness matrix at every time step, and are therefore relatively expensive, but unconditionally stable. By default the Hilber-Hughes-Taylor is used.
- **Explicit Dynamics** In contrast, explicit methods de-couple the equilibrium equations, hence removing the necessity for stiffness matrix inversion. Explicit methods are only stable for a range of time steps, determined by the problem being analysed, and the discretisation adopted. Explicit methods are automatically invoked by specifying explicit dynamic elements. In this instance the central difference scheme is mandatory and chosen by default. For explicit analysis lumped masses must be used.

Starting procedure

To start a dynamic analysis a knowledge of the initial conditions is required. The initial conditions for the Hilber-Hughes-Taylor integration scheme are:

$$V_1 = V_0 + \left[(1-\gamma)A_0 + \gamma A_1\right]\Delta t$$

where:

$V_{0,1}$ are the velocities at time steps 0,1

$A_{0,1}$ are the accelerations at time steps 0,1

D_t is the time step

γ is the Hilber-Hughes-Taylor integration constant gamma

The initial velocity, V_0, and initial acceleration, A_0, can be defined in an implicit dynamics analysis.

The starting conditions in explicit dynamics must be consistent with the central difference integration scheme:

$$V_{1/2} = V_{-1/2} + A_0 \Delta t$$

where:

$V_{-1/2,1/2}$ are the velocities at times -Δt/2, Δt/2

A_0 is the acceleration at time 0

Only the initial velocity, V, (actually relating to time -Dt/2) can be defined in an explicit dynamics analysis. The displacements, d, (relating to time zero) and accelerations, A, (relating to time -Dt) are assumed to be zero. Because of the nature of the central difference integration scheme, an initial velocity will generate accelerations at time zero. Accelerations relating to time zero are used to compute displacements at time Dt and will in fact be output at time Δt.

In general, the values output at any time t will be:

$$d_{t}, V_{t-1/2}, A_{t-1}$$

This means that in the output for any time step, the displacements will relate to the current response time while the accelerations effectively lag one time step behind the displacements.

Impact Dynamics

In addition to using nonlinear joint models to represent contact and impact, a specialised procedure is available for modelling impact in dynamic analysis. This procedure uses a slideline technique, and permits the surfaces of 2D, axisymmetric, and 3D structures to register and react to contact with one another. See **Slidelines** for more information.

Thermal / Field Analysis

A thermal/field analysis can be defined initially when creating a new model or by adding a general thermal analysis to an existing structural base analysis.

Transient thermal/field analysis controls are defined as nonlinear and transient properties of a loadcase. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object created provides the means to modify the control settings. The default state for additional thermal analyses added to an existing one is to inherit the assignments from the first, but the additional thermal analyses can 'override' some or all of those assignments.

Thermal / Field Analysis Types

Two types of thermal/field analysis may be performed:

- **Steady State** thermal/field analysis
- **Transient** themal/field analysis

Facilities for **thermo-mechanically coupled** analysis are also available.

Applications

The quasi-harmonic equation defines the behaviour of a variety of field problems.

Some of the more common quasi-harmonic applications, and the associated field variables, are listed in the table shown right.

Application	Field Variable
Thermal conduction	Temperature
Seepage flow	Hydraulic head
Incompressible flow	Stream function
Soap film	Deflection
Elastic torsion	Warping function
Electric conduction	Stress function
Electrostatics	Electric potential
Magnetostatics	Magnetic potential

Method

The modelling and solving of a thermal/field problem follows an identical process to that of modelling and solving a general structural problem. The domain is discretised using a series of field elements, **thermal material properties** are specified, **thermal loads** are applied, and the equations solved for the values of the field variable at each nodal point. Thermal link elements or the specification of **thermal surfaces** determine how heat is conducted, convected or radiated across gaps and spaces between different domains.

Steady State Thermal / Field Analysis

In a manner similar to static structural analysis, steady state thermal/field analysis assumes that the loaded body instantaneously develops an internal field variable distribution so as to equilibrate the applied loads. Note that the use of temperature dependent material properties or loads renders the problem nonlinear.

Transient Thermal Analysis

A transient thermal/field analysis should be performed when time effects are significant in a thermal/field problem. In a similar manner to structural dynamics, transient field analysis involves the evolution of a new field variable distribution from a set of initial conditions via a set of transition states evolving through time. The initial conditions of the body must firstly be prescribed. This may be done by performing the appropriate linear or nonlinear steady state analysis.

Defining a transient thermal analysis

A transient thermal/field analysis can be defined initially when creating a new model or by adding a general thermal analysis to an existing structural base analysis.

Transient thermal/field analysis controls are defined as nonlinear and transient properties of a loadcase. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object created provides the means to modify the control settings. Thermal/field analysis options can be specified separately for each analysis present.

Linear transient analysis

Integration Scheme	beta
Crank-Nicholson	1/2
Euler	0
Galerkin	2/3 (default)
Backward difference	1

The transient problem is integrated through time using a 2-point integration scheme. The type of integration scheme maybe changed on Advanced Time step parameters dialog accessed from the nonlinear and transient control dialog by specifying the parameter `beta`. Some of the more common 2-point integration schemes and their associated `beta` values are shown in the table above.

Notes

- In the limit the final solution should be the same as the steady state analysis subject to the new loading and boundary conditions; the transient analysis merely models the thermal inertia in moving from the initial to the final conditions. The body property which is used to describe this inertia is the effective heat capacity.
- When choosing an increment of time, the stability of the incrementation scheme must be examined. When `beta` is greater than or equal to 0.5 the solution is unconditionally stable (the Crank-Nicholson, Galerkin and Backward difference schemes are of this form).
- When `beta` is between the limits 0 and 0.5 the solution is stable provided that:

$$\Delta t < \frac{2}{(1-2\beta)\lambda_{max}}$$

  where β is the input parameter `beta` and λmax is the maximum eigenvalue of the system.

- The time step used for implicit algorithms is dependent upon the number of modes that influence the response of the system. Generally, the major part of the response is governed by the lower modes so that:

$$\Delta t < \frac{1}{3\lambda}$$

where λ is the minimum eigenvalue of the system.

- The Galerkin scheme is recommended since it generally provides good accuracy and is the least susceptible to oscillations.

Nonlinear Transient Analysis

For nonlinear transient analysis the backwards difference algorithm must be used (`beta` = 1.0). The backward difference algorithm is unconditionally stable, and the time step length considerations are the same as for linear analyses.

For analyses including a phase change, there is either an absorption or release of energy in order to create or break the molecular bonds. This is modelled by varying the effective heat capacity in the transient analysis. To do this the material property of enthalpy is introduced.

Enthalpy, H, is defined as:

$$\frac{dH}{dt} = \frac{dH}{d\varphi}\frac{d\varphi}{dt} = C\frac{d\varphi}{dt}$$

where C is the effective heat capacity including the effects of the latent heat of evolution due to phase changes and φ is the temperature. In the material data input both H and C may be specified. For analyses where phase changes are not represented, the effective heat capacity value C is used in the calculations.

For analyses where phase changes are represented, tabular input should be used to define the variation of H with temperature, together with an initial value of C. Providing a variation in temperature exists at a point, the effective specific heat is then interpolated from the enthalpy values. If no variation exists, for example, in an area of the problem that has experienced no change in temperature from the initial temperature, then the initial value of C is used.

For nonlinear analysis the nonlinear control parameters are used to define the iterative strategy. The convergence section is utilised to provide tolerances for defining steady state, and either a field norm (temperature equivalent to the displacement norm), or a residual flow norm (equivalent to the residual force norm) may be used.

Coupled Analysis

A coupled analysis can be defined initially when creating a new model or by adding a general thermal analysis to an existing structural base analysis, or a general structural analysis to an existing general thermal analysis structural base analysis. Coupled analysis options can be accessed via the Coupled analysis options object in the Analyses Treeview.

Coupled analysis

The flow of heat through a body and the corresponding distribution of temperature is described by the quasi-harmonic equation; the body geometry is assumed to remain constant. The displacements of the same body, subjected to various forces, is described by equations of static or dynamic equilibrium; the temperature distribution is assumed not to vary with displacement. To include the effect of the change in geometry in the thermal analysis, and the change of temperature in the static analysis, requires that this information is separately calculated by the appropriate analysis and then transferred. This process is known as **thermo-mechanical coupling**.

Thermo-mechanical coupling may be sub-divided into two classes depending on the nature of the problem.

- **Semi-*c*oupled** analysis
- **Fully coupled** analysis

Semi-coupled analysis

In a semi coupled analysis, for instance, the structural response is influenced by the temperature field, but the thermal response is independent of the structural response, or vice-versa. In such a case, the thermal analysis is performed prior to the structural analysis, and either a single or series of nodal temperature tables are created. These are read during the structural analysis at the required loadcase or time step.

Fully coupled analysis

In a fully coupled analysis, the thermal and structural analyses must be performed simultaneously with a continuous transfer of information between the two analyses. For instance, in addition to modelling the influence of the thermal field on the structural response, the effect of the structural response on the thermal field is represented. Temperatures are transferred from the thermal to the structural analysis, and the updated geometry is transferred from the structural to the thermal analysis. The analyses may be coupled on the incremental or iterative levels (iterative coupling is machine dependent).

For true full coupling of two nonlinear fields, information transfer has to occur on an iteration level within each increment, so that in addition to preserving equilibrium of the local thermal and structural fields, equilibrium of the combined system is maintained. Iterative coupling is essential for strongly coupled systems, e.g. structure to structure contact. For weaker thermo-mechanical coupling, information transfer at an increment level should provide an adequate solution.

Heat dissipated due to plastic work

The heat flux produced due to plastic work can be considered in a coupled analysis. In this type of problem the structural analysis is started first and the heat dissipated through elasto-plastic deformation is transferred to a thermal analysis. The nodal temperatures may then be returned to the structural analysis where they can be used to produce thermal strains and compute temperature dependent properties. The following points should be considered when using this facility:

- The heat flux generated due to plastic work is a function of the time increment over which the work is done. For a meaningful solution to this type of problem a dynamic structural and/or a transient thermal analysis should be undertaken.

- In general, it is recommended that reading and writing to the data transfer file is carried out at the same point in the analysis. This avoids any inconsistency occurring between the time of generation of plastic work and the time of diffusion in the thermal analysis.
- The thermal softening facility is only valid for nonlinear material models which allow input of a heat fraction. The heat fraction takes a value between 0 and 1 and represents the fraction of plastic work converted into heat.

Initialisation of Structural Temperatures

LUSAS structural elements allow you to input both an initial temperature field and a current temperature field. The structure is not strained if its current temperature field is the same as the initial temperature field; variations in temperature, defined by the current temperature field, from this initial temperature field, cause thermal straining. The nodal temperatures transferred from the thermal to the structural analyses are read directly into the current temperature field and the thermal strains are then calculated from the difference between the current and initial fields. The initial field in this case is zero everywhere unless it is directly input using the structural temperature loads. It is possible to initialise the initial temperature field to the current temperature field which is read from the data transfer file. Further data transfers will be read into the current temperature field only.

Initialisation of Reading And Writing Commands

To maintain consistency between reading and writing on a specific increment both data reads and writes are performed at the end of the current increment, i.e. if data is required for use in the 100th step then it must be read in the 99th step. Similarly, if data is required to initialise the structural temperature or geometry field, it must be read on step zero.

Data Transfer Between Joints And Links

The physical nature of the joint and link elements is essentially different. Heat flow can occur between two unconnected bodies via convection and radiation across the intervening medium. On the other hand, joint elements introduce stiffness against displacement, implying a physical connection between two bodies. Whilst both may be true simultaneously, more usually, only one condition will apply. In these circumstances it is necessary to introduce dummy joints with springs of zero stiffness or links with zero conductivity to ensure that the appropriate element data is correctly transferred.

Fourier Analysis

Fourier analysis controls are defined as properties of a loadcase. To do so, in the Analyses Treeview access the loadcase context menu and choose the **Controls** menu item. Fourier analysis options can be specified separately for each analysis present.

Fourier elements

Fourier elements offer an efficient method to solve problems in which axisymmetric structures are subjected to non axisymmetric loading, provided that the displacements are small and linear theory applies. The circumferential displacements and variations of load are expressed as the sum of the components of a Fourier series, whilst the axial and radial variations are described by the standard finite element formulation. Each term of the Fourier series is analysed individually and the results are then combined to provide the overall solution.

Fourier elements can be used to model both solid and thin walled structures; in particular they offer an ideal method to obtain an initial estimate of the eigenvalues of thin walled structures without the expense of performing a full shell analysis on the complete structure. The choice between a full structural discretisation using solid or shell elements and the use of the Fourier element depends upon the number of Fourier terms that are required to accurately describe the load; if only a few terms are required then the Fourier element should be considered.

Attributes and settings

A Fourier analysis can be considered as a generalisation of the standard axisymmetric analysis. The finite element mesh is defined in the XY-plane and may be axisymmetric about either the X or the Y axis. **Loading** is applied to the mesh in the standard manner using the loadcase properties, with its circumferential variation defined using the **curve definition**. Finally the Fourier components to be computed are input using the **Fourier control** as part of the loadcase properties.

Supports are defined in the usual manner, with the declaration free, restrained or spring supports. For the n=0 harmonic the spring stiffness per unit radian must include a factor of 2p for the implicit integration around the surface. For harmonics other than n=0 the factor should be p. Certain restrictions are applied to the freedoms of nodes lying on the axis of symmetry. These conditions, shown in the accompanying table, are automatically imposed on the centre line nodes.

Axisymmetric about X axis	Axisymmetric about Y axis
n=0 v, w=0	n=0 u, w=0
n=1 u=0	n=1 v=0
n>1 u, v, w=0	n>1 u, v, w=0

Dynamic, Eigenvalue And Harmonic Response Analyses

A Fourier analysis processes each harmonic individually as they possess their own unique stiffness, mass and damping matrices and load vector. By selecting just one harmonic a dynamic, eigenvalue or harmonic response analysis can be executed for that particular harmonic. The complete structural response can be obtained by superimposing the different results from the selected harmonics.

The Fourier control should specify just one harmonic of a series. The automatic calculation of the load coefficients from a given load input is suspended and you must input the appropriate load coefficient; if this is not known it may be obtained from a static analysis. Note that to represent a global load, the applied load will have components in both the tangential and the radial directions (see the *Theory Manual* for details of the loading calculations). Only one loadcase may be processed.

Inertial Loading

The operation of the inertial loading, input using the load type **body force**, is slightly different to the other standard loads. Inertial loads are calculated from element volumes and applied accelerations. The specification of linear accelerations, angular velocities and angular accelerations is enough to define the forces acting on the structures since the element volume and density from which the mass is calculated are element properties. Depending on the input data, loads are applied for the n=0,1,2 harmonic components. However, the body force data must still be associated with a dummy load curve and must be declared in the first loadcase.

In addition to the input accelerations, angular velocities and angular accelerations you can input an offset origin about which the rotations are applied. The local rotation about the finite element axis of symmetry should not be confused with the global rotation about the global axes. The local rotation implies that the body is rotating with respect to the finite element axes, while the global rotation is a rigid body rotation of the complete finite element model. For further details see the *Theory Manual*.

Centripetal Load Stiffening

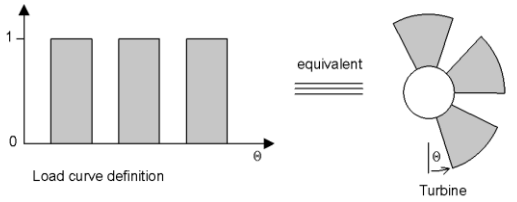
Centripetal load stiffening has been applied to the n=0 harmonic, but there is no nonlinear stress stiffening contribution.

Special Application to Non-Axisymmetric Structures

In some instances, the structure may not be truly axisymmetric but it may be desirable to obtain an approximate response from an axisymmetric analysis. An example of this is a turbine where the turbine axis is axisymmetric but the fan blades are not. The use of the standard Fourier material properties is inappropriate for the fan blades since the hoop stresses introduced by the element material model provide significant artificial stiffening. To alleviate this problem the use of the plane stress material model input using the **orthotropic materials properties** is permissible provided that the element is given adequate torsional restraint. The use of this material model can be thought of as smearing the individual stiffness of the fan blades into an equivalent axisymmetric structure.

For Fourier elements using orthotropic properties, body forces are applied using the associated load curve.

If the load curve is input as a series of 1s and 0s this is equivalent to selectively integrating the internal forces for each fan blade. The scheme is illustrated in the diagram.



Thermal Problems

If temperature dependent material properties are used the temperature field must be axisymmetric. For non-temperature dependent materials, a general temperature field can be input in the same manner as the other element loads. Temperature loads cannot be used in dynamic or harmonic response analyses.

Post-Processing

Fourier results may be expanded using the **Graph Wizard**.

Influence Analysis

An influence analysis identifies where, on a structure, the application of load will result in the most onerous load effects for a specific point of interest. This region where the most onerous loading will occur is commonly known as the adverse area. In bridge analysis, and considering carriageways in particular, this adverse area is typically required by design codes to be used to apply live loads such as vehicle and lane loading. Additionally, for some design codes, traffic load intensities are a function of the length of lane which is loaded and where this is this case an influence analysis is key to determining where, and to what intensity, loading should be applied.

Influence Analysis Methods

Influence analysis can be carried out using the following methods:

- **Reciprocal Theorem** - also known as the Muller-Breslau Theorem, or Maxwell's Theorem, is a means of calculating an influence by the use of an **auxillary structure**, to which a deformation corresponding to a load effect of interest is applied.
- **Direct Method** - is a more general and powerful way of calculating an influence where the effect of a specified point load is assessed at each node or grid location on a loadable area of a structure. The value of the load effect of interest at each specified location is then used to construct an influence line or surface for that location. The Direct Method can calculate influences for user-defined results components; for 3D beam models including torsional effects; for slice resultants from shells; calculate influence as a post-processing facility; and can calculate influence for many locations and quantities with a single analysis. Solving an influence analysis using the Direct Method will take the same time regardless of the number of influence locations solved.

Both methods require influence attributes to be defined and assigned to a model prior to an influence analysis being carried out. Note that **Vehicle Load Optimisation** analysis requires influence attributes to be assigned to a model in order to generate influence surfaces and calculate the most adverse vehicle load patterns on bridge structures.

Notes

- Only one Reciprocal Theorem analysis entry is added to the Analyses Treeview.

- Multiple Direct Method influence analysis entries can be added to the Analyses Treeview.

Which Influence Analysis Method to Use?

The main differences between the two influence attribute types are summarised in this table

Reciprocal Theorem	Direct Method
Provides only limited options to solve influences for grillage, slab and deck models	**Provides more general and powerful options to solve influences for line beam, frame, grillage, slab and deck models**
List of available influence types is limited to Shear, Moment, Reaction and Displacement	**List of available results entity / components for influence location is only limited by the elements used within the model**
Unable to calculate influence for user-defined results components	**Can calculate influence for user-defined results components**
Unable to calculate influence surface for 3D beam models including torsional effects	**Can calculate influence surface for 3D beam models including torsional effects**
Unable to calculate influence for derived results such as My-Fx*Rz	**Can calculate influence for derived results such as My-Fx*Rz**
Unable to calculate influence for slice resultants from shells etc	**Can calculate influence for slice resultants from shells etc**
Unable to calculate influence as a post-processing facility	**Can calculate influence as a post-processing facility**
Unable to calculate influence for many locations and quantities with a single analysis. One solution (solve) is required for each influence location	**Can calculate influence for many locations and quantities with a single analysis (solve)**

In summary, the reciprocal theorem method provides less functionality than the Direct Method, but is useful when a load direction is not known as, for example, to identify where to apply wind to a tower in order to produce a certain effect within the internal structure members. It is also retained to support legacy models. The Direct Method is useful for when a load direction is known as, say, downwards in the case of bridge loading. This method also provides more general functionality.

Influence Analysis Attributes

- ❑ **Reciprocal Theorem** influence attributes are defined from the **Attributes > Influence> Reciprocal Theorem** menu item. The influence type may be a Shear force, a Reaction, a Moment or a Displacement only. For each influence type the influence direction and either a positive or negative displacement direction needs to be specified. For more information see **Reciprocal Theorem Influence Attributes**
- ❑ **Direct Method** influence attributes are defined from the **Attributes > Influence> Direct Method** menu item. The influence type may be set by defining any Entity of interest (such as a Reaction, or a Force/Moment, or a Stress), an influence direction (such as an axis of a member, or a path along a structure, or a material

direction) and a Component of interest (such as My). For more information see **Direct Method Influence Attributes**

Solving an Influence Analysis

The presence of assigned influence attributes on a model determines that an influence analysis will automatically be carried out when a model is solved.

To solve an influence analysis choose the **Solve Now** menu item on its context menu or press the Solve button on the main toolbar menu. Influences for the Reciprocal Method can also be solved selectively by choosing the **Influences to Solve** context menu.

A successful solve will add influence loadcase results to the Analyses Treeview.

Writing a Datafile with Influence Attributes

Once all influence attributes have been assigned to a model, they can be exported to a data file using the menu option **Files> Export Solver datafile...**. LUSAS will automatically identify the datafile to be one that will require an **Influence analysis** as opposed to a general analysis and as such data file names will be generated from the specified file name and the influence number. For example, if the specified file name is **bridge**, then files **bridge1.dat** and **bridge2.dat** will be created for influence lines 1 and 2 respectively.

Viewing results of an Influence Analysis

After solving a model with assigned influence attributes the influence shape for each influence point of interest can be viewed on the **Influence shape** layer in the Layers Treeview. This layer name is a renaming of Deformed mesh layer and cannot be added manually.

The influence shape for each influence can be seen by setting each influence loadcase active, in turn, in the Analyses Treeview.

By adding a Contours layer to the Layers Treeview contours of **Influence result** can be displayed. By referring to the contour key regions of the model where positive or negative loading effects take place can be seen.

Notes

- Influence shapes created from a Reciprocal Method influence analysis cannot be used directly in a Vehicle Load Optimisation analysis. The Generate Influence Surfaces check box on the Vehicle Load Optimisation dialog should be checked to calculate the influence surfaces required.
- When a Direct Method influence analysis has been solved for one assigned attribute the effect of a unit load on any part of the structure can be seen immediately for any subsequently assigned influence attributes. Additional solving is not necessary.

Accuracy of influence solutions

The accuracy of influence solutions, as with all finite element analyses, improves with increased mesh refinement and, in the case of the Direct Method, with closer spacing of loading points in the loading grid.

When using the Reciprocal Method, the influence ordinate at any location is generally obtained to the accuracy of the element shape function and mesh refinement. A specific issue arises where a discontinuity occurs in an influence surface - such as where the influence of shear "jumps" at the central support in a two-span structure. At such a discontinuity, the peak value given will be that associated with the node adjacent to the discontinuity. Therefore in order to minimise inaccuracy, small element sizes should be used adjacent to the discontinuity meaning, for this example, a short element adjacent to the support.

When using the Direct Method, the influence ordinate is affected not only by the element shape function and mesh refinement but also by the density of the grid of points in the loading grid (if such a grid is used). With a coarse mesh or coarse loading grid, faceting could lead to significant inaccuracy, especially near cusps or discontinuities. In all such cases, increased accuracy is gained with closer spacing of grid points. Taking the example of shear force adjacent to an internal support, a loading grid which "follows" the FE mesh may give a less accurate answer than use of an independent loading grid of similar spacing since it the key results are obtained by the loads each side of the support rather than directly applied to the supported node.

Direct Method Influence Analysis

A Direct Method Influence Analysis requires Direct Method influence attributes to be defined and assigned to a line beam, frame, grillage, or slab and deck model. The locations at which a prescribed load will be applied to nodes or points on the model also need to be specified. Influence attributes are defined and assigned from the **Attributes> Influence** menu. Load locations are defined on the Direct Method Influence dialog, which is displayed either when a Direct Method Influence attribute is first assigned to a node or point on the model, or when the **Analyses> Direct Method Influence Analysis** menu item is selected or edited.

Previously defined settings can be accessed and edited from the context menu for each Direct Method Influence entry in the Analyses Treeview.

Load locations

- **Search area** - A search area can be used to limit the area over which the unit load is applied so that the effect of the load on certain features may be removed from the analysis. If a search area is not specified the default loading area is Whole model.
- **Nodes in search area** - restricts the application of the applied load to just those mesh nodes within the search area, or to the whole model, as appropriate.
- **Grid** - predominantly used with line beam models where the geometric beam section represents a loadable top slab. For this, the grid of points is used to represent the loadable region of the slab. The Grid method also provides the means to define a grid of points that can be used as an alternative to the nodes or points present in a shell or

plate model that are used in an influence analysis. For a model with a very fine mesh, loading a grid having fewer points will reduce the solution time. For line beam models the use of a reference path using lines representing the centreline of a loaded region of a structure is recommended if the grid loading method is used. For plate and shell models a reference path may represent the centreline of a carriageway on which a loading grid would be positioned.

- **Centreline** - the centreline of the grid can be defined with reference to a previously defined Path or the X, Y or Z global axes. For most types of modelling the use of a reference path is recommended. Specifying X, Y or Z causes the grid to be created along each of those axes at X=0, Y=0 or Z=0.
- **Transverse width** - the width of the grid to be loaded (must be greater than 0.1). Grid settings control the number of load locations within this defined width.
- **Grid settings** - These control the density of the grid of points that will be loaded. Longitudinal and transverse values can be specified. If one of the global axes has been selected as the centreline of the grid, then the overall extent of the model will be calculated and a suitable grid generated to encompass the whole model.

Loading settings

Loading settings provide the means to specify the magnitude and direction of the loading applied to a grid of points used for an influence analysis.

Vehicle Load Optimisation

Vehicle load optimisation is used to identify the most onerous vehicle loading patterns on bridges for a chosen design code and to apply these loading patterns to LUSAS models. It reduces the amount of time spent generating models and leads to more efficient and economic design, assessment or load rating of bridge structures.

Vehicle load optimisation analysis is available in Bridge software products only. It is accessed from the **Analyses > Vehicle Load Optimisation** menu item.

See **Vehicle Load Optimisation** in *Application Manual (Bridge, Civil & Structural)* for more details.

Cable Tuning Analysis

A cable tuning analysis calculates load factors for selected lines in a model that represent cables in order to achieve defined target values set for various components or features.

Cable tuning analysis is available in selected Bridge and Civil & Structural software products only by use of the **Analyses > Cable Tuning Analysis** menu item.

See **Cable Tuning Analysis** in the Application Manual (Bridge, Civil & Structural) for more details.

LUSAS Solver Types

Solver type and options can be specified via the Solver Options dialog which is accessed via the Model Properties or Solver option control objects in the Analyses Treeview. The current solver options supported are as follows:

- **Standard Frontal**- an element-by-element frontal solver which does not require assembly of the global stiffness matrix. It uses a direct, sparse solution technique based on Gaussian elimination. Stiffness and load arrays are read into memory and assembled into the structural stiffness matrix and load vector. This solver is applicable to all types of analysis present in LUSAS Solver. The frontwidth of the problem may be reduced by optimising the order in which the elements are presented. See **Frontal Optimisation** for details.
- **Fast Multi Frontal**- a global frontal solver which assembles global stiffness and load data. It uses a direct, sparse solution technique based on Gaussian elimination. Stiffness and load arrays are read into memory and assembled into the structural stiffness matrix and load vector. This solver is applicable to all analysis types except for superelements, Guyan reduction, and nonlinear problems involving branching and bracketing.
- **Iterative (Conjugate Gradient)** selects an iterative, sparse solution technique for solving static, linear analyses. The global stiffness matrix and load vector(s) are assembled, and is designed to run entirely in-memory. Three preconditioning techniques are available to assist the convergence rate of the conjugate gradient method:

    - **Standard -** The incomplete Cholesky preconditioning technique is the most robust (provided an appropriate drop tolerance is chosen), and is applicable to all analyses for which the conjugate gradient solver may be used.

    - **Decoupled -** The decoupled incomplete Cholesky preconditioning technique may be used for all analyses except those involving tied slidelines, thermal surfaces and Fourier elements. It generally leads to faster overall solution times than Incomplete Cholesky preconditioning, although more iterations are required for convergence. For less well conditioned problems, the conjugate gradient algorithm may not converge using this technique, so care should be taken.

    - **Hierarchical** - The hierarchical decoupled incomplete Cholesky preconditioning technique is only available for models consisting entirely of two- and three-dimensional, solid continuum, quadratic elements, and offers excellent convergence properties. It is by far the most effective technique for models of this type, and when used in conjunction with fine integration allows solutions to be obtained for relatively ill-conditioned problems. For very ill-conditioned problems of this type (e.g. where the average element **aspect ratio** is high), an extra preconditioning option exists which will often yield a solution faster than using a direct solver.

- **Assembly** Assembles a stiffness matrix only and is primarily intended for research use.
- **Fast Parallel Direct Solver** a high-performance, robust, memory efficient solver for solving large sparse symmetric and non-symmetric linear systems of equations on shared memory multiprocessors. This solver cannot currently be used for any form of eigenvalue analysis, or for superelement or Fourier analyses, or for nonlinear problems using branching and bracketing.
- **Fast Parallel Iterative Solver** a high-performance, robust, memory efficient solver for solving large sparse symmetric and non-symmetric linear systems of equations on shared memory multiprocessors. This solver cannot currently be used for any form of eigenvalue analysis, or for superelement or Fourier analyses, or for nonlinear problems using branching and bracketing.

In general, care should be taken when solving problems with high aspect ratios (thin or elongated), or excessively curved or distorted elements, or extreme or widely disparate material properties, since all of these can lead to ill-conditioning. Note also the convergence of the Iterative (Conjugate Gradient) solver is related to the condition number of the stiffness matrix which becomes worse for ill-conditioned problems.

Notes relating to the Iterative (Conjugate Gradient) solver

The Iterative (Conjugate Gradient) solver can be configured using the following parameters:

- **Drop tolerance**- a value between 0.0 and 1.0 which measures the amount of new non-zero entries (known as *fill-in*) allowed to remain in the preconditioning matrix during the incomplete Cholesky factorisation of the stiffness matrix. The default value is 1.0, leading to a very sparse preconditioning matrix suitable for well conditioned problems. For more ill-conditioned problems, however, this value should be decreased exponentially, and values in the range [1.0e-3, 1.0e-6] are recommended. The lower this parameter becomes, the larger the preconditioning matrix will be, giving rise to fewer iterations during the conjugate gradient solution, although each iteration will take longer to process. This parameter affects all preconditioning techniques, although the effect is less pronounced for the decoupled techniques.
- **Maximum number of iterations**- an upper limit on the number of conjugate gradient iterations to be processed; the default value is 5000. If the convergence criterion has not been satisfied when the iteration limit is reached, LUSAS Solver will issue a warning and then continue the analysis.

The following points need to be taken into account:

- Conjugate gradient methods can only be used for problems having symmetric positive-definite matrices. By definition, standard linear, static analyses yield positive-definite matrices in general, but mixed-formulation problems (such as pore pressure models) do not.

- Problems involving constraint equations cannot currently be solved with the iterative solver, since the resulting stiffness matrix is non-positive-definite.
- For problems with multiple loadcases, iterative solvers are less efficient since a separate iterative process is required for each loadcase, and the total time taken will increase in proportion to the number of load cases. By contrast, direct solvers incur very little extra cost when solving for multiple loadcases.
- **Guyan reduction** and **superelement** analyses cannot be solved iteratively, since matrix reduction does not take place.
- When using hierarchical basis preconditioning, if any midside degrees of freedom are supported or prescribed, their corresponding vertex neighbours must also be supported or prescribed. For example, if a midside node is fixed in the x-direction, all nodes on the same edge of that element must also be fixed (or prescribed) in the x-direction.
- The iterative solver will perform very poorly if there is not enough physical memory for the solution to proceed in-memory. To guard against this, a data check (OPTION 51) may be performed (as with the direct solvers), which will estimate the amount of memory the iterative solver would use with the specified drop tolerance and choice of preconditioning technique.
- The iterative solver has limited error diagnostics to warn against ill-defined or incompletely specified models. If this is suspected, the analysis should be run through the standard frontal solver for more comprehensive error diagnostics.

For further information see the *Solver Reference Manual*

Choosing a Solver

The default solver is the standard frontal solver and is used unless the fast solver option has been licensed in which case the fast multi-frontal solver is used. An alternative solver may be set from the **Solver Options** dialog under the **Model Properties> Solution** tab.

Direct (e.g. frontal) solvers are more robust than iterative solvers and are applicable to all types of analysis. Direct solvers should always be used for very **ill-conditioned** problems since the time taken to obtain a solution is independent of the problem conditioning. Iterative (e.g. conjugate gradient) solvers are usually only applicable to static, linear analyses, and will perform best on large, well conditioned problems, since the time taken for solution is less dependent on the size of the problem than for direct solvers. Iterative solvers require far less storage (memory + disk space) than direct solvers. This means that iterative solvers sometimes have the advantage of remaining in memory where a direct solver would have to run "out of memory". Iterative solvers are only applicable for a single loadcase.

Frontal Optimisation Options

The frontal optimisation options are set from the Solver Options dialog which is accessed via the Model Properties or Solver option control objects in the Analyses Treeview.

Frontal optimisation is only required when using the standard frontal solver. It is not required for the fast multi-frontal solver. When no optimiser is specified the Sloan optimiser will be used to optimise the front width for the standard frontal solver. The frontwidth of the problem may be reduced by optimising the order in which the elements are presented.

Optimising methods supported are:

- **Standard** uses the standard LUSAS optimiser.
- **Akhras-Dhatt** uses the Akhras-Dhatt optimiser. A number of iterations must be specified for this optimiser. The iterations are used to find the best starting point in the structure for the optimisation. The higher the number of iterations, the better the chance of locating the optimal starting point, but the longer the optimisation process takes.
- **Cuthill-McKee** optimises the solution based on the Cutill-McKee optimiser. This algorithm bases its optimisation on a specified parameter. Options are: maximum bandwidth, RMS wavefront, bandwidth and profile. This optimiser was originally written by E.H. Cuthill and J.M. McKee, and was improved by G.C. Everstine.
- **Sloan** uses the Sloan optimiser (default).

If no optimiser is specified then the Sloan optimiser is used by default.

Solver Licence Selection

When running an analysis, LUSAS Modeller passes all details of the licence it is running with to LUSAS Solver. It includes the minimum set of licence options required to solve the job, and a teaching and training identifier if Modeller is running in teaching and training mode.

To find a suitable Solver licence with which to run an analysis Solver does the following:

- By default a Solver licence with the same licence key number or 25 character key as the Modeller licence is sought and if available and valid is used.
- If the Solver licence that matches the Modeller licence is in use, or is invalid or unavailable, Solver will re-order all suitable Solver licences and internally list them such that the least functional Solver licence that is still able to solve the job is listed first.
- Once listed, Solver will tumble through the licences in the list until one is found that is both valid and available. This is then used for the duration of the analysis.

Creating shortcuts

Shortcuts can be created to tie a licence type to a shortcut used to run LUSAS Solver. For more information see the **LUSAS Configuration Utility.**

Pre-Analysis Checks

Prior to running an analysis it is prudent carry out a few basic checks:

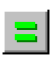
1. Check the consistency of your co-ordinate systems between the finite element model and any engineering drawing that you have worked from.
2. Check key drawing dimensions against co-ordinates of respective points in the model.
3. **Check the mesh for cracks and voids**. Checks for cracks must be made to ensure that the features form a continuous structure.
4. **Check for correct material properties and assignments**.
5. Check for consistent units.
6. **Check for correct orientation of beam properties.**
7. **Check for correct boundary conditions (loads/supports).**
8. **Check element thickness against original model data (plates/shells).**
9. **Check reversed normals for plates/shells/2-D.**
10. Check element shapes for **aspect ratio**, skew, **warp**, taper, **curvature** and **centrality of mid-side nodes**. Warning messages will be present in the output file for all of the above.
11. From the LUSAS datafile dialog, **File> Export Solver datafile...** menu item, click on the **Output** button to check that the output provides sufficient checking information in the LUSAS Solver output file (e.g. reactions).

Notes

- It is often a good idea to carry out a 'pilot' analysis on a crude model to check load paths and equilibrium.
- In order to ensure that an **adequate mesh density** is used a mesh sensitivity study should be carried out.
- It is good practice to keep an up to date log book with adequate plots (including hidden line views) to cover all parts of the model. It is also useful to set-up a reference system to select individual regions of the model using the groups facility.
- Keep a log of analysis runs for future reference. Note information such as element types, numbers of loadcases, frontwidth, file sizes, run-times, etc.
- Keep regular backups of model. This is done automatically by LUSAS Modeller. See **Model Files** for more details.

Running an Analysis

There are various ways in which an analysis can be run from within LUSAS Modeller.

The **Solve now** toolbar button  allows particular analyses to be selected for solving.

A context menu for each analysis name in Analyses Treeview provides additional Solving options:

- **Solve now** allows individual analyses to be solved in isolation from others that are present in the Analyses Treeview, and solves all loadcases defined within that analysis.
- **Loadcases to solve** allows selection of loadcases to be solved (the checkbox is ticked) when the Solve now toolbar button or context menu option are chosen. Loadcases where the check box are unticked will not be solved and these loadcases are denoted in the Analyses treeview with a red cross to show that they have been turned off for solving.

When using the Solve now toolbar button, the initial state of the checkbox alongside each analysis name on the Solve Now dialog shows whether an analysis needs to be solved (because results for that analysis do not exist or are out-of-date due to changes being made to the model after an initial solve had been done). Descriptive text alongside each analysis name shows the reason why an analysis needs to be solved or not. Analyses needing to be solved will be shown checked; those not requiring to be solved will be shown unchecked. An analysis will be solved (whether it needs to be or not) if the checkbox is ticked on. No solving will take place if a checkbox is off when the OK button is pressed.

Note that solving a model automatically creates a Solver data file, an output file and, if successful, a solver results file for each analysis.

If an Analysis is Successful

Results loadcases for each analysis are loaded into the relevant analysis entry in the Analyses Treeview in readiness for optional processing and viewing of results.

If an Analysis Fails

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

Errors and warnings reported in the Text Output window can be double-clicked to open the Identify Object facility which can be used to locate the referenced items. For an explanation of the errors which may occur see **Appendix B - LUSAS Solver Trouble Shooting**

When the termination of an analysis is due to a failure of the solution process rather than a problem with the modelling of the structure, if solver restart files were being written as the model was being solved, the solution may be restarted from the last converged increment with a different or modified solution strategy. See **Solver Restart Files** for details.

Identifying objects causing errors during a solve

Command confirmation and error messages are displayed in the Text Output window.

Double-clicking on an error or a warning reported in the Text Output window opens the Identify Object facility which can be used to locate the referenced geometric feature or mesh object. Options are provided to select the offending item in the View window, move it to the centre of the view, scale it to fit the screen, have its properties displayed, or be identified by an annotation arrow or a temporary indicator.

Post-Analysis Checks

All Analysis Types

1. Check the **deformed mesh** for each loadcase to ensure the model has deformed as expected.
2. Compare your finite element results with estimates of stress and deflection from hand calculations. This may not always be possible to do very accurately, but an estimated should be obtainable.
3. Check **reactions** for equilibrium.
4. Check magnitudes of displacements and stresses. If possible compare to hand calculation.
5. Check for matrix conditioning messages. **Small pivot** and **diagonal decay** warning messages are invoked when the stiffness matrix is poorly conditioned. Diagonal decay means that round-off error during the solution has become significant which could lead to inaccurate results. A poorly conditioned stiffness matrix is the result of a large variation in magnitude of the diagonal terms. This could be caused by large stiff elements being connected to small less stiff elements or elements with highly disparate values of stiffness (e.g. a beam may have a bending stiffness that is orders of magnitude less than it's axial stiffness).

   A negative pivot in a non-linear analysis usually means that a limit or bifurcation point has been encountered. However, negative pivots sometimes occur during the iterative solution (which sometimes means that the load step is too big) but disappear when the solution has converged. If negative pivots occur and the solution will not converge then first try reducing the load step.

   If the solution still does not converge a limit or bifurcation point may have been encountered in which case the solution procedure may need to be changed. Running the problem under arc length control gives the best chance of negotiating a limit or bifurcation point. A load limit point can also be overcome by using prescribed displacement loading.
6. Check the LUSAS Solver output file for other warning or error messages.
7. Check **adequate mesh density**.
8. Check average nodal stress calculations are not carried out across discontinuities.

9. Check the model summary information available in the LUSAS Solver output file. This gives the total length, area, volume and mass for the structure together with the centre of gravity, moments of inertia and resultant applied load at the origin.

Dynamic Analysis Types

1. Check the first natural frequency against hand calculation.
2. Check of convergence of the eigenvalue extraction algorithm.

Non-linear Analysis Types

1. Check convergence of the non-linear analysis.

Technical Support

Assistance with Modelling, Analysis and Results

Engineers in LUSAS Technical Support are available to help all clients with a current support and maintenance contract and assist with any modelling or analysis problems that may be encountered. Support may be requested by telephone or by email. Email is preferred contact method. All relevant files must be sent. See below for details.

Please note that if LUSAS was supplied to you by a LUSAS distributor then they should be your first point of contact for support. Your local distributor will have access to the main LUSAS Technical Support Centre for assistance if required.

What information should be provided?

As a guide, providing some or all of the following information will help the support engineers get to the bottom of your problem more quickly:

- The exact text of any warning or error message(s).
- Machine specification operating system, memory and available disk space.
- A copy of the model file data that is causing a problem (along with all associated model files - see below for details)
- A list of the last commands used in LUSAS Modeller or a copy of the session file.
- The contents of the last LUSAS Modeller error log LUSASM_x.ERR
- Full details of the LUSAS Modeller/LUSAS Solver version numbers in use, the LUSAS Solver version number is written to the header section of the output file and the LUSAS Modeller version number is obtained from the **Help> About LUSAS Modeller** menu item..

- For telephone support with complex or difficult to describe problems, emailing or faxing a simple diagrammatic representation before telephoning can aid in the understanding of a problem.

Sending files to LUSAS Technical Support

Small files may be emailed in isolation but note that LUSAS model files normally refer to many other associated files. For these models, use the **LUSAS Support Tool** to create a compressed file of all relevant data to send to LUSAS Technical Support.

If within sending and receivable email limits, files may be emailed, but if larger, files can be uploaded to a LUSAS Support Area using FTP.

LUSAS Technical Support can be contacted at support@lusas.com

Web Resources

A number of resources are also available via the main LUSAS website.

Main LUSAS website

http://www.lusas.com

General support services

http://www.lusas.com/support

LUSAS User Area

http://www.lusas.com/usrcheck.html

Note that the LUSAS User Area requires a username and password for access. This will be provided initially to the named LUSAS contact within your organization who should be consulted if necessary. It can also be requested by and sent to registered users of LUSAS. The User Area contains:

- Answers to the most frequently encountered problems when using LUSAS
- Detailed instructions for carrying out various tasks with LUSAS
- Theory on many aspects of LUSAS and finite element analysis in general
- Explanations and remedies for specific LUSAS error/warning messages
- Technical and ‘How to’ notes
- Knowledge base of recently fixed software errors and outstanding issues
- Links to online copies of LUSAS manuals for recent versions (PDF format)
- Details of recent LUSAS software releases, LUSAS scripts and other useful files.
- And more...

LUSAS Support Tool

The LUSAS Support Tool ensures that if a support query is raised all relevant files can be submitted to LUSAS Technical Support to aid with the resolution of the query. It is accessed using the **Help > Support Tool** menu item. The following data can be optionally included in the archive:

- ❑ **Vehicle Load Optimisation settings and input files**
- ❑ **Model and session files**
- ❑ **Results file (.mys)**

A single file is created in a compressed file format containing selected data relating to the current model. The file is created in the current working directory with a .zip file extension, using the model name and date and time of creation as the filename.

Automated Error Reporting

LUSAS Modeller makes use of a third party crash reporting facility (CrashRpt) to allow users to optionally submit crash reports to LUSAS for investigation. If a crash occurs a crash dump is also assembled.

The automatically assembled error report contains useful information relating to the operating system in use, the amount of memory in use by Modeller at the time, where in LUSAS the error occurred, and the state of the program when the error occurred. No personal information whatsoever is obtained.

It is requested that crash report information be sent to LUSAS for the benefit of other users, and to aid in the continual improvement of the software. Additional information, such as an email address, and a brief description of what was been carried out at the time of a crash can be optionally given.

An option to **Restart LUSAS Modeller after submitting the error report** is also provided. This enables a **model recovery** to be carried out.

Recovering from a crash

If a crash has occurred, and if the option to restart LUSAS Modeller after submitting the automated error report was chosen LUSAS will automatically be re-started, and display startup dialogs culminating in the ModelRecovery dialog where options for recovering the previous session are presented. See **Model Recovery** for more details.

Not-automated Error Reporting

For a fully documented investigation of a problem a Support Query should be raised by email or telephone with **LUSAS Technical Support**. This will normally require a model and all associated files to be provided.

Model Recovery

The model recovery dialog will appear if a model selected for opening had, in a previous LUSAS session, been interrupted unexpectedly before it could be saved, resulting in a recovery file being created. This may have been because of a power failure, a program error, or from a deliberate interruption. The last command carried out prior to the interruption is reported in the dialog. There is a choice in how the model should be recovered from this point:

- **Recover whole session** Choose this option to recover the whole session (carrying out all the commands you made) including the last command shown. This would generally be chosen unless the last command consistently caused similar problems that prevented a recovery from taking place.
- **Recover all except last command above** Choose this to carry out all commands except the one shown. This would be chosen if it was known, or subsequently found, that the last command shown caused the interruption or prevented a model recovery from being made.
- **Open last saved model** Ignores all commands and changes made to the model since the last save. The last saved model file of the same name is opened.

- **Preserve a backup copy of the model and recovery file** This option is selected by default and is used to save all files related to an interrupted session for sending to LUSAS Technical Support.

Chapter 8 : Viewing the Results

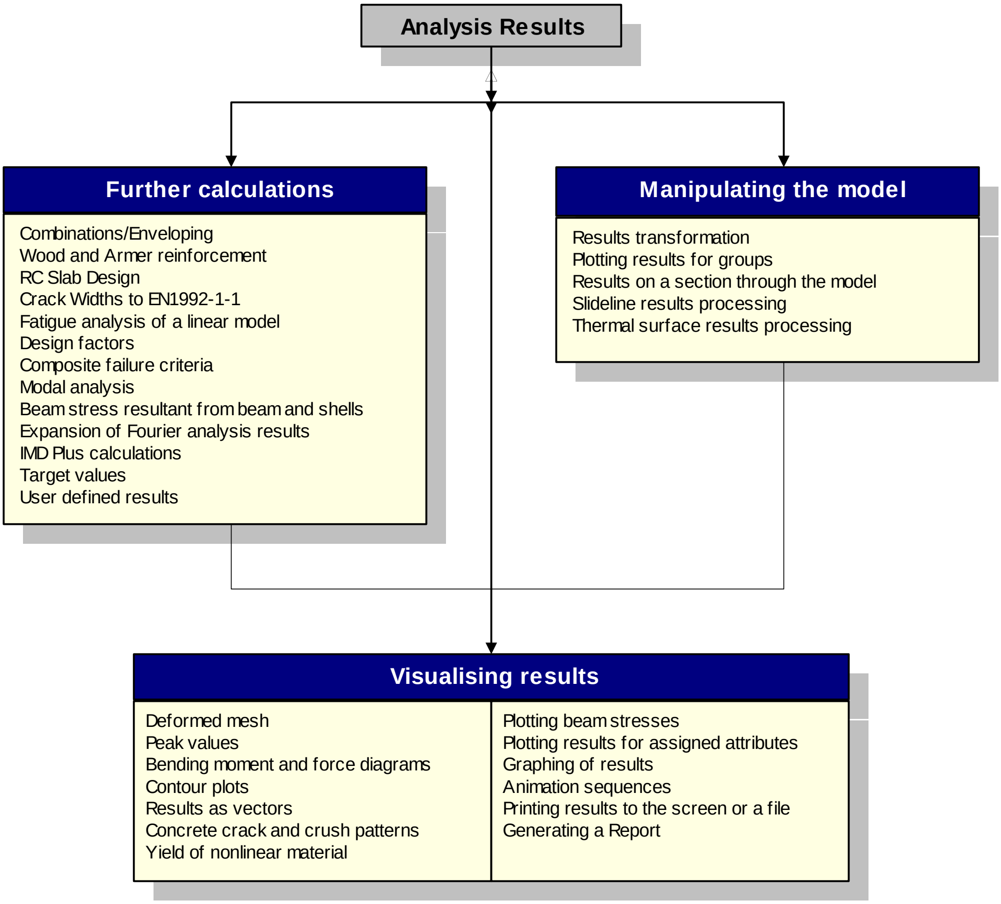
Introduction

This section deals with procedures for results processing. It covers manipulation of results files, selection of the correct results type and loadcase, and differences between results viewing coordinate systems.

- **Results Processing** provides an overview of results processing.
- **Results Files** covers manipulation of results files.
- **Results Selection** covers selection of the active loadcase, a fibre location and a composite layer. It outlines all the different results types available during post-processing.
- **Results Transformation** provides options for transforming results to a consistent or alternative coordinate system

Results Processing

Results processing, also known as post-processing, is the manipulation and visualisation of the results produced from an analysis. Prior to visualising and extracting results, further calculations may be carried out to create or assemble results, or the results model can be manipulated to create results at particular model locations for a particular results viewing use. This diagram summarises what is possible:



Further Calculations

- Combinations and envelopes
- Wood and Armer reinforcement
- RC Slab Design
- Crack Widths to EN1992-1-1
- Fatigue analysis of a linear model
- Design factors
- Composite failure criteria
- Modal analysis
- Beam stress resultant from beam and shells
- Expansion of Fourier analysis results
- IMDPlus calculations
- Target values
- User defined results

Manipulating the Model

- Results transformation
- Plotting results for groups
- Results on a section through the model
- Slideline results processing
- Thermal surface results processing

Visualising and Extracting Results

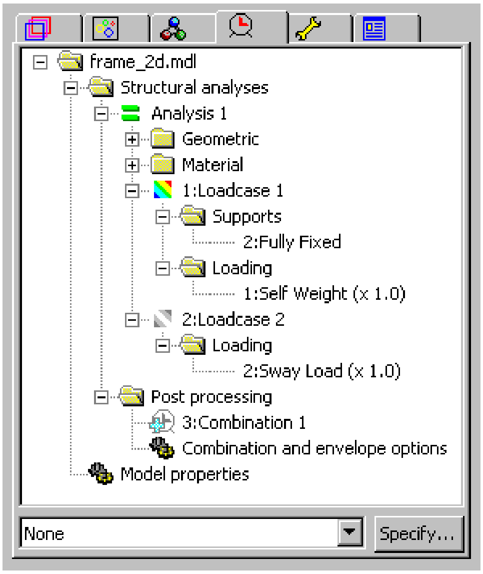
- Deformed mesh
- Peak values
- Bending moment and force diagrams
- Contour plots
- Results as vectors
- Concrete crack patterns
- Yield of nonlinear material
- Plotting beam stresses
- Plotting results for assigned attributes
- Graphing of results
- Animation sequences
- Printing results to the screen or a file
- Generating a Report

Results Files

When an analysis is performed by LUSAS Solver a results file will be created. For historical reasons this has a **.mys** extension. By default the results file is automatically loaded into LUSAS Modeller after **LUSAS Solver** has been run.

The information in the Solver results file is stored in a binary form and may only be accessed using LUSAS Modeller. The results file will contain the results of the analysis and sufficient model information to process the results. Full details of the finite element mesh (nodes and elements), material and geometric property numbers, support positions and equivalent nodal loads are stored in the results file.

Opening available results files for a model



When a model is opened, Modeller will attempt to open all available associated results files for that model. The **File > Open Available Results Files** menu item also exists for the same purpose. Results files are loaded on top of the current model and results loadcases are added to the relevant Analysis entries in the Analyses Treeview.

Opening a model with results files enables the visibility of the model to be controlled by the assigned attributes. In addition, all model data including group information is present to aid results manipulation. Supports and loading attributes as assigned for each loadcase can also be seen. When results are loaded on top of a model file, only results data is loaded from the .mys file. Multiple results files may be loaded to allow accessing of results from a number of analyses at the same time. See **Manage Results Files** for details.

The Analyses Treeview (right) shows a typical listing obtained when a results file is loaded for a model.

Closing Results Files

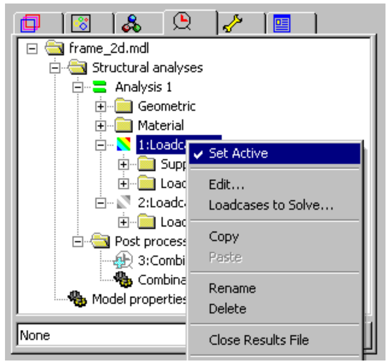
All results files for all analyses can be closed by selecting the **File> Close all Results Files** menu item.

Results files for an individual analysis can be closed by using the context menu item **Close Results File** that is provided on each loadcase. Note that if the results loadcase selected was the first results loadcase loaded (meaning that by default it contains mesh data used by all other results files opened after it was loaded), all results files will be closed.

Results Selection

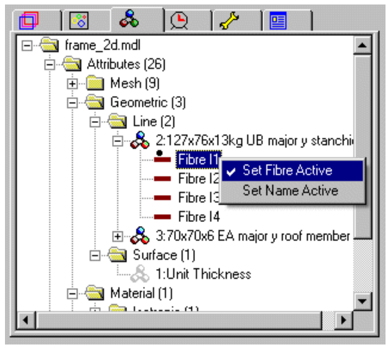
Setting the Active Loadcase

The active loadcase is the loadcase that provides the results for the current window. It is set by choosing the Set Active menu item from the context menu of a loadcase in the Analyses Treeview. In this way a single window is used to plot results from a single loadcase, and multiple windows can be used to compare results from different loadcases.



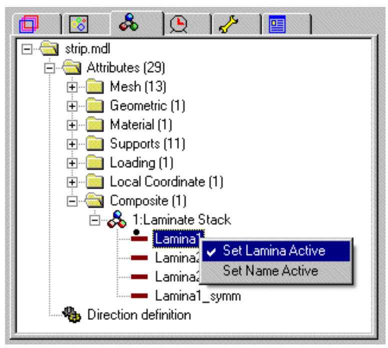
Setting the Active Fibre Location

For plotting diagram results on bars and beams <u>only</u>, the active fibre location for which results will be plotted must be chosen. It is set by choosing the Set Fibre Active menu item from the context menu of a fibre entry held within a geometric line attribute in the Attributes Treeview. Fibre locations can visualised by double-clicking on the Geometric Line name and selecting the Visualise button on the dialog presented. Setting the Fibre Active shows results for just that fibre. Setting the Name Active shows results for all similarly named fibres throughout the model.



Setting the Active Composite Layer

With composites analysis, in order to plot results on composites laminae a particular lamina must be made active. It is set by choosing the Set Lamina Active menu item from the context menu of a lamina entry held within a Laminate stack attribute in the Attributes Treeview. Setting a Lamina active shows results for just that lamina. Composites layer information can be seen and visualised by double-clicking on the Laminate stack name and selecting the Visualise button on the dialog presented.



Results Types

There are several different types of results entities available in each results file. Full details of the results available for each element type may be found in Appendix K of the *Element Reference Manual*.

Each time results are displayed the results entity must be specified. Only result entities that are actually contained in the loaded results file are presented for selection. The following is a summary of all the results types:

Structural Analysis

- ❑ **Displacement**
- ❑ **Stress**
- ❑ **Strain**
- ❑ **Loading**
- ❑ **Reaction**
- ❑ **Reaction Stress**
- ❑ **Residual**
- ❑ **Velocity**
- ❑ **Acceleration**
- ❑ **Plastic Strain**
- ❑ **Creep Strain**
- ❑ **Rubber Stretch**
- ❑ **Strain energy**
- ❑ **Plastic work**
- ❑ **Slideline**
- ❑ **Named variables**
- ❑ **State variables**
- ❑ **Slab Design (RC)**

Thermal Analysis

- ❑ **Potentials**
- ❑ **Fluxes**
- ❑ **Gradients**
- ❑ **Thermal Surface**

Notes

- Stresses are stress resultants for beams, plates and shells (i.e. forces for beams and force/unit width for plates and shells).
- For plates and shells top, middle and bottom stress and strain are available.
- When applicable, Wood Armer results, composite failure values, design factors and beam stresses are available from the Stress results type.
- The results calculation and display may be controlled independently using the Result Plots dialog activated from the top level of the Groups context menu. See **Groups** for details.
- See **Results Transformation** for general notes regarding the orientation of results.

Results Transformation

Viewing Results in a Consistent Direction

When viewing results in Modeller it is important to note that:

- **By default, displacements, loads, reactions, residuals, velocities and accelerations are output relative to the global Cartesian axis system.**
- **By default, stress/strain and creep/plastic strain results (except for beam/bar elements and interface elements) are output relative to the global axes. Forces and moments (stress resultants) for beams, shells and joints are output relative to the element local axes.**
- **For shell and plate elements, and in accordance with established theory, moments are given along the chosen axis. For beams results moments are given about the chosen axis.**

If the elements in a model are orientated such that their local axes vary from one another (and note for surfaces they may also vary from the surface orientation), or if an alternative coordinate system is required, the results may be transformed to a consistent direction.

- Results displayed on contour, values or vector layers in a particular view window can be transformed using the **Transform results** button on the layer properties dialog for each layer.
- Results plotted using the graph wizard, or obtained using the print results wizard, or the report writer can be transformed using the **Transform results** button on their respective dialogs.

Results may be transformed to the following directions:

- ❑ **No transformation applied (consult Solver manual)** Results may be in global and/or element local axes according to element formulation. Depending upon the type of model and elements used a results transformation may be required.
- ❑ **Local axes or element/node** Transforms nodal results (displacements, reactions etc.) according to the local transformed freedoms or element results (stresses, strains etc.) to element local directions.
- ❑ **Local coordinate of parent feature** Transforms element results according to the local axes of the line, surface or volume feature.
- ❑ **Global axes** Transforms nodal or element results to the global axes.
- ❑ **Material direction** Transforms element results to the local element material directions.
- ❑ **Specified local coordinate** Transforms results to a consistent direction using a **local coordinate**.

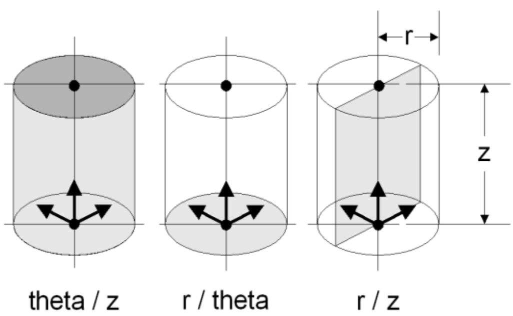
- ❑ **Shell plane for resultants** enables the plane for shell moments and shear forces to be defined when using cylindrical or spherical coordinates. See **Shell plane for resultants** below.
- ❑ **Reference path** Transforms nodal or element results relative to the angle or route of a **reference path**.

Notes

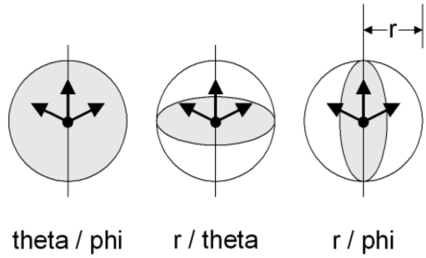
- Different model view windows can each have different Results Transformation settings.
- Beam and bar elements always have their local x direction along the length of the element.
- For plates and shells the normal direction is defined by the local element z axis and this cannot be transformed. A results transformation only applies to the orientation of the local element x and y directions in the plane of the shell element. Top/Middle/Bottom, Wood Armer top and bottom, and sign conventions rely on the element z axis.
- When viewing Wood Armer results a transformation is used to set the direction of the X reinforcement. A skew angle may then be set independently to define the Y reinforcement direction. The default skew angle is 90 degrees.
- When viewing shell moments and shear stresses using a cylindrical or spherical local coordinate the shell plane for resultants needs to be defined as theta/z, r/theta or r/z.
- Using the transformation options for joint elements has no effect.
- With the exception of the **No transformation applied** option the type of transformation used is written to the Contour key.

Shell plane for resultants

The following diagram illustrates the shell planes available for a cylindrical coordinate system.



The following diagram illustrates the shell planes available for a spherical coordinate system.



Combinations and Envelopes

Combinations and envelopes enable the results from several loadcases to be presented as a single set of results.

Basic combinations

These enable results from individual loadcases (possibly from more than one analysis), other basic combinations, smart combinations or envelopes to be factored and added together.

Smart combinations

These enable results from individual loadcases (possibly from more than one analysis), basic combinations, other smart combinations or envelopes to be factored with an adverse or relieving factor and added together.

- A **Permanent load factor** is always applied
- A **Variable load factor** is only applied if the effect is adverse. This means a maximum smart combination (Max) will assemble results from the loadcases selected using just the permanent factors for negative load effects, and using permanent and variable factors for positive load effects, and a minimum smart combination (Min) will assemble results from the loadcases selected using just the permanent factors for positive load effects, and using permanent and variable factors for negative load effects.

The Permanent load factor and Variable load factor names may be optionally displayed as **Beneficial load factor** and **Adverse load factor** if the check box on the Combination and envelope options dialog (see later) has been selected. Note that:

- Beneficial load factor = Permanent load factor
- Adverse load factor = Permanent load factor + Variable load factor

To support combinations for design codes check box options can be used to limit the total number of loadcases and the number of variable loadcases to be included in the smart combination:

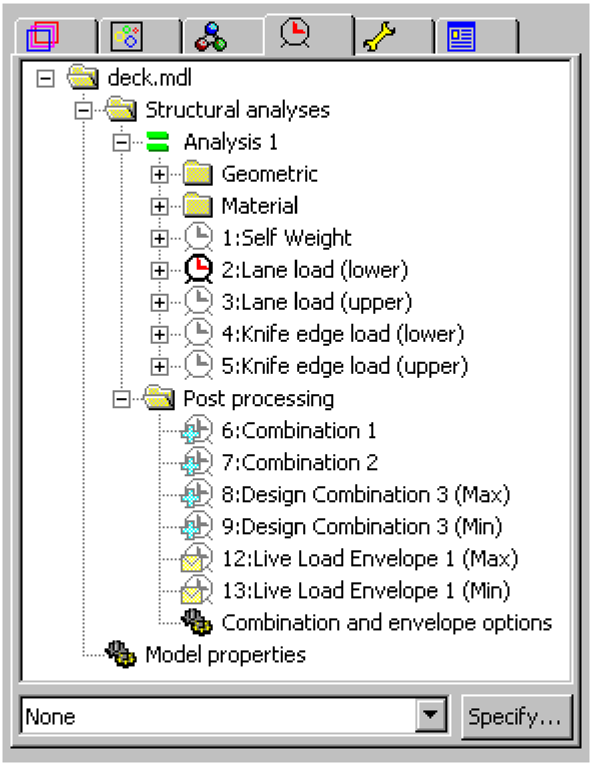
- If the **Loadcases to consider** option is chosen and an appropriate number entered, the smart combinations will be assembled in the manner described above, but the number of loadcases considered will be restricted to the number of loadcases specified. The loadcases used will be the most adverse for each combination i.e. the most positive for maximum combinations and the most negative for minimum combinations. All other load effects will be discarded.
- If the **Variable loadcases** option is chosen and an appropriate number entered, the smart combinations will be assembled in the manner described above, but two further criteria will be invoked. The number of variable load factors used will be restricted to the number specified. The most adverse variable factors will be used in each combination and the remaining loadcases, which produce load effects of the correct sign, will be included with their permanent factors only.

Envelopes

Envelopes enable the maximum and minimum results from individual loadcases (possibly from more than one analysis) or other from other combinations or envelopes to be determined.

Defining a Combination or Envelope

The Analyses menu is used to define a combination or envelope. Combinations or envelopes may be defined at either the modelling stage prior to analysis, or at the results processing stage. Once defined, combinations and envelopes are treated as loadcases and are listed in Post processing folder in the Analyses Treeview. Their properties may be edited by double-clicking on the relevant Basic or Smart combination, or Envelope entry.



User-defined loadcases such as envelopes, basic and smart combinations, and fatique, IMD and target values loadcases (that are also held in the Post processing folder) are held with the model file. Other results loadcases are held for each analysis entry that is present in the Analyses Treeview.

Combinations and envelopes definitions are saved in the model file when it is saved.

Superposition

Combinations are only generally applicable to linear elastic analyses. This is because the principal of linear superposition is used where, if Load A produces Effect A and Load B produces Effect B, then Load A and Load B applied together produce an effect equal to Effect A + Effect B. In a linear static analysis this assumption holds true for all results components that are calculated by a LUSAS Solver, but note that it does not hold true for all results components that can be calculated by LUSAS Modeller. For example, Wood Armer calculations are not linear calculations - so superposition is not safe for these. Similarly, the definition of any user-defined results components (that can be used within combinations) may include equations with terms or constants that are not scalable, meaning that care should be taken when using these also. Superposition also is not true when lift-off or contact supports are used in loadcases used in load combinations.

Expanding a basic combination

LUSAS assumes superposition is valid when combinations are assembled. In situations when superposition may not hold true (as when lift-off supports are present) a nonlinear solution may be required. The factored loadcases from a basic combination may be created as a new loadcase from the basic combination definition provided the basic combination does not include any smart combinations or envelopes.

A basic combination can be expanded to create a new loadcase for single or multiple combinations by using the context menu for a combination or for the post processing folder.

Combination and envelope options

When the first load combination or envelope is added to the Analyses Treeview, a Combination and envelope options object is also created. Double-clicking on this object displays a dialog on which results components can be selected for calculating and saving in the **Modeller results file**. When results components are selected prior to an analysis being carried out the results for the primary components chosen will be available for results processing immediately after the results file is loaded because results for these components will have been automatically saved (cached) in the Modeller results file. When results components are selected on the Combination and envelope options dialog after an analysis has been carried out an option to calculate results for the components selected is provided. These results will also be cached in the Modeller results file to speed-up results viewing. If a model is saved these calculated results will also be saved.

Visualising the results from Combinations and Envelopes

Once defined, combinations and envelopes may be manipulated in the same way as other **loadcases**. In the Analyses Treeview, right-click on a combination or envelope and select **Set Active** to view results.

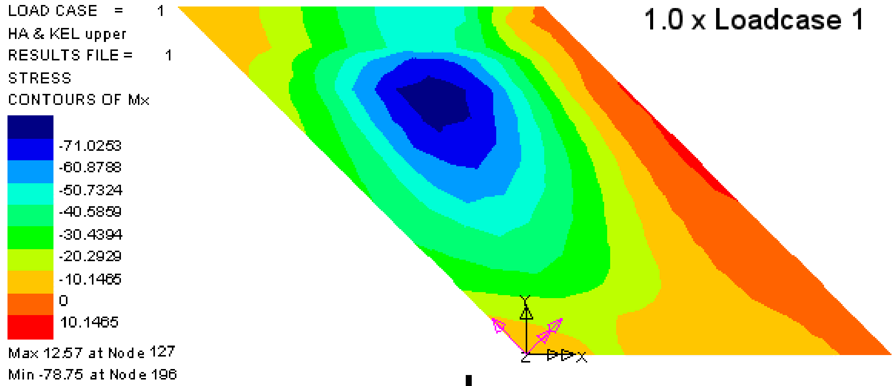
When using smart combinations or envelopes the **Set Active** menu item will prompt for the primary component on which to base the combination or envelope. For an envelope this component will be used to decide which is the maximum or minimum loadcase and for a smart combination which factor to apply to each loadcase. When displaying or printing results the values shown for other components will be the coincident effects. For envelopes, if no component is specified all components are enveloped independently.

An active loadcase is identified by a coloured icon in the Analyses Treeview. Non-active loadcases are greyed-out. For envelopes and smart combinations either the maximum or minimum can be set active.

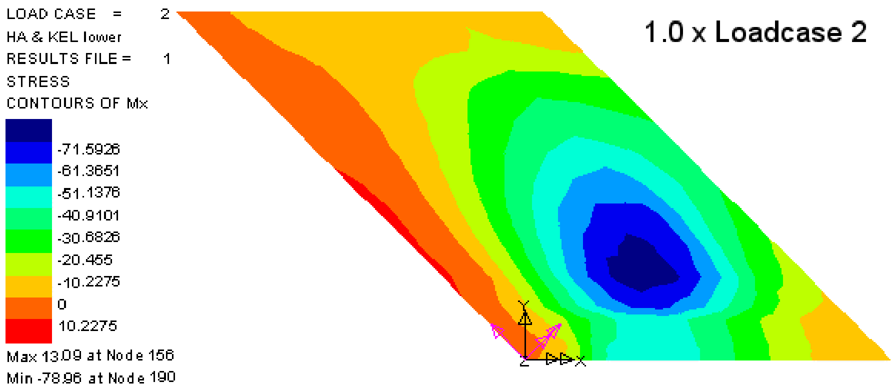
Notes

- When an envelope or smart combination includes another envelope or smart combination both the (Max) and (Min) loadsets should be included in the definition. The envelope or smart combination (Max) will ignore the envelope (Min) or smart combination (Min) loadsets and the envelope or smart combination (Min) will ignore the envelope (Max) or smart combination (Max) loadsets.
- Sometimes it may be convenient to split the loadcases into more than one analysis. The loadcases from the separate analyses may then be subsequently enveloped or combined.
- When envelopes of envelopes or combination results are calculated they are automatically cached. Saving the model will also save the cached results to the Modeller Results File so they are available for future use. Subsequent access to these results will be similar in speed to accessing results from a single loadcase.
- When combining or enveloping results from multiple results files the mesh must be identical across all results files.
- If the loadcase IDs which contain the most adverse effects are not required, enveloping can be significantly speeded up by placing the envelope within an envelope as the results will then be cached.
- Spectral (IMD) loadcases can be combined with other loadcases such as those defining dead and live loads. Since spectral loadcases are computed from an eigenvalue analysis the sign of the displacements are always positive but the most adverse effects can be obtained by creating a combination including dead/live load and a spectral loadcase both with load factors of 1 and then creating a combination including a dead/live load with a load factor of 1 and a spectral loadcase with a load factor of -1.
- For further examples regarding Smart Combinations see **Appendix A.**

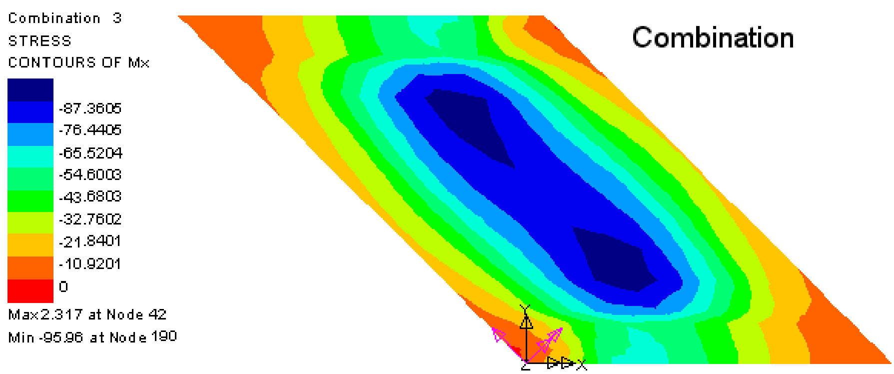
Basic Combination Example



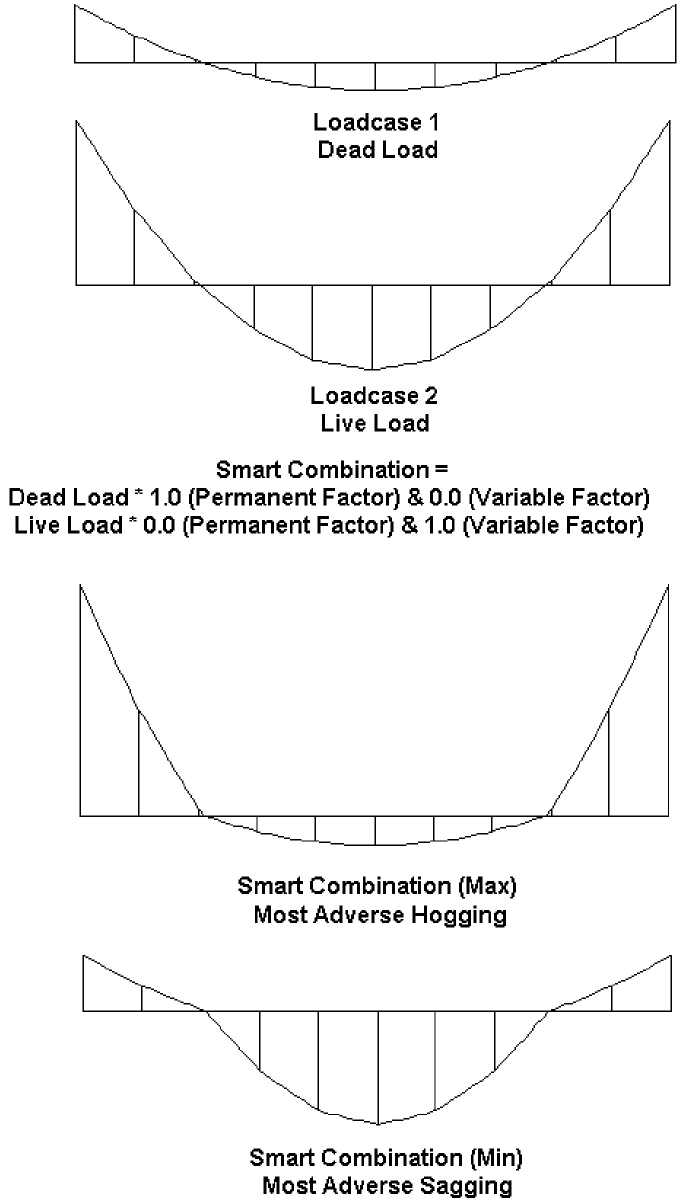
+



=



Smart Combination Example



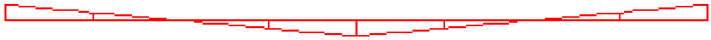
For details of smart combination calculations see **Appendix A**

Enveloping Example

Individual Loadcase Bending Moment Results

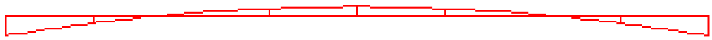
Loadcase 1

Loadcase 2



Loadcase 3

Loadcase 4

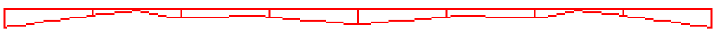


Maximum Enveloped Bending Moment Diagram

Maximum

Minimum Enveloped Bending Moment Diagram

Minimum



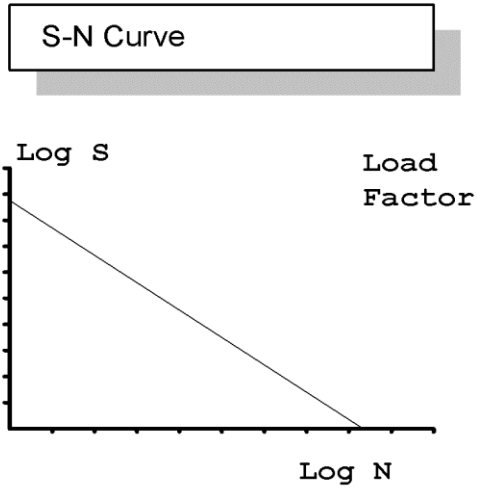
Fatigue Calculations

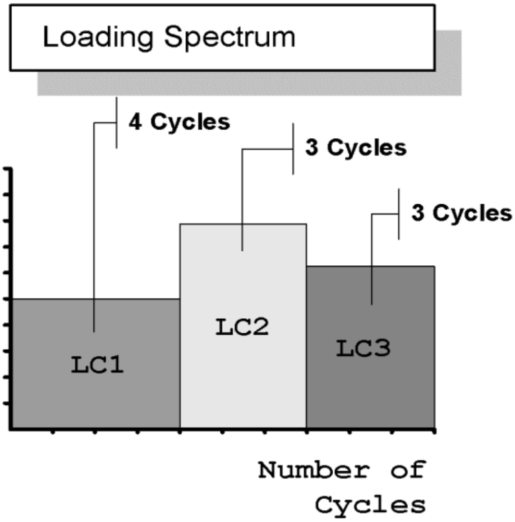
Fatigue calculations can be performed on the results of a linear finite element stress analysis using the total life approach. This can be done for continuum elements only. The fatigue life may be expressed in terms of the damage that is done to the structure by a prescribed loading sequence or as the number of repeats of the sequence that will cause failure of the structure. Contour plots illustrating the fatigue life of the entire structure can be generated. The results from fatigue calculations may be viewed using any of the standard plotting techniques.

Fatigue calculations of the life of a structure are defined from the **Utilities> Fatigue** menu item.

S-N Curves

S-N curves contain the variation in stress/strain values with the number of cycles to failure and are defined on a Log Log scale. An S-N curve is used to calculate the number of cycles to failure for each loadcase. Miner's rule is then used to combine the damage for each loadcase to give the total damage to the structure for the specified loading sequence.



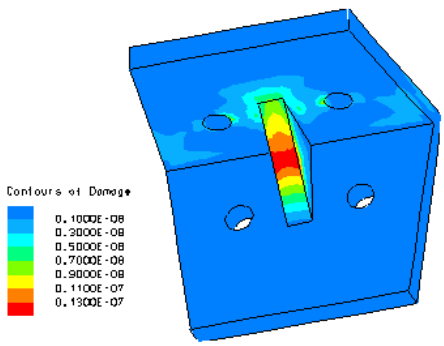


Notes

- Log values are used because of the large variation in magnitude of the input data.
- If plotting the number of cycles to failure, then the number of cycles entered in the fatigue spectrum should sum to unity.
- For information on standard S-N curves refer to BS5400.

Fatigue loadcase

Fatigue loadcases contain the loading spectrum defining the loading sequence in terms of a series of loadcases, each of which has an associated load factor, the number of cycles, and the component to be used in the fatigue calculation.



Once defined fatigue loadcases are saved in the Analyses Treeview. Their properties may be edited by double-clicking on the fatigue loadcase.

Fatigue Results

Results from the fatigue calculations may be viewed using any of the standard methods, once the loadcase is **set active**. Fatigue results come from a component of stress, which is specified in the Fatigue loadcase. Two results are obtained from the calculations:

- ❑ **Loglife** is the life expectancy of the structure based on the applied load. Results are presented as $\log_{10}$ of the number of cycles to failure.
- ❑ **Damage** is a factor representing the damage the material has sustained due to the applied loading and number of cycles. A value greater than 1 indicates failure.

Notes

- Fatigue calculations are only applicable to linear elastic analyses and for continuum elements only.
- Fatigue loadcases may be saved in the model file or in a new model file when a results file only is opened.

Interactive Modal Dynamics

The Interactive Modal Dynamics (IMD) facility within Modeller calculates the modal response of a system to a given input using the eigen modes and eigenvectors from an **eigen analysis**. (Note that the eigen analysis must have been performed with mass normalised eigen modes). An IMDPlus software option extends this capability to solve 2D and 3D seismic and moving load analyses using modal superposition techniques in the time domain. For details see the *IMDPlus User Manual*

Interactive Modal Dynamics calculations within Modeller may be performed on single or multiple eigen modes, and at a single node or over the whole structure:

- ❑ **Response at a node** Use the Graph Wizard to produce a graph of a specified results type against sample frequency range or series of time steps.
- ❑ **Response for all nodes** Use an IMD loadcase to calculate the modal response of the whole structure to a specific frequency or at a particular response time. The results from the IMD calculation are then viewed using any of the standard plotting techniques such as contouring.

Assumptions

The working assumptions for the modal dynamics facility are as follows:

- ❑ **Linear** The system is linear in terms of geometry, material properties and boundary conditions.
- ❑ **No Cross-Coupling** There is no cross-coupling of modes caused by damping. This is reasonable as long as the damping of the structure does not exceed 10% of critical damping.
- ❑ **Low Modes Dominant** The response is dominated by the lowest few modes.

Performing Modal Response Calculations

Both the **Graph Wizard** and **IMD Loadcase** commands are initiated from the **Utilities** menu. The basic steps for both methods are as follows:

1. Decide whether to perform a response analysis on a single node (Graph Wizard), or the whole structure (IMD loadcase).
2. **Select Eigen Modes** Specify which modes to include in the analysis.
3. **Damping** Specify the amount of modal damping.
4. **Excitation** Apply a form of dynamic excitation to a specific node or at the supports.
5. **Results Type** Choose the IMD calculation required, then set the required parameters.

Modal Damping

Modal damping is the damping associated with the displacements defined by the eigenvectors. Its value has no physical significance since the eigenvector contains an arbitrary normalising factor.

Damping values can be specified explicitly or alternatively can be extracted from the results file. LUSAS Solver will only provide modal damping estimates if the relevant damping control data has been included in the eigenvalue analysis. By default, LUSAS Solver values will be zero. Damping is specified using percentage viscous and structural damping values.

Notes

- Structural damping is not used in modal response calculations in the time domain.
- Structural damping is not applicable in spectral response calculations.
- All damping ratios are expressed as percentages of critical damping.
- Modal superposition techniques are not usually appropriate for structures with damping ratios higher than 10%, due to coupling between modes. Step-by-step dynamic analysis should be considered in such cases.

Dynamic Excitation

Eigenvalue analyses do not consider any applied load. Therefore, to carry out a dynamic structural analysis a load must be applied to the structure. These loads are specified either in terms of **forces**, or as motion by use of the **large mass method**:

- ❑ **Point Force** To set the modal excitation to point force via a node number and a nodal freedom. This is equivalent to applying a unit concentrated force at the selected nodal degree of freedom. The value of the unit force depends on the chosen system of units. In the case of SI units, a force of 1 Newton or a moment of 1 Newton metre will be applied. The amplitude of the load may be specified.
- ❑ **Point Displacement** (Large mass method) Sets the modal excitation to point displacement via a node number and a nodal freedom. A force equal to the large mass is applied at the support point, thereby inducing a unit acceleration response, see note on large mass. Displacement control is effected by integrating the response twice in

the frequency domain. Time domain displacement excitation is not currently supported.

- ❑ **Point Velocity** (Large mass method) Sets the modal excitation to point velocity via a node number and a nodal freedom. A force equal to the large mass is applied at the 'support' point, thereby inducing a unit acceleration response, see note on large mass. Velocity control is effected by integrating the response once in the frequency domain. Time domain velocity excitation is not currently supported.
- ❑ **Point Acceleration** (Large mass method) Sets the modal excitation to point acceleration via a node number and a nodal freedom. A force equal to the large mass is applied at the 'support' point, thereby inducing a unit acceleration response, see note on large mass.
- ❑ **Real/Imaginary/Real & Imaginary Loading** A load vector is extracted from specified eigenvalues (loadcases). In this case the modal forces are complex and therefore the modal forces are stored in two arrays - one for the real component and one for the imaginary. (For excitation types other than these the imaginary components will always be zero). Note that it is assumed that all applied forces are in phase. The chosen load vector need not be in the same results file as the eigenvectors used for the response calculation - typically only one load vector will be stored with the modes.
- ❑ **Support Motion** To set the modal excitation to support motion. This is used when all the supports move together, for example in the analysis of an earthquake or when a small component attached to an airframe or vehicle chassis has a known vibration level. The calculations are based on participation factors calculated by LUSAS Solver. A participation factor defines the modal force resulting from a unit acceleration loading applied to the whole model in a specified direction. The participation factors for each mode in each of the global directions are stored in the results file and are used as modal forces in the support motion calculations. X, Y, Z direction motion or motion in any vector direction can be represented. General support motion can be modelled by applying a unit acceleration field to the model using body force loading, and selecting the resulting equivalent nodal forces as modal excitation. In the frequency domain absolute or relative support motion may be selected, the default being relative.

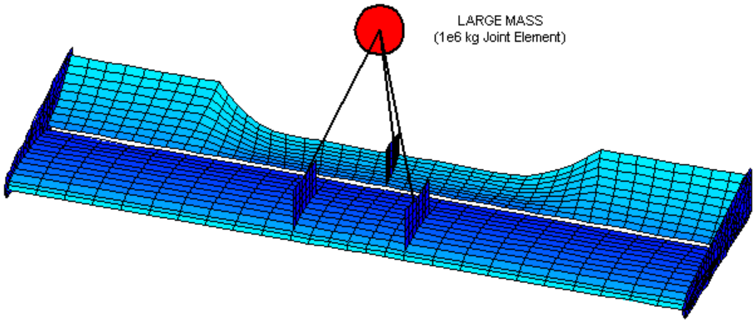
Notes

- Displacement and velocity excitations are not allowed for modal response calculations in the time domain.
- Support Motion excitation may be the most appropriate if all the supports move together.
- Large Mass point displacement, velocity or acceleration excitation may be the most appropriate if the supports do not move together.
- For Point Displacement, Velocity and Acceleration excitation, the displacement response is absolute and not relative to supports.

- When using Support Motion excitation in the frequency domain, the displacement response may be specified as relative or absolute. If absolute is chosen then the motion of the support is added to the structural motion to give motion with respect to ground. This is useful for comparison with measured data.
- When using the IMD Impulse excitation option, the magnitude of input will be the Impulse value (not strictly a force). The Impulse unit will be [force x time].

The Large Mass Method

Interactive Modal Dynamics frequently employs an analysis technique referred to as the Large Mass Method. The reason for this is to earth the structure via a 'moveable' object rather than a strict support to ground. This allows subsequent application of a force to the mass, in effect applying an acceleration to the structure. The size of this mass should be sufficient to ensure **mass dominated** local response, so that the motion of the point is described by Newton's Second Law (i.e. F = ma). A mass of 1E6 kg works well for most structures. Unduly large values may cause ill-conditioning problems. A force equal to the large mass is applied at the **support** point, thereby inducing a unit acceleration response.



Modal Dynamics Results Types

- ❑ **Frequency domain response** Harmonic, or forced, response analysis is used to investigate the effects of structural resonance (where structures are forced to vibrate harmonically at or near their own natural frequencies). Solution of the harmonic response problem as a modal analysis avoids the need to perform a full transient dynamic analysis. Simultaneously applied excitations may contain phase differences.
- ❑ **Time domain response** (impulse or step-by-step dynamics) to dynamic excitation. The forcing function and the consequent response of a structure are defined in terms of time histories. The Fourier transform of the time domain gives the corresponding quantity in the frequency domain.
- ❑ **Spectral response analysis** An analysis in which a defined response spectra for a generic earthquake ground motion is used to estimate the maximum displacement or pseudo-velocity or acceleration during the earthquake, without the need for direct integration of the model over the complete duration of the event. Dynamic excitation is applied to all the supports simultaneously. A response spectrum curve defines the

magnitude of excitation. If the damping in the response spectrum curve differs from that defined for the model a damping correction may be applied using one of formula provided. The maximum displacements, forces and stresses are computed throughout the structure for each eigenmode. These values may then combined to produce a single positive result using a spectral combinations. The spectral combinations available are CQC (default), SRSS and Absolute Sum. For further details see the *LUSAS Theory Manual*.

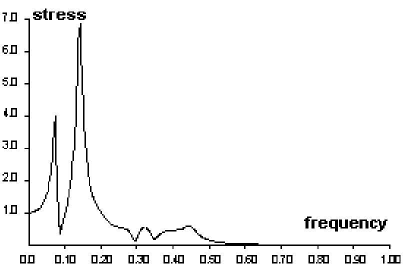
- **Power Spectral Density** (PSD) Analyses the frequency response to a random modal vibration, such as aerodynamic loads acting on an aircraft component. A frequency PSD defines the frequency content of the random loading. Dynamic excitation should be applied to all the supports via support motion excitation.

Frequency and Time Domain Response

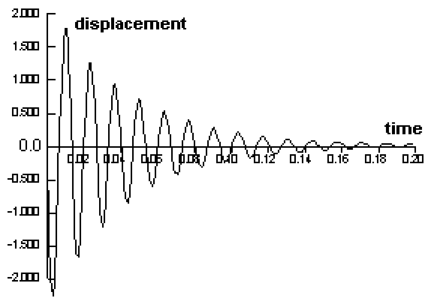
Frequency and time domain response calculations are the most commonly used. Usually, a node is excited across a frequency (or time) range to generate a graph for the frequency or time response (using the **Utilities> Graph Wizard** menu item). From the response across a range, a single frequency may be selected to perform an IMD calculation on the whole structure (using an IMD loadcase). The following diagrams illustrate this procedure:

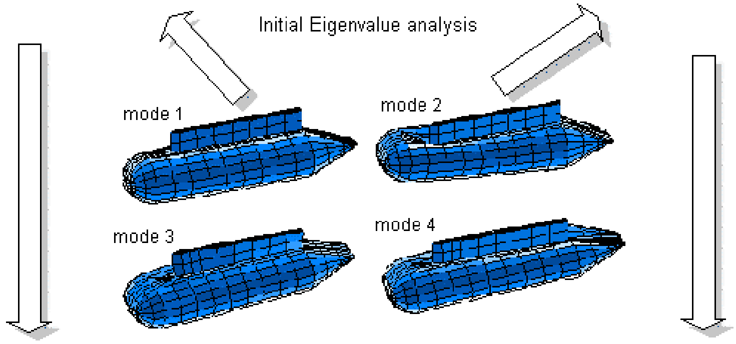
Results across a frequency range, or time history

Response at a node calculated across a specified frequency range



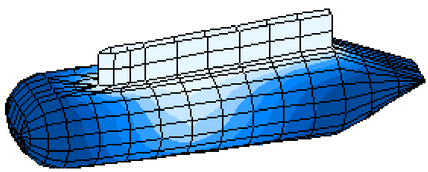
Response at a node calculated across a specified time history



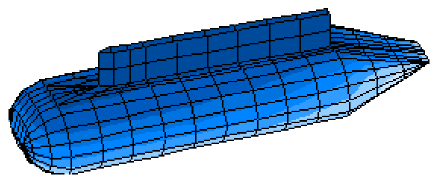


Results at a specific frequency, or time step

Response calculated at a specified frequency for the whole structure



Response calculated at a specified time for the whole structure



Frequency Power Spectral Density

A Power Spectral Density (PSD) defines the frequency content of a random loading, such as turbulent pressure acting on an aircraft component, and are for use in modal random vibration response analysis. At present, random modal vibration calculations are restricted to single-input systems, where the loading at all points is fully correlated.

The value of PSD used in response calculations will be interpolated from a table of frequency/PSD values. A range of linear and logarithmic interpolation schemes are provided, in accordance with typical PSD specifications. Dialog input requires: Interpolation type, options are: Linear/Linear, Log/Linear, Linear/Log, Log/Log; Frequency; Amplitude. More copious PSD tables may more easily be defined by copying and pasting the data from a text file or spreadsheet.

Response Spectrum

Response spectrums for use in spectral analyses are defined from the IMD loadcase dialog.

The value of the frequency used in spectral calculations will be interpolated from a table of frequency-amplitude or period-amplitude values. More copious Response Spectrum tables may more easily be defined by copying and pasting the data from a text file or spreadsheet.

- ❑ **Frequency/Period/Displacement/Velocity/Acceleration** The value of the frequency used in spectral calculations is interpolated from the values defined. The type of values entered, i.e. displacement, velocity, or acceleration should match the type of support motion which is used to excite the structure. For earthquake analyses it is usual to specify Acceleration, and to later specify a support motion using acceleration.

- **Spectral curve damping** This value defines the percentage damping inherent in the response spectrum curve itself. If the Eurocode or Kapra damping correction formula are specified, the spectral response curve is adjusted to the viscous modal damping value specified in the IMD loadcase. When using other damping correction formulae the spectral curve is adjusted using only the viscous damping. For more details see *Theory Manual*.

Case Study. Forced Vibration of a Simply Supported Cantilever

Consider the forced vibration of a simply supported cantilever beam. The beam is supported at one end and subjected to a uniformly distributed load. An eigenvalue analysis is carried out for 9 modes. Normalisation with respect to Global mass must be selected when defining the eigen control.

Calculation of the modal results in the frequency domain is required for **Displacement** type response for the end node for the **Y-direction** displacement. Results are to be calculated over a suitable frequency range using a specified frequency step. The results type required will be **Amplitude**.

Modal response calculations are carried out as follows:

1. Read in the results file from the eigenvalue analysis. Use **File> Open** and specify the results file name.
2. Click on **Utilities> Graph Wizard**. Choose **Modal Expansion** then click on the **Next** button.
3. Click on **Frequency** to specify the frequency domain.
4. Choose a **Point** force excitation, then click on the **Set** button to define the parameters. Excite the structure at the unsupported end node in the Y-direction. Enter the node number or, if the node is selected, choose the number from the drop-down list. Click on the **OK** button, then click on the **Next** button.
5. Specify the results entity as **Displacement**, component **DY** and the results type as **Amplitude**. Take care to specify a realistic start, end and frequency step. Specify which node to calculate the response at, then click on the **Next** button.
6. Specify a Title and X and Y axis labels, then click on the **Finish** button. This will create a standard frequency vs. amplitude plot for the response from the specified excitation. Two graph datasets will be created, the first containing the frequency range in the steps specified and the second containing the required amplitude values.

Target Values

The target values facility is a post-processing facility that provides a general method of varying load factors in a linear analysis to try and achieve target values defined for particular feature types or results components. Target values can be set by use of the **Analyses > Target Values** menu item. By using the tabbed dialog presented, a solution method is selected, and model or results loadcases specified to try and achieve any target values defined. An 'exact' method, an optimisation facility and two best-fit solution methods are provided. The target values loadset created is similar to a combination, but the load factors are calculated automatically by the facility in response to the defined targets. Double-clicking a target values loadset in the Analyses Treeview (or selecting Edit from its context menu) will show the settings used to create it.

Loadcases

In the calculation of target values each included loadcase can be marked as having a **Constant factor** meaning that the loadcase is used by the optimisation algorithm with a constant load factor, or marked as having a **Calculated factor** meaning that the loadcase will be factored by the optimisation algorithm to try to achieve the defined target values.

For a loadcase marked as having a calculated factor this must be set as either **Positive**, or **Allow negative**. For loadcases marked as having a constant factor this choice is not available.

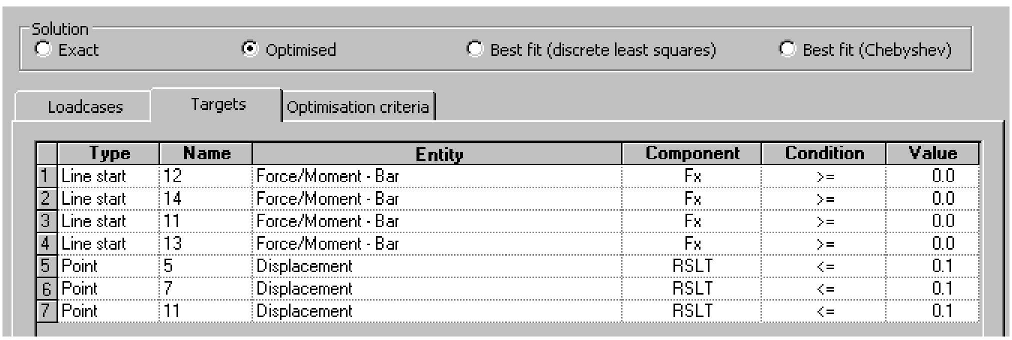
Loadcases marked as having a calculated factor may have an **Importance level** assigned to them (a weighting) but only in an optimised solution. A default value of 1.0 is applied for each if an importance level is not specified.

Cable tuning analysis

Cable tuning analysis is possible using this general target values method but is simplified by using the **Cable tuning analysis** facility, which automatically assigns an initial axial stress of unity to all lines representing cables and automates the definition and assignment of the various loadcases required.

Solution types

The choice of solution type can be made on any of the target values dialog pages.



The type of solution that can be chosen is dependent upon the number of lines selected and the number of equality conditions as seen in the Condition column on the Targets grid. The solution type can be set to be:

- **Exact**. This requires the number of equality conditions (as seen in the Condition column on the Targets grid) to be the same as the number of lines (representing cables) present in the Included panel on the Cables page. No inequality conditions can be defined for the exact method. Using this option, only one solution is possible.

Note that the exact solution for a given set of target values may produce negative factors for some loading arrangements. If this occurs an optimised solution should be investigated where all of the points of interest are permitted to move a small distance to try and achieve a better solution. In the case of Points being used to restrict displacement in a bridge deck this would amount to specifying a distance that would slacken the cables.

- **Optimised**(an under-determined solution) requires the number of equality conditions (as seen in the Condition column on the Targets grid) to be less than the number of lines (representing cables) present in the Included panel on the Cables page that are marked as having a Calculated factor. Any number of targets specifying inequality conditions can be defined. For this option a range of solutions is possible provided that they are not mutually exclusive. For this option a range of solutions is possible.

Options to minimise or maximise the calculated factors for a variety of in-built or specified criteria are provided on the optimisation criteria page.

- **Best fit (discrete least squares)** and **Best fit (Chebyshev)**(both over-determined solutions) require the number of equality conditions (as seen in the Condition column on the Targets grid) to be greater than or equal to the number of loadcases in the Included panel on the Cables page that have a Calculated factor stated for them. Any number of targets specifying equality or inequality conditions can be defined. For this approach no unique solution is guaranteed.
    - **Discrete least squares** provides positive and negative load factors. No inequality conditions can be stated for this option.
    - **Chebyshev** will always produce positive load factors. Inequality conditions can be stated for this option.

Related pages

The target values dialog pages **Loadcases**, **Targets** and **Optimisation** criteria should be visited to specify all data required for a target values calculation.

Validating target values parameters

- **Validate input** The Validate Input button should be used to check if the number of equality conditions and the number of loadcases marked as having a calculated factor are valid for the chosen Solution method. Three different outcomes exist.

    - Input is valid - the analysis can proceed.

- An error message regarding an imbalance in the equality conditions and the number of loadcases marked as having a calculated factor will be displayed, requiring correction.
- The message "Exact solution type required" will appear indicating that, because the number of equality conditions and number of lines representing cables is the same, an optimised or best fit solution cannot be obtained. An Exact solution should be used instead.

Identifying a feature in the grid

- **Identify selection** locates a selected feature (a point or a line) in a model. When the row for a feature is selected in the grid on the dialog, and this checkbox is ‘on’, the corresponding feature (point/line) is located in the View window by animating concentric shrinking squares.

Saving target values parameters

When the **OK** or **Apply** button is pressed all input is saved even if it contains errors/inconsistencies. If the input was valid a Target values entry will be added to a Post processing folder in the Analyses Treeview. A Target values entry will also be added. If the input is invalid the invalid data sign will appear instead and corrective measures will be needed.

Solving a target values loadset

To solve a target values loadset press the Solve button on the main toolbar menu.

Target values results options

When a model containing target value data has been solved the context menu for each target values loadset will contain the following menu items:

- **Calculated Factors** - displays the load factors calculated to achieve the target values. All entries in the grid are read-only.
- **Create Combination** - creates a load combination using the load factors as calculated by the target values analysis. The menu item is greyed out unless the loadset has been solved.
- **Create Loadcase** - creates a loadcase (that could be used in a nonlinear analysis) of all the separate variable loadcases with each factored by the load factor that was calculated by the target values analysis. The menu item is greyed out unless the loadset has been solved.
- **Factor loadcases** - similar to Create Combination but modifies the original input loadcases via their assignment factors. The menu item is greyed out unless the loadset has been solved.

Wood Armer Reinforcement

The Wood Armer facility allows reinforced concrete slabs to be designed to resist a combination of moments [Mx, My] and a twisting moment [Mxy] using orthogonal (or skew) reinforcement. Following a linear elastic analysis the Wood Armer facility determines the design moments [Mx(T), My(T), Mx(B), My(B)]. The procedure was originally developed for a moment field obtained from a plate analysis but may also be applied to shell or grillage analyses.

For a slab modelled as a shell, subject to a moment field [Mx, My, Mxy] and a stress field [Nx, Ny, Nxy] the Wood Armer principles are extended (using the Clark Neilsen calculation). The final output [Nx(T), Nx(B), Ny(T), Ny(B), Fc(T) and Fc(B)] incorporates both bending and in-plane load effects. In order to obtain equivalent in-plane forces from the applied moments, it is necessary to establish the distances between the centroid of the reinforcement layers and the middle surface of the slab. This is calculated using the thickness of the shell elements entered in the geometric properties and the distance to the reinforcement centroid from the face of the slab, entered in the Wood Armer dialog.

An approximate approach is also available for grillage elements whereby bending and twisting moments are converted into ‘equivalent’ plate moments so that the Wood-Armer equations can then be used. An extra geometric property, 'effective width', has to be defined for the grillage to compute these equivalent moments. The effective width is defined in the **grillage geometric attribute**.

Wood Armer properties are specified on the Wood Armer dialog that is activated when a Wood Armer component is selected from the contour, values or vector layer properties or from the print results wizard. The properties specified in the Wood Armer dialog are applicable to all layers.

Wood Armer Assessment

The Wood Armer calculation is generally based on a rationalised set of equations intended to enable efficient design. However, it is sometimes necessary to consider arrangements of reinforcement which have design strengths based on a different rationale. A typical application is the assessment of an existing structure.

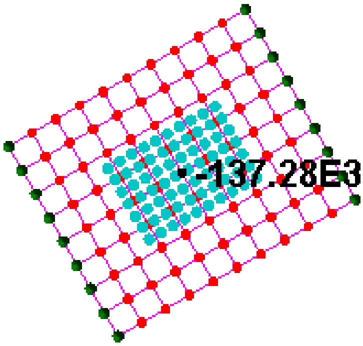
If the capacity of the section in the direction of the reinforcement is known the 'utilisation factor' based on optimal use of the available capacity can be computed. A utilisation factor greater than 1 signifies the slab is under-reinforced and extra reinforcement is required.

Alternatively, the Wood Armer 'K factor' may be entered to proportion the applied twisting moment between the reinforcement directions. This allows any spare capacity that exists in either direction to be utilised.

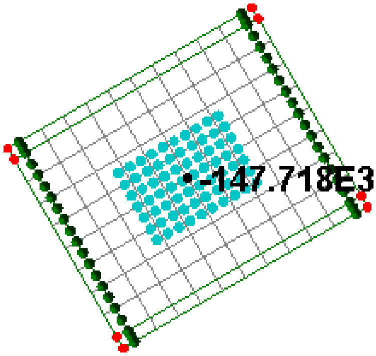
Notes

- Grillage Wood-Armer results are only valid for small skew angles.
- Wood Armer skew angles are measured anticlockwise from the x-axis towards the y-axis.

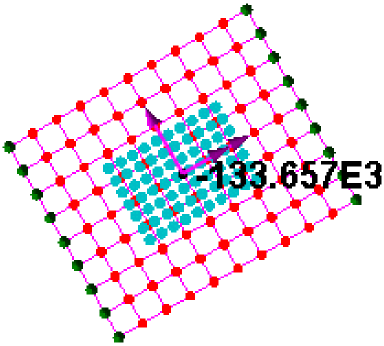
- Wood Armer results for grillage models are only valid if they are relative to the grillage member directions.
- For grillage models where members are not in either the global X or Y directions, Wood Armer components (given in global directions) must be **transformed** such that they are given relative to the local grillage member directions. An angle between the global axes and the local axes in the XY plane can be set.



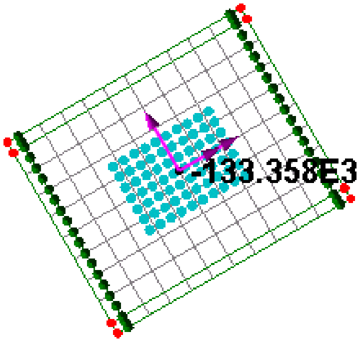
Wood Armer moment (MXB) in global X axis for a grillage model



Wood Armer moment (MXB in global X axis) for the equivalent plate model



Wood Armer moment (MXB) in transformed angle direction for a grillage model



Wood Armer moment (MXB) in transformed angle direction for the equivalent plate model

Comparison of untransformed and transformed values for grillage and plate models

See the *Theory Manual* for further information.

Crack Width Calculation Methods

Crack width calculations and plotting of crack width contours and values can be carried out:

- For plane strain and solid models, when using the Smoothed Multi Crack Concrete Model (Model 102). See **LUSAS Multi Crack Concrete Models.**
- For slabs, for design codes that support crack width calculation using the RC Slab Designer. For more details see **RC Slab Designer: Overview.**
- In accordance with EN 1992-1-1:2004 Eurocode 2 when using the Crack Widths calculation utility with the Smoothed Multi Crack Concrete Model (Model 102). See **Crack Width Calculation to EN 1992-1-1.**

Fourier Results

Before the results for the Fourier can be established, it is necessary to define a combination in which the Fourier harmonics are combined to provide an overall result. The combination is created using **Utilities> Load Combination> Basic…** menu item.

Once the combination has been generated for all the harmonics required and the analysis run to create a results file, the combination loadcase should be set active.

There are then two ways to view results from a Fourier analysis:

- By manually specifying the angle around the circumference at which the results are required and using contours, values and vectors to view the results at that position in the model.
- By using the graph wizard to display the variation of results around the circumference for a specified node.

Design Factors

Design factors provide a means of assessing the reserve strength capacity of a component or structure. To use this facility the results file must be loaded on top of the model file. The material strength and design factor are defined from the **Attributes> Design Strength** menu item and assigned to the model. The reserve strength can then be visualised over the entire model using the **Utilities> Design Factors** menu item. The following criteria are available:

- Maximum Stress Theory (Rankine)
- Maximum Shear Theory (Coulomb, Tresca)
- Maximum Strain Energy Theory (Beltrami)
- Maximum Distortion Energy Theory (Huber, von Mises, Hencky)
- Maximum Strain Theory (St Venant)

Results from each of the above criteria can be displayed as:

Design Factor	Calculation	Failure
Failure/Yield Index	actual/allowable	>1
Factored Failure/Yield Index	(DF*actual)/allowable	>1
Factor of Safety	allowable/actual	<DF
Reserve Factor (RF)	allowable/(DF*actual)	<1
Margin of Safety	RF-1	<0

where:

DF = Design factor defined with material strength

allowable = allowable stress

actual = actual stress

See the *Theory Manual* for further information.

Composite Layers

When viewing composite results it is often useful to change the results orientation to material directions to view the results in fibre and off fibre directions. See **Local and Global Results** for more details. A layer of a solid composite element is treated like a shell. Thus top, middle or bottom stresses can be obtained.

Composite Failure Criteria

Composite failure criteria is defined as an attribute from the **Composite> Composite Failure** menu item. It provides a means of assessing the reserve strength capacity of composite components without carrying out a full nonlinear analysis. To use this facility the results file must be loaded on top of the model file.

- Longitudinal and transverse tensile and compressive strengths must be defined along with a shear strength (which cannot be zero)
- Interaction type can be set to use **Default**, **User**, or **Cowin** values

Once created the Composite Failure attribute should be assigned to the model geometry so that the reserve strength can be visualised over the entire model. Once assigned to the model the following stress components are available for display via selections made on the Contours properties dialog:

- **Tsai Hill**
- **Hoffman**
- **Tsai Wu**
- **Hashin Fibre**
- **Hashin Matrix**

Results may be printed or displayed using standard results processing facilities. Note that a failure criteria greater than 1 indicates failure. See the *Theory Manual* for further information.

Composite Failure Contours

When using the Hashin failure material model the failure indicator (IFFLR) can be contoured. The indicator has the following values:

Indicator (IFFLR)	Description
0-1	No failure
1-2	Matrix failure
2-3	Fibre failure
3+	Matrix and fibre failure

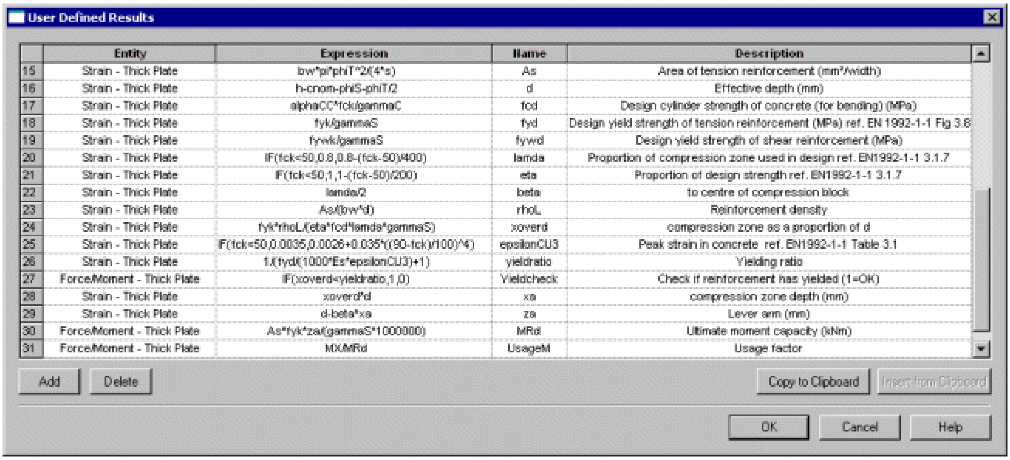
Results may be printed or displayed using standard results processing facilities.

User Defined Results

Defining expressions

The User Defined Results dialog is accessed from the **Utilities> User Defined Results** menu item. It allows results components to be defined by creating arithmetic expressions based upon LUSAS results entities, components, model data and other user-defined results component calculations. A component name and description can also be entered.

User defined results expressions are normally created after a solve has been done and with results loaded, but they can also be defined as part of the modelling process after all relevant features have been assigned an element mesh.



	Entity	Expression	Name	Description
15	Strain - Thick Plate	bw*pi*phiT^2/(4*s)	As	Area of tension reinforcement (mm²/width)
16	Strain - Thick Plate	h-cnom-phiS-phiT/2	d	Effective depth (mm)
17	Strain - Thick Plate	alphaCC*fck/gammaC	fcd	Design cylinder strength of concrete (for bending) (MPa)
18	Strain - Thick Plate	fyk/gammaS	fyd	Design yield strength of tension reinforcement (MPa) ref. EN 1992-1-1 Fig 3.8
19	Strain - Thick Plate	fywk/gammaS	fywd	Design yield strength of shear reinforcement (MPa)
20	Strain - Thick Plate	IF(fck<50,0.8,0.8-(fck-50)/400)	lamda	Proportion of compression zone used in design ref. EN1992-1-1 3.1.7
21	Strain - Thick Plate	IF(fck<50,1,1-(fck-50)/200)	eta	Proportion of design strength ref. EN1992-1-1 3.1.7
22	Strain - Thick Plate	lamda/2	beta	to centre of compression block
23	Strain - Thick Plate	As/(bw*d)	rhoL	Reinforcement density
24	Strain - Thick Plate	fyk*rhoL/(eta*fcd*lamda*gammaS)	xoverd	compression zone as a proportion of d
25	Strain - Thick Plate	IF(fck<50,0.0035,0.0026+0.035*((90-fck)/100)^4)	epsilonCU3	Peak strain in concrete ref. EN1992-1-1 Table 3.1
26	Strain - Thick Plate	1/(fyd/(1000*Es*epsilonCU3)+1)	yieldratio	Yielding ratio
27	Force/Moment - Thick Plate	IF(xoverd<yieldratio,1,0)	Yieldcheck	Check if reinforcement has yielded (1=OK)
28	Strain - Thick Plate	xoverd*d	xa	compression zone depth (mm)
29	Strain - Thick Plate	d-beta*xa	za	Lever arm (mm)
30	Force/Moment - Thick Plate	As*fyk*za/(gammaS*1000000)	MRd	Ultimate moment capacity (kNm)
31	Force/Moment - Thick Plate	MX/MRd	UsageM	Usage factor

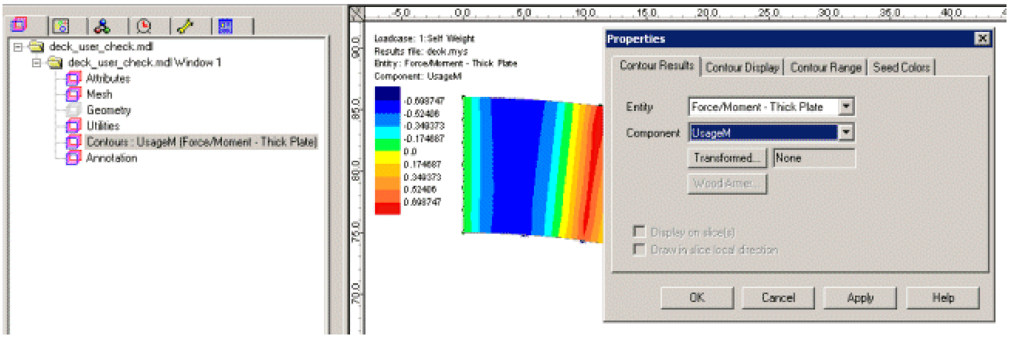
- Clicking on the drop-list button  in the Entity cell lists all entities available for the particular results file loaded.
- Clicking on the launch dialog button  in the Expression cell displays a dialog populated with valid variables for the selection made in the Entity field. By use of

these variables and usual arithmetic syntax a user defined results expression can be built and assigned a name and a description.

For details of expressions and functions supported see **Input and Output of Real Numbers in LUSAS** in Appendix E.

Using named expressions

After definition, the user defined results components created can be selected by name from the Component drop-down list on the Contours, Values, and Diagrams layers properties dialogs. All standard LUSAS results processing, viewing, animating, graphing, printing and report capabilities can be used with any user-defined results components.



Visualising The Results

Results visualisation is performed using results **layers** in the Layers Treeview. Each method of viewing results, described below, requires the relevant layer to be present in the Layers Treeview for the active current window. The properties of the results layers may then be set to display the required results in the specified style.

- **Deformed Mesh** Draws the deformed mesh shape of a structure when subjected to applied load or due to mode of vibration. Note that when influences are being examined the Deformed Mesh layer is named Influence shape.
- **Contour** Display the results on the model as colour fringes or lines of equal value.
- **Vector** Displays vector results quantities as an arrow (or pair of arrows) on the model.
- **Values** Marks result values on the model as symbols and/or values.
- **Diagrams** Displays beam element shear force and bending moment diagrams.

In addition to those layers listed above, an Annotation layer is used to display contour key and other annotation data that may be added to the results viewing window.

Results Orientation

When viewing results in Modeller it is important to note that:

- **By default, displacements, loads, reactions, residuals, velocities and accelerations are output relative to the global Cartesian axis system.**
- **By default, stress/strain and creep/plastic strain results (except for beam/bar elements and interface elements) are output relative to the global axes. Forces and moments (stress resultants) for beams, shells and joints are output relative to the element local axes.**
- **For shell and plate elements, and in accordance with established theory, moments are given along the chosen axis. For beams results moments are given about the chosen axis.**

If the elements in a model are orientated such that their local axes vary from one another (and note for surfaces they may also vary from the surface orientation), or if an alternative coordinate system is required, the results may need to be transformed to a consistent direction. See **Results Transformation**.

Active loadcases

All results visualising uses the results from the **active loadcase** for the current window. The active loadcase is shown with a coloured icon in the Analyses Treeview, and may be changed using the Set Active menu item from the loadcase context menu. Non-active loadcases are shown with a greyed-out icon. An analysis that contains an active loadcase is identified by the addition of a black dot to its analysis icon.

Depending upon the elements used it may be necessary to transform the results to make them consistent. See **Results Transformation** for details.

Inserting a Results Visualisation Layer into the Current Window

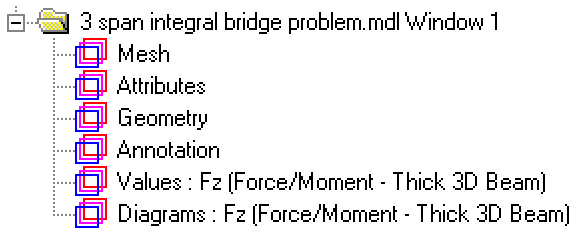
Layers can be added (or removed) from the current window using the **View> Drawing Layers** menu item. A tick is displayed on the menu next to each layer being displayed in the current window. Alternatively, with nothing selected, right-click in the model view window to display a context menu and choose the appropriate layer. Another alternative is to select a layer name from the context menu of the Window name in the Treeview.

The display of layers in the current window can be turned on or off by right-clicking on the layer name in the Treeview and selecting / deselecting the **On/Off** option. See **Using Layers** for more information

Tips

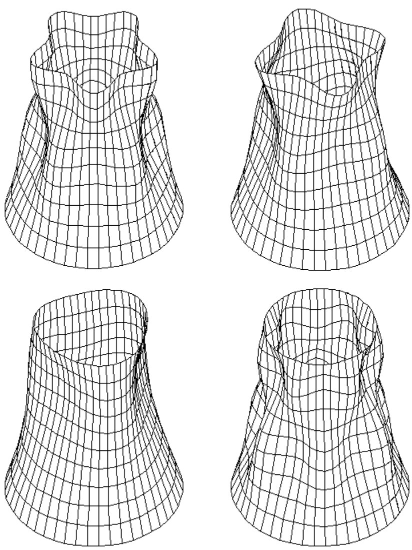
- Use the **annotation** tools to label the display. The annotation toolbar may be displayed using the **View> Toolbars** menu item.
- When comparing different loadcases for the same results type using multiple windows, use a global scale (and a global contour range) so that scaling and contouring in each window is relative to the first window.
- For animations, fix the scale instead of using an auto calculated scale.

Layer names for results layers



Contours, Diagrams, Vectors and Values results layer names as added to and seen in the Layers Treeview have the component (such as Mx, My or Fz, for example), appended to the layer name followed by the entity type such as (Force/Moment - Thick 3D Beam). If different component and entities are selected at a later time the layer name in the Layers Treeview is updated to reflect the chosen selection.

Deformed Mesh Plots



The deformed mesh may be displayed at any time for a single loadcase. For a structural analysis this is the shape under load, whereas for an eigenvalue analysis this is the shape corresponding to the selected eigen mode.

- **Mesh Scaling** is specified as either a deformation factor or a deformation magnitude. The deformation magnitude specifies the maximum deformation to be displayed on the page in millimetres.
- **Mesh Style** The deformed mesh style may be altered as required using wireframe, solid colour, hidden mesh, and element effects.

Drawing the mesh and the deformed mesh together but in different pens and with a suitable scale factor for the deformed mesh is useful for visualising and clarify the deformations.

Notes

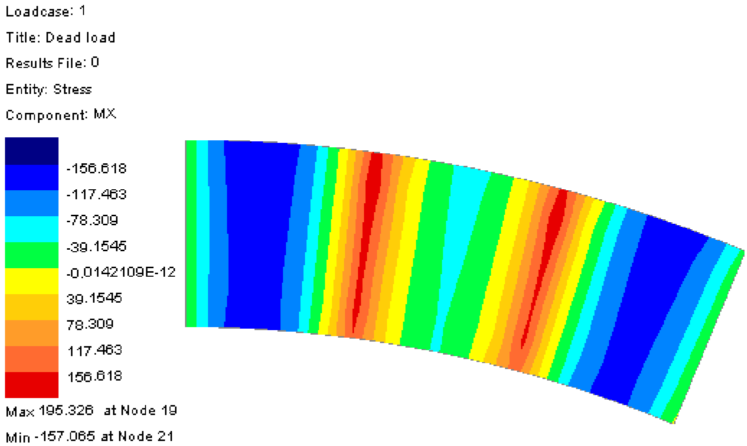
- Other results visualisations may be displayed on the undeformed or deformed shape as required.
- When carrying out contact analysis a unit deformation factor should be used to avoid misleading results.
- When an analysis involves the activation and deactivation of elements, inactive elements may be shown using the **Show activated only** option on the View tab of the View properties dialog.

Contours

Contours display the results of the **active loadcase** on the model as colour fringes or lines of equal results value.

Contour Display Features

- ❑ Contours may be plotted using colour fill and/or contour lines. Fill and line contours displayed together are useful for emphasising the contour levels. Contour labels are available if required.
- ❑ Contours may be plotted on the deformed (loaded) shape or the undeformed shape.
- ❑ Contours may be plotted using averaged nodal results (smoothed) to give a smoothed plot, or use unaveraged (unsmoothed) nodal results to contour the results on an element-by-element basis, revealing any inter-element discontinuities. This is useful for checking mesh discretisation error and for displaying results across geometry or material discontinuities.
- ❑ Contours can be plotted on shells, solids, bar and beam elements, on fleshed members, on layers of composite elements, on slidelines, thermal surfaces, slices and influence shapes (for the Direct Method Influence only)
- ❑ The maximum and/or minimum results value and node location can be annotated.
- ❑ The appearance of the contour key can be adjusted to specify the number of significant figures, draw an outline around each colour in the key and draw red or blue uppermost.



Setting the Contour Levels

By default the levels at which contours are plotted are calculated automatically but they can also be set explicitly. A contour level corresponds to the boundary between adjacent colour fringes, or the line of equal value.

- ❑ Setting the range automatically can be done by specifying either the number of contours or the interval value between contours.
- ❑ Automatic contours levels can be set to pass through a certain value. The maximum and minimum contours may also be specified.
- ❑ A global range can be used to fix the contour levels between different contour windows. It is useful to fix the contour levels manually when comparing results from different loadcases. In both these cases the loadcase containing the maximum value of interest would normally be used to set the contour range.
- ❑ For animations use a manual range to ensure all frames use the same range

Tip. Contours use the colour map to define the colours used. It is often useful to adjust the colour map so that low stress contours are set to white. This makes contour plots easier to understand, and also avoids excessive use of a particular coloured ink when printing.

See **LUSAS Multi Crack Concrete Models (Model 94 and Model 102)** for details of plotting contours and values of crack widths.

Vectors

Vectors are used to visualise both the **magnitude** and **direction** of specified results components. Vectors may be displayed at either nodes or Gauss points.

Scale

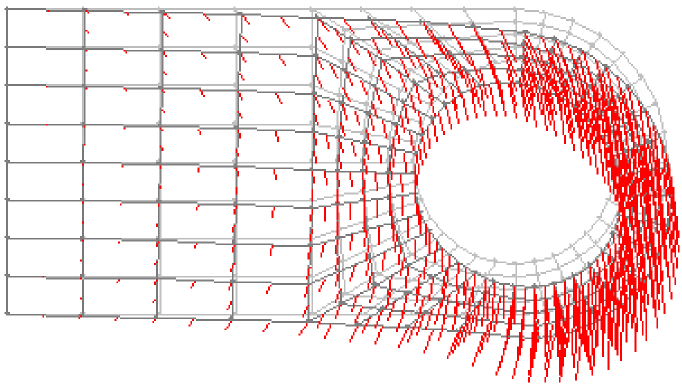
By default, vectors will be scaled such that a vector length of 6mm represents the maximum displayed results component. Alternatively, a scale factor can be specified.

Style

Vectors may be drawn as lines or arrows. By default the colours used for drawing vectors in tension and compression are red and blue but the pens can be altered as required.

Deformed Shape

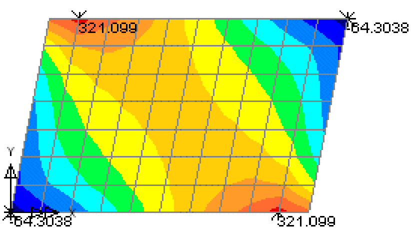
This image shows displacement vectors displayed on the deformed shape.



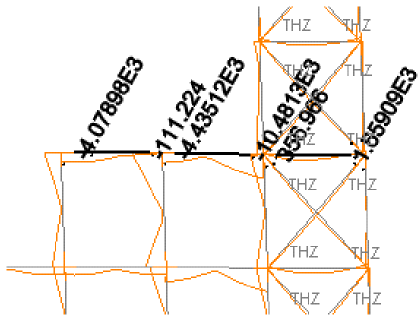
Values

Values are used to identify the location and value of results for both averaged nodal (smoothed), element nodal (unaveraged) and Gauss point values. Either maximum and/or minimum values may be visualised, and a percentage may be specified to determine whether values lying within a maximum/minimum range are displayed. Values may also be displayed at selected nodes.

The Values layer is also used to display the locations of yield symbols and of concrete cracking planes and symbols denoting regions of localised crushing. See **Nonlinear Material Plots** for details.

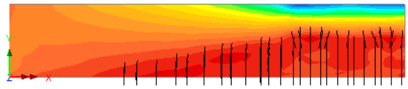


Maximum and minimum values marked



Values plotted for selected (shrunk) elements

Visualisation of yielded material



Visualisation of concrete crack patterns on a 2D model

Tips:

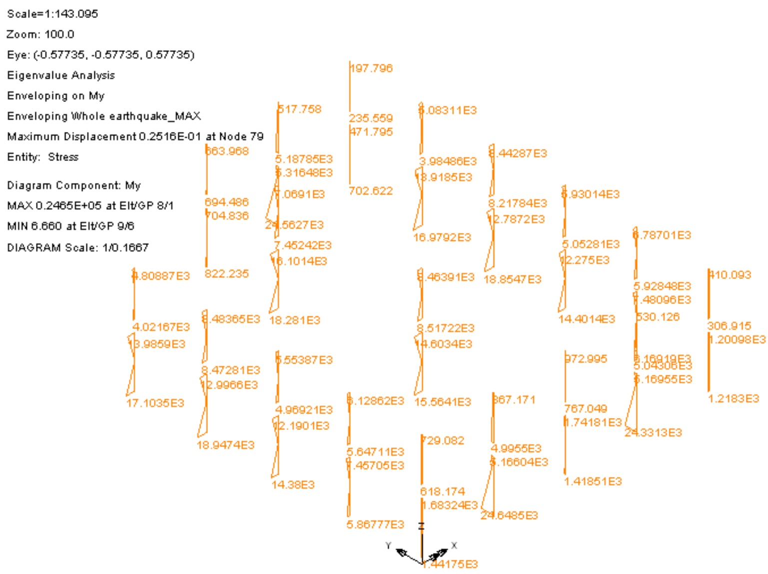
- The percentage range sets the range of values to display, starting from the maximum and/or maximum values. Setting this value to 100% displays all values, this is useful for displaying nodal results on the screen for a subset of a model. Setting this value to 0% shows only the extreme maximum and/or minimum setting.
- To prevent text labels from overlapping each other the elements on which that are drawn may be shrunk by specifying a percentage score of elements length remaining on the Mesh layer properties dialog. Additionally text may be given a rotation angle on the Values layer properties dialog to prevent text labels from overlapping each other.
- Use the **Show values of selection** option to isolate only those values of interest, or to restrict labelling on complex models to selected nodes and elements only.
- Gauss point values display the computed values from the analysis. These values may be particularly useful when examining results for nonlinear materials.
- Unaveraged values display the computed values after they have been extrapolated to the element nodes but before they have been averaged. These values may be of particular interest when examining results around discontinuities in geometry of materials.

Diagrams

Bending moment and force diagrams may be drawn for any 2 or 3 dimensional frame structure comprised of **bar** or **beam** elements. All of the results for diagrams are located within the Stress results entity, and the results components available will depend upon the element type (see *Element Reference Manual*). The diagrams may be drawn using the element axes or screen axes.

The following quantities may be represented:

- ❑ **Axial Forces** local x direction beam forces.
- ❑ **Shear Forces** through-thickness shear forces.
- ❑ **Bending Moments** bending moment results.
- ❑ **Torsional Moments**



Plotting Results for Groups

By default results are computed and displayed on the visible model. When appropriate layers are present in Layers Treeview, results can be selectively plotted for groups held in the Treeview by choosing the following Results Plots context menu items for each named group:

- **Show Results**
- **Do Not Show Results**
- **Show Results Only On This Group / Attribute**

When combined with group visibility options that can also be accessed via the context menu for each group name parts of the model can be isolated and have results plotted just for those regions.

Pairs of symbols adjacent to each group name in the Treeview show the status of model visibility and results display.

When viewing results:

(green tick) All of the objects in this group are visible, but no results are being shown.

(blue tick) Some of the objects in this group are visible, but no results are being shown.

(red cross) None of the objects in this group are visible and no results are being shown.

(green tick, green border) All of the objects in this group are visible and showing results.

(blue tick, blue border) Some of the objects in this group are visible and some are showing results.

(green tick, blue border) All of the objects in this group are visible but only some are showing results

Notes

- When choosing to show results for a material, geometric property or element mesh type that does not support (for example) the same Contour entity as that used for the previously plotted item, no results will be shown until a valid Entity and Component for the new selection is picked on the Contours property dialog accessed via the Layers Treeview.
- A black dot next to a group symbol denotes the current group into which all new geometry will be added when created and hence has no relevance when viewing results.

Plotting Results for Assigned Attributes

By default results are computed and displayed on the visible model. With appropriate layers present in Layers Treeview, results can be selectively plotted for attributes held in the Attributes Treeview by right-clicking on the attribute name and then choosing Results Plots. The following context menu commands are available for Results Plots:

- **Show Results**
- **Do Not Show Results**
- **Show Results Only On This Attribute**

When combined with visibility options that can also be accessed via the context menu for each attribute selected features of the model can be isolated and have results plotted just for those features. This provides a means of producing isolated results for particular material types, geometry, or element mesh types without having to define individual groups for each of these items. But, if required, groups of features or elements may be defined and the Results Plots entry may be used to display selected results for a chosen group.

Notes

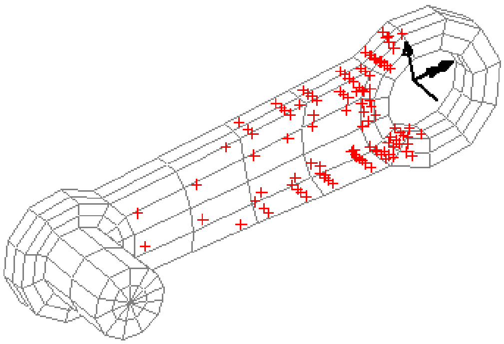
- When choosing to show results for a material, geometric property or element mesh type that does not support (for example) the same Contour entity as that used for the previously plotted item, no results will be shown until a valid Entity and Component for the new attribute selection is picked on the Contours property dialog accessed via the Layers Treeview.

Nonlinear Material Results Display

Two types of nonlinear material results may be displayed. Yield flags and crack/crush patterns. The availability of each depends on the material model assigned to the elements during the modelling stage. Crack and crush patterns are available for concrete models and yield flags are available for all nonlinear materials.

- **Yielded material**

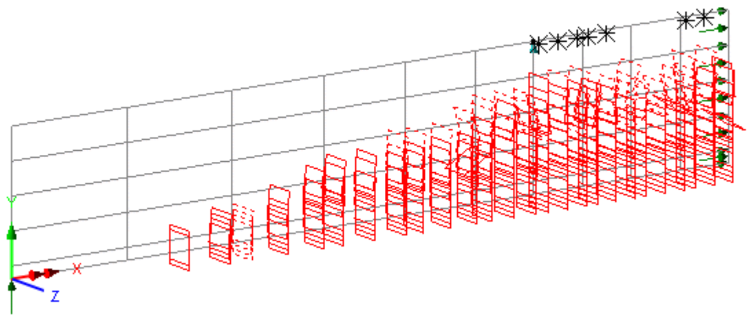
  Available from the **Values layer**, specify the results entity as **Stress**, and the type as **Yield**. Yield flags show the extent of the yielded material within a structure and are plotted at Gauss points. The nonlinear example here demonstrates how the spread of yielded material is visualised using a symbol at element Gauss points.



  This display is especially useful when used in conjunction with contours or an animated sequence when the onset and spread of yield can be highlighted.

- **Crack patterns and Crushing** Crack patterns and crushing symbols can be displayed for models that use the concrete material models 94 and 102. These are visualised using the **Values layer** by specifying the Results Entity as **Stress** and the Type as **Crack/Crush.** Three types of display are possible:

  - Localised crushing is shown by black star symbols
  - For the Multi Crack Concrete model regions of cracking are shown by crack planes drawn in red full lines, and closed cracks are shown by crack planes drawn in red dashed lines.
  - For the Smoothed Multi Crack Concrete Model crack planes are drawn in a common line style and pen colour whether open or closed in nature.



Open and closed crack planes and crush symbols resulting from the loading and unloading of a 2D concrete shell model

Notes

- When a model includes a nonlinear material model the Plastic strain entity is available for all elements (even though some elements are not valid)
- See **LUSAS Multi Crack Concrete Models (Model 94 and Model 102)** for details of plotting contours and values of crack widths.

Results On Sections / Slices Through A Model

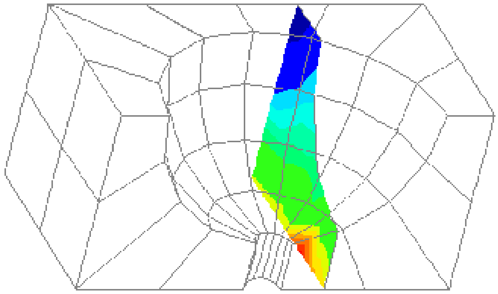
Results can be plotted on sections or slices through a model by choosing the following menu commands available from the **Utilities** menu:

- **Section through 3D** Cross-sections may be taken through a three-dimensional solid model to display results on a slice. Results are calculated at pseudo-nodes formed at the intersections of the slice with the element edges by linear interpolation of the nodal results.
- **Graph through 2D** Graphs of results may be created along a line through a 2D continuum model or on a line through a section from a 3D model. Results are calculated at the intersections of the line section and the element edges hence a finer mesh will produce more sampling points. Force and moment values along the slice may also be computed.

Section Through a 3D Model

Slice sections may be cut at arbitrary positions through the model using the cursor to define either a horizontal or vertical slice in the View window. Slices may be generated in any plane by rotating the model to the desired orientation before a section is cut.

By default the location of an arbitrary slice section through a model is not saved with the model. However, when cutting a slice section through the model an option to create an annotation polygon is provided. This annotation polygon effectively defines the location and orientation of the cutting plane and does get saved with the model. Annotation polygons may be re-selected if a model is reloaded at a later date in order to create a slice at the same location.



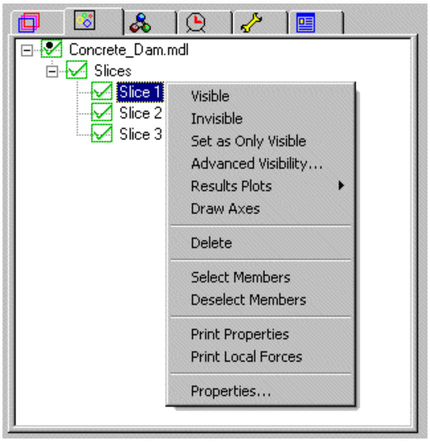
Slice sections can also be created using surfaces that are created at slice section locations. These surfaces do not have to surround a model, they must simply be defined in the orientation of the cutting plane required.

To use a saved cutting plane or a surface defining in cutting plane instead of indicating the cut using the cursor, the annotation polygon or surface should be selected before choosing the **Utilities> Slice through 3D** menu item, and **By selected polygon / surface** option.

Notes

- No results visualisation or printed results for slices are available unless the **Display on slice(s)** option has been selected on the Contours and Values properties dialogs or on the Print Results wizard dialog.
- 3D sectioning/slicing is only available for 3-Dimensional solid models. For surface models refer to **Graph Through 2D** below.
- A slice can form the basis for a line section using **Graph Through 2D** to graph results along a line through the centre of a three dimensional solid.

Manipulating The Slice



Once a 3D slice section has been defined, a slice exists as a **group** in the Treeview, and may be manipulated using the **View> Group** menu item or from the group's context menu. For more details on viewing results on groups see **Plotting Results for Groups**.

Notes

- The mesh and nodes on the slice are displayed using the mesh layer properties.
- The slice local axes and origin are displayed and moved from the slice properties accessed from the slice context menu.
- The resultant local forces on the slice and the slice properties may be printed from the slice context menu.
- Groups created from slice sectioning a 3D model cannot currently be retained when a model is saved.

Graph Through 2D

Arbitrary line sections may be taken through any surface model or on a slice cut through a three dimensional solid model. The process of cutting a slice will generate two graph datasets, the first containing the distance along the line section and the second containing the specified results along the line. The graph datasets are plotted automatically using the **Graph Wizard**. Graph datasets are stored in the Utilities Treeview.

By default line sections may be cut at arbitrary positions through the model using the cursor. Lines can also be defined to start and finish at points located on an underlying grid. When cutting a line section through the model an option to create an annotation line is provided. This may be used later for repeating the cut if a graph along the same line is required. To use an existing line (or annotation line), instead of indicating the cut using the cursor, the line (or annotation line) should be selected before choosing the **Utilities > Graph through 2D**menu item.

Notes

- Care should be taken when slicing through voids, or holes, in the model as this can give misleading results.
- Care should be taken when slices pass through parts of the model with non-uniform properties, such as parts of different materials or of different thickness.

- Sometimes, due to model size, grid points are too close together to be usable. In these cases simply increase the grid size so that individual grid points can be selected.

Graph Wizard

Two types of graphs may be plotted:

- **Results quantities** Any available result entity may be plotted against distance along the slice.
- **Axial force and bending moment** This generates three datasets of distance, axial and bending stress along a line through a section: the axial force per unit width, moment per unit width and distance of the neutral axis to the midpoint of this line. For axisymmetric solids the moment per unit radian is printed as well. From these datasets, graphs of axial and bending stress versus depth of section are plotted.

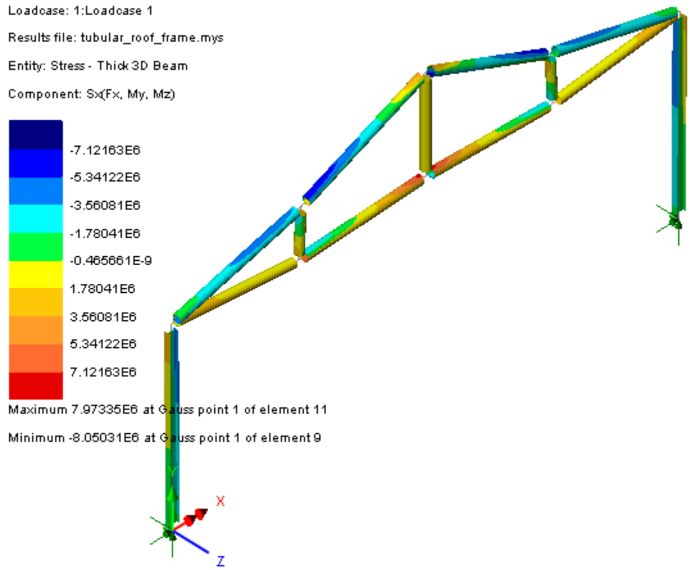
For more details on viewing results on graphs see **Plotting Results on a Graph**.

Displaying Beam Stresses

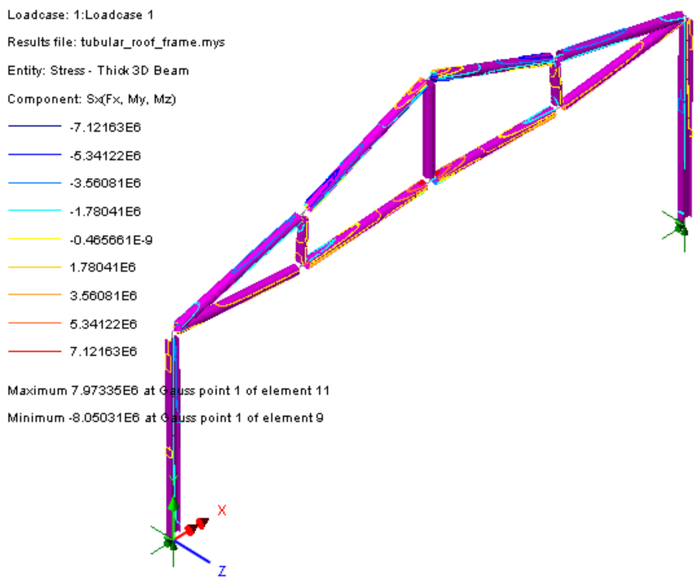
Beam stresses can be displayed on the fleshed section or at selected **fibre locations**. These stresses are computed using engineering beam theory which assumes that the normal stress is constant across the width of the beam cross section. This assumption can introduces significant errors due to shear lag when wide flanged sections are being used so these stresses should be used with caution.

Beam stresses can be displayed as contours on the fleshed section or as values, diagrams or beam contours at fibre locations using the standard layer controls. When viewing stresses at fibre locations the value, diagram or contour is drawn at the actual position of the fibre on the cross section.

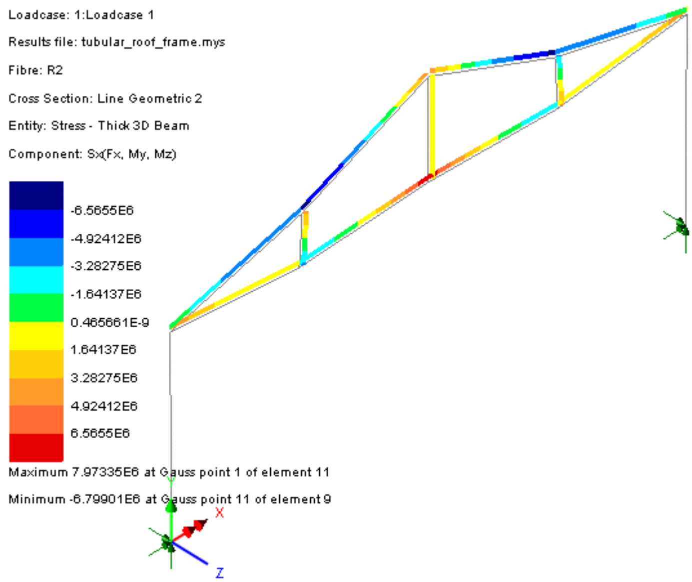
Examples follow:



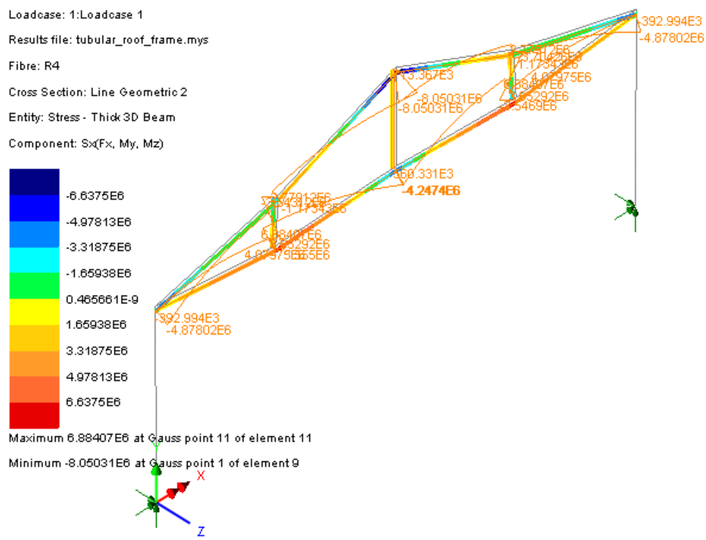
Filled contours plotted on fleshed beam sections



Line contours plotted on fleshed beam sections

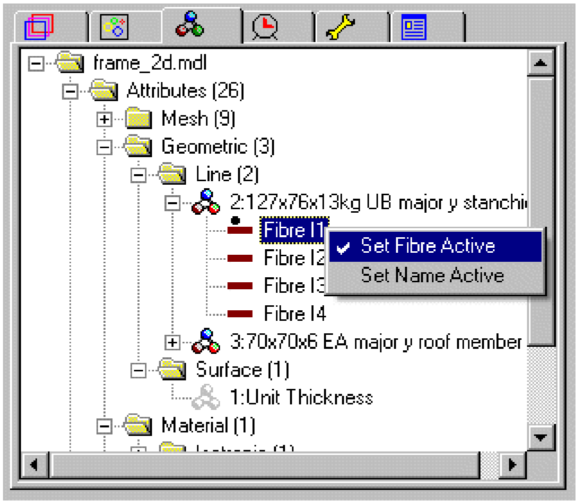


Line contours plotted at top beam fibre location



Line contours and diagram stress results plotted at bottom beam fibre location

Fibre locations



When plotting selected contour or diagram results for beams, a fibre location must be used to specify the place(s) on the beam section at which the results should be calculated and plotted. Results for individual fibres are plotted by setting that fibre location active for a particular geometric line attribute in the Treeview. The active fibre is denoted with a black dot next to the fibre name. R1

- Right-clicking on a fibre name in the Treeview and choosing **Set Fibre Active** will display results for just that fibre.
- Right-clicking on a fibre name in the Treeview and choosing **Set Fibre Name** will display results on all members with that fibre name.

Notes

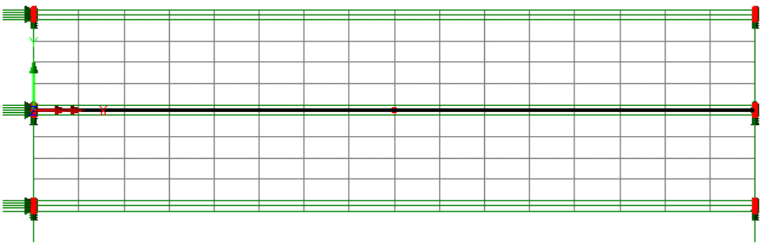
- Standard sections extracted from the section library include extreme fibre locations for all sections.
- The standard section property calculator automatically includes extreme fibre locations for all cross- sections that it supports.
- Models created prior to version 14.2 will not have any fibre locations data stored for each beam. However, the relevant fibre location data can be added automatically by double-clicking on each Geometric line entry in the Treeview and re-selecting the same section size from the appropriate sections library.
- User-defined beam cross-sections require fibre locations to be defined manually in order for stresses to be displayed on the diagrams and values layers.

Beam Stress Resultants From Beams and Shells

Equivalent beam stress resultants can be calculated for flat or curved thin and thick shell models by use of the **Utilities> Slice resultants from beams and shells** menu item. This allows the conversion of the results of a complex shell model into an equivalent beam analogy for use in design codes of practice.

Orientation of model

Prior to defining slice locations, the model should generally be viewed along one of the primary view axes that relate to a plan (usually) or elevation view of the model. Once slice locations have been defined the slice resultants are computed using valid visible 3D beam and shell elements. Invisible elements are ignored.



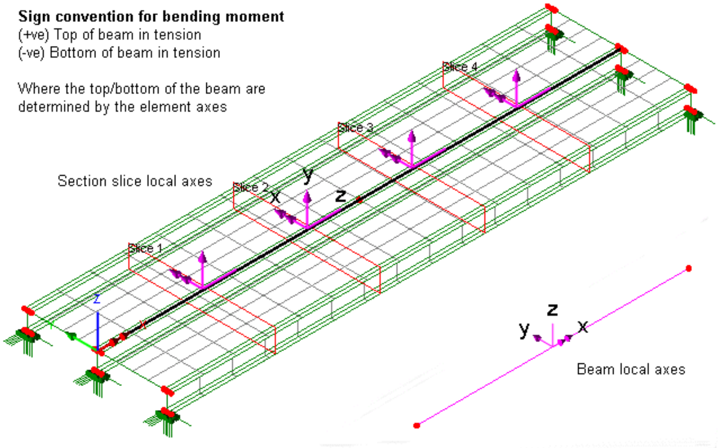
Model viewed along Z axis (plan view) with path selected prior to defining section slice locations

Slice locations

The **slice locations** are defined using a path which can consist of straight lines and arcs or combined lines that contain straight lines and arcs in the selection. The path must be continuous without any branching characteristics. The slice path orientation is defined by either the order of selection when more than one line/arc is selected or the orientation of the line/arc if only a single line/arc has been selected. The locations for the slices along the path can be defined using three methods:

- ❑ **From points or nodes in the selection** If nodes or points are selected, these are projected onto the path perpendicular to the appropriate path segment tangent to obtain the slicing locations
- ❑ **Incremental distances from start of path** Incremental distances can be entered to define the distances along the path for the slicing. Distances can be both positive and negative but the running total distance should remain within the length of the path. For example, 1@0;1@10;1@-5 will cut slices at distances of 0, 10 and 5 along the path
- ❑ **Absolute distances from start of path** Absolute distances can be entered to define the distances along the path for the slicing. Distances must be positive and within the length of the path. For example, 0;5;10 will cut slices at distances of 0, 5 and 10 along the path
- ❑ The **Distance from reference origin to start of path** can be also be entered. If this value is non-zero, the value is added to the distances along the path.

The orientation of the slice local axes is different from the sign convention used in accordance with beam theory where the beam local x axis is orientated along the beam. The orientation of the slice local axes is defined from both the tangent of the path at the location of the slice and the direction that the model is viewed from. If the slicing path is in the plane of the screen the positive slice local z axis will be defined by the tangent of the path and the positive slice local y axis will be orientated in the out of screen sense. If the slicing path is not in the plane of the screen, the positive slice local y axis will be defined perpendicular to the path tangent in the path tangent/out of screen plane. The positive slice local x axis will be defined perpendicular to both the slice local y and z axes.



Isometric view of beam and shell model showing slice locations and slice local axes used. Inset shows beam local axis.

Slice options

- ❑ The calculation of moments can be about either the **Neutral axis** or the **Path intersection** with the slice plane.

Note: For the calculation of beam stress resultants it has to be assumed that plane sections remain plane under the action of the loading. The strain distribution over the whole section also is assumed to remain linear. The location of the neutral axis can therefore be calculated directly from the areas and stiffness of the contributing materials in the composite cross-section (see Gere & Timoshenko, Mechanics of Materials, 3rd SI Ed., pg 301-). For the calculation of the neutral axis location, the Transformed-Section Method is used which incorporates Modular Ratio techniques.

Additional options available to control the slicing include:

- ❑ **Effective width** If the effective width option is selected, the width of visible elements to include in the calculations can be specified. This effective width is centred on the slicing path in the screen plane at a perpendicular to the path. If the effective width is not used, all valid visible elements will be included in this direction. For both options, the slice is infinitely deep in the slice local y axis
- ❑ **Include whole elements only** If the effective width option is selected, the option to include whole elements only is available. If selected, partial elements intersected by the current slice will be ignored in the calculations

- ❑ **Smooth corners on path** If the smooth corners on path option is selected, the average tangent of the path at a connection between two lines/arcs will be used for the slicing if the distance along the slice path exactly matches this path connection. If the smooth corners on path option is not selected, two slices will be taken at the connection using the tangents for both of the lines/arcs connecting at this location
- ❑ **Slicename prefix** Allows user to input a user defined prefix for the slice names

Loadcase

Slice resultant results can be output for one or more loadcases

- ❑ **Active** - prints slice results for the active loadcase to a slice output window.
- ❑ **All** - prints slice results for all loadcases to a text file named SliceResultantsBeamsShells.prn in the current working directory.
- ❑ **Selected** - prints slice results for entered loadcases to a text file named SliceResultantsBeamsShells.prn in the current working directory. As an example, entering 1-5,7 would select loadcases 1 to 7 excluding loadcase 6.

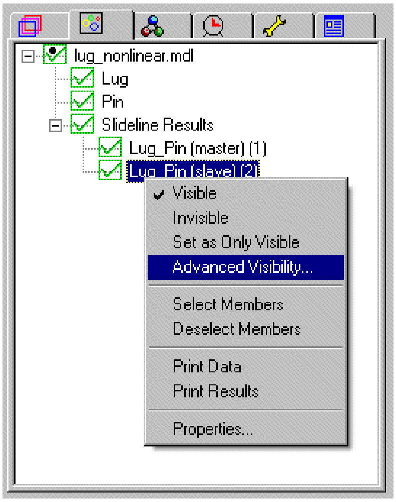
Notes

- The assumption that plane sections remain plane which is required for the calculation.
- Linear variation of stress is assumed for the approach and therefore low order flat shells are supported (3 or 4 noded thin and thick shells) and high order flat shells are calculated ignoring the mid-side nodes if they are coplanar with collinear edges. For curved high order shells the element is subdivided into constituent pseudo elements and each pseudo element interrogated using linear interpolation.
- The slicing path must be defined using straight lines, arcs or combined lines containing only these two line types. Splines and annotation lines cannot be used.
- Slice forces can only be computed relative to the intersection of the slicing path with the slice plane or the neutral axis based on the sliced section. No facilities are available for the transformation of slices without defining a separate path and recalculating the slices.
- Taking slices at the free end of a structure can lead to overestimation of the forces and moments on the section. This occurs due to the stresses in the section not returning to zero at a free and unloaded end.
- Taking slices at a supported end of a structure can lead to discrepancies in the forces and moments when compared to a beam model due to the end effects taken into account by the full 3D shell modelling
- Where a slice passes through a node in the structure, results are presented for the contributions of the elements on both sides of the slice. This is presented as results for the negative and positive local Z sides of the slice

- Engineering thick beam elements are supported (BMS3 and BTS3) along with thin beam (BS3, BS4, BSL3 and BSL4) elements
- Option 380 must be used for BTS3 elements when using eccentricitices/offsets
- When using 3 or 4-noded shell elements the mesh density should be sufficiently fine to capture the behaviour of the structure
- When modelling curved structures it is recommend that regular quadrilateral elements wherever possible
- Printing results for more than one loadcase will overwrite the text file named SliceResultantsBeamsShells.prn each time
- See the worked example 'Section Slicing of 3D Shell Structure' in the *Application Examples Manual (Bridge, Civil and Structural)* for the use of this facility.

Slideline Results Processing

When a results file is read into Modeller from an analysis that involves slidelines, a group is created for each slideline surface in the model. These groups can be accessed from the Group Treeview.



A context menu for each group name provides options for making the members of a group visible or invisible and for selecting and deslecting the members of a group.

Limited results for a selected slideline group can be printed by using the Print Results menu item on the context menu for that group. Full slideline results for the model as a whole can be printed using the Print Results Wizard.

Printing Slideline Results

The print results wizard is accessed from the Utilities menu. When Slideline results are selected the following six types of results can be chosen for printing:

- ❑ **Summary**
- ❑ **System forces**
- ❑ **Gap Forces**
- ❑ **Contact forces**
- ❑ **Contact Stresses**
- ❑ **Section Results**

Each type of result is described below.

Summary

The summary presents a list of the maximum and minimum values of each slideline results in a table.

System, Contact and Gap Forces

There are three categories of force results:

- **System forces** The normal and tangential gap forces at each slideline node are distributed across both slideline surfaces. These forces are assembled together and transformed into the global system directions to give the System Forces.
- **Gap forces** Gap forces are computed from the normal penetration and the tangential movement of a node. They are the basic quantities used in the contact formulation. For example, with the penalty method the normal gap force at a node is obtained by multiplying the normal penetration with the contact stiffness. Coulomb’s law of friction also uses gap forces. To check the application of the law, the normal and tangential gap forces should be compared.
- **Contact forces** The normal and tangential gap forces at each slideline node are distributed across both slideline surfaces. These forces are assembled together and kept in the local directions, normal and tangential to the contact surfaces, to give the Contact Forces. The contact pressures and stresses are based on these forces.

Contact Stresses

This category contains results for the contact pressure normal to the surface and the contact stresses tangential to the surface at each node.

Section Results

This category contains generic contact results. It includes the status of each node as to whether it is in contact or out of contact, the normal penetration for each contacted node, the contact stiffness, the nodal contact area and the zonal contact distance.

Graphing Slideline Results

To graph slideline results, the slideline group of interest must be set visible from the group context menu in the Group Treeview .

For two-dimensional analyses the graph wizard can be used to generate the variation of a particular slideline result along the slideline surface. The variation along the surface can either be graphed against distance or angle.

For three-dimensional analyses the slideline results are graphed in the same manner as other results.

Plotting Contours, Values and Vectors for Slidelines

For three-dimensional analyses, contours of contact results can be displayed by selecting **Slideline Results** as the entity on the **Contours** dialog. The full list of slideline results will then be available in the Component combo.

Slideline values and vectors can be displayed for both two and three-dimensional analyses by selecting **Slideline Results** on the **Values** or **Vectors** dialog. With values the full list of slideline results components is available, whilst with vectors only Contact and System Forces are available.

Note. When looking at the deformed mesh from a contact analysis, the exaggeration factor should be set to 1.0 to avoid a misleading visualisation.

Table of Slideline Results Availability

Slideline Results components	Label	Contours	Values	Vectors	Printing	Graphing
System forces						
Contact force in system x direction	ForceX	Yes	Yes	Yes	Yes	Yes
Contact force in system y direction	ForceY	Yes	Yes	Yes	Yes	Yes
Contact force in system z direction	ForceZ	Yes	Yes	Yes	Yes	Yes
Resultant contact force	RsltForce	Yes	Yes	Yes	Yes	Yes
Gap forces						
Tangential gap force in local x direction	TanGapFrcX	Yes	Yes		Yes	Yes
Tangential gap force in local y direction	TanGapFrcY	Yes	Yes		Yes	Yes
Resultant tangential gap force	RsltTanFrc	Yes	Yes		Yes	Yes
Gap force normal to contact surface	NrmGapForc	Yes	Yes		Yes	Yes
Contact forces						
Tangential contact force in local x direction	TanForcex	Yes	Yes	Yes	Yes	Yes
Tangential contact force in local y direction	TanForcey	Yes	Yes	Yes	Yes	Yes
Resultant Tangential contact force	RsltTanFrc	Yes	Yes	Yes	Yes	Yes
Contact force normal to contact surface	NrmForce	Yes	Yes	Yes	Yes	Yes
Contact stresses						
Contact stress in local x direction	ContStresx	Yes	Yes		Yes	Yes
Contact stress in local y direction	ContStresy	Yes	Yes		Yes	Yes
Contact pressure normal to contact surface	ContPress	Yes	Yes		Yes	Yes
Section Results						
Contact stiffness	ContStiff	Yes	Yes		Yes	Yes
Penetration normal to contact surface	NrmPen	Yes	Yes		Yes	Yes
In-contact/out-of-contact status	ContStatus	Yes	Yes		Yes	Yes

Slideline Results components	Label	Contours	Values	Vectors	Printing	Graphing
Nodal contact area	ContacArea	Yes	Yes		Yes	Yes
Zonal contact parameter	Zone	Yes	Yes		Yes	Yes
Zonal contact detection distance	ZnCnDetDst	Yes	Yes		Yes	Yes
Contact stiffness coefficient	IntStfCoef	Yes	Yes		Yes	Yes

Thermal Surface Results

The results from analyses involving thermal surfaces may be processed in a similar manner as other results. Extra result types are available for thermal surfaces.

When reading a results file a **group** is automatically created for every thermal surface used in the analysis. These are accessed from the Group Treeview .

For 2D slidelines the graph wizard can be used to generate the variation of the thermal surface variables along the surface or as a history though the analysis. When multiple thermal surfaces are present results for specific thermal surface are viewed by setting the appropriate surface as only visible from the group context menu.

For 3D problems contours of the thermal surface results may be displayed on the thermal surface. Values and vectors can be displayed for both 2D and 3D problems. The results on a thermal surface are displayed by selecting the **Thermal Surface Results** entity on the appropriate property dialog.

Thermal surface results may be printed using the print results wizard.

Thermal surface flow results	Label
Gap and environmental flow	GapEnvFlw
Radiation flow between segment	RadFlwSeg
Radiation flow to environment	RadFlwEnv
Total nodal flow	TotalFlw

Plotting Results on a Graph

The Graph Wizard is used to draw XY graphs. The following results graphs may be plotted if the results are available:

- ❑ **Time history** A history for a specified results type throughout an analysis with respect to time or increment. Graphs may be plotted for a named variable such as Total Load Factor or Response Time verses a specified results quantity such as displacement in X, equivalent stress or sum of reaction over a set of specified nodes. The following history datasets are available:

    - **Nodal** Averaged or summed.

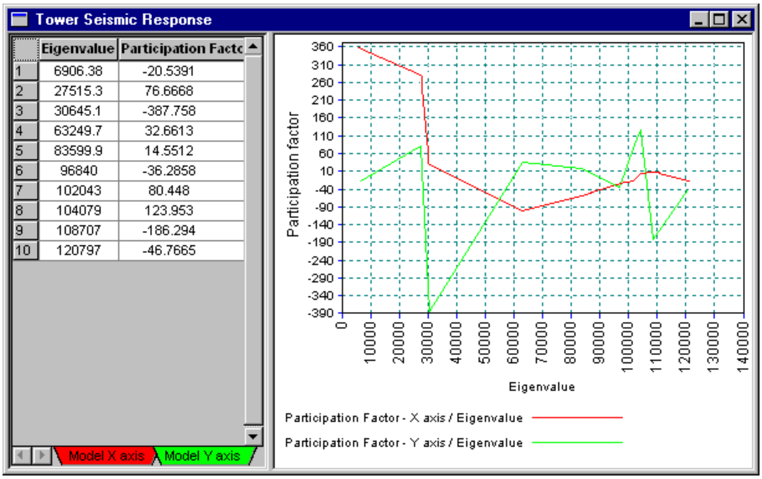
- **Gauss Point** results for a selected element and Gauss point (Element Gauss numbers may be determined using the labels layer).
- **Named** Named result variables for a linear analysis such as loadcase ID, a transient analysis such as response time, or a nonlinear analysis such as total load factor.
- **Element** results for all or selected elements.
- **Strain energy and plastic work** Total strain energy or total plastic energy for the elements showing results.
- **Previously defined**

- ❑ **Fourier expansion** The displacements, stresses and strains output from a Fourier analysis are coefficients of corresponding sine and cosine functions. The evaluation of these functions around the circumference of the model is achieved graphically using the Graph Wizard.
- ❑ **Modal expansion** Graphs the modal response of a structure to dynamic excitation using the results from an eigenvalue analysis. Various results entities may be plotted against a frequency range or sampling time for selected eigenmodes.
- ❑ **Load curve** Graphs a defined load curve.
- ❑ **Variation** Graphs a variation function.
- ❑ **Specified datasets** Graphs two previously defined datasets.
- ❑ **Thermal surfaces** Graphs results along a thermal surface.
- ❑ **Slideline (assigned to line)** Graphs results along a slideline**.**

Note: To graph a specified results entity against distance along a slice through a planar structure or on a slice of a three dimensional solid structures see **Graph Through 2D**.

Graph Properties

LUSAS uses the **Graph Wizard** to take you through each step of creating the X and Y datasets and placing them into a graph. The graph wizard is started from the **Utilities** menu. The X and Y datasets are then stored in the Utilities Treeview.



The graph window is split in to the **graph area** on the right and the **graph data table** on the left.

Tip. Zoom in on a part of the graph by boxing with the mouse. To unzoom right-click on graph area to display the context menu and select **unzoom.**

Plotting Families Of Curve Data On The Same Graph

If a graph already exists, then further curves may be added to the first graph by choosing the **Add to existing graph** option of the final page of the Graph Wizard. In this way families of curves may be drawn on the same graph using a different colour for each one.

Editing Graphs

Once a graph has been plotted, the appearance and even the data on the graph may be modified. This may be done by selecting the **Edit> Graph Properties** menu item from the graph context menu.

Editing Graph Data

To change individual data points on the graph (for example to add an origin to a curve), make the graph data table editable by checking the **Editable graph table** box on the General Graph Properties page of the Graph Properties dialog. The following facilities are then available:

- New data points may be added by adding new rows to the grid either by using **Insert Row** from the **Edit** menu or by pressing the **Tab** key when the cursor is in the final cell.
- Data points may be deleted by deleting the data in the grid.
- Blocks of data from a spreadsheet may be pasted. New rows will be added to the grid as necessary.

Note. These changes are made to the graph only and are not stored in the corresponding graph dataset.

Pasting Graph Data To Spreadsheets

To paste graph data to a spreadsheet, use **Copy** from the **Edit** menu when the data has been highlighted in the graph data table, then paste into the spreadsheet.

It is also possible to export selected or all graph datasets to a .csv file by using the context menu for the Graph entry or graph dataset name in the Utilities treeview. Columns of data are created with graph dataset names being written to row one of each column.

Printing Graphs

Graphs may be printed using the **File> Print** command. Because graphs are created as separate windows a single graph will be printed on each page.

To print multiple graphs together use **Copy** from the **Edit** menu when the graph area is active, then paste into a suitable word processor.

Case Study. Plotting Families Of Curves

To compare results at different nodes a graph will be plotted of the stress throughout an analysis at three nodes, say 6, 25, and 60. Using a suitable results file, first select the nodes with the cursor, then:

1. Start the **Graph Wizard** from the **Utilities** menu.
2. Choose **Time History**, click **Next**.
3. For the X axis choose **Named** variable, click **Next**, choose **Response Time**, click **Next**.
4. For the Y axis choose **Nodal**, click **Next**, specify the results entity as **Stress**, specify node number **6**, click **Next**.
5. Either type suitable graph and axes titles, or leave them blank to use default names, click **Finish**. The graph is displayed.
6. Repeat steps **1** and **2**.
7. For the X axis choose **Named** variable, click **Next**, but this time choose **Previously defined**, click **Next**. Select **Response Time** from the drop-down list, click **Next**.
8. For the Y axis choose **Nodal**, click **Next**, specify the results entity as **Stress**, specify node number **25**, click **Next**.

9. Choose **Add to existing graph**, make sure to specify the correct graph from the list if there is more than one, click **Finish**. The new data will be added to the first graph.
10. Repeat steps **1**, **2** and **7**.
11. For the Y axis choose **Nodal**, click **Next**, specify the results entity as **Stress**, specify node number **60**, click **Next**.
12. Repeat step **9**.

Creating Animation Sequences

A LUSAS animation displays a sequence of pictures showing the status of the model or results type for selected loadcases. Animations are useful for viewing the effects on a structure of a moving load, checking that a staged construction modelling process has been defined correctly, or visualising the changing results of a nonlinear, dynamic or transient analysis. Sometimes the manner in which a structure deforms is not always obvious when comparing its undeformed and deformed shapes, and it may be better understood using animation.

The structure may be animated in two ways. Both types of animation are created from the **Utilities> Animation Wizard** menu item.

Animating an active loadcase

- **Active Loadcase** (for results loadcases only) The results from a single loadcase, or eigenvector mode shape, may be animated according to a **deformation function** (sine, square, saw tooth, or ramp). A full sine wave (-1 to 1) is useful for animating mode shapes obtained from an Eigen analysis.

Animating Load History

- **Load History** animates chosen model or results loadcases producing a animation frame for each.

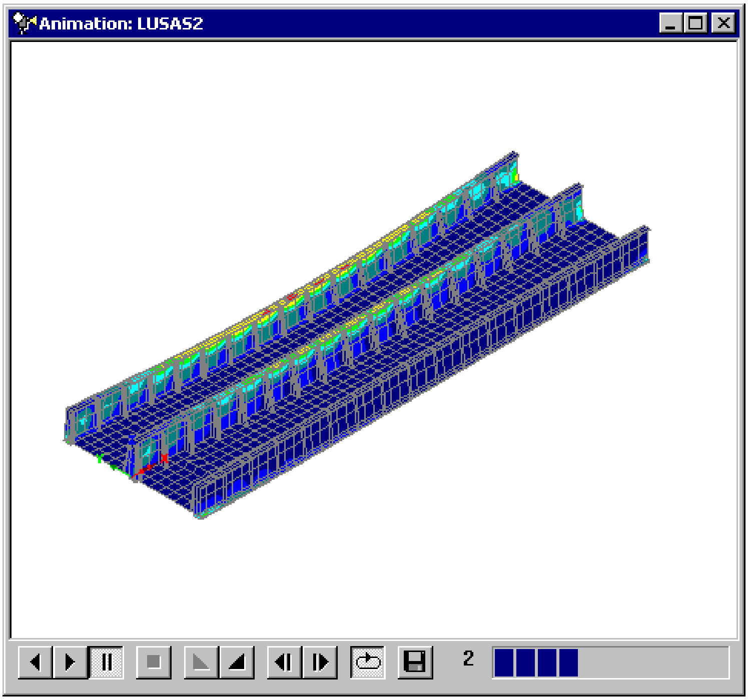
The load history can be animated for both model or results loadcases.

- **All loadcases** animates all available results loadcases.
- **Final increment** animates all available results loadcases, using only the last increment of each nonlinear results loadcase.
- **Specified** allows a user-defined selection of model or results loadcases for animation.

Resolution

The resolution and aspect ratio of an animation can be set by specifying the number of pixels in the horizontal and vertical directions. Initial default values are related to the model view window size and set to be either 800 pixels horizontally or 600 pixels vertically according the view window aspect ratio.

The content of the animation sequence is defined by the contents of the current window when the Animation Wizard is started. For example, to animate contours on a deformed mesh, a contour and deformed mesh layer need to have been added to the current view window prior to running the animation wizard.



Notes

- If contours are to be included in an animation, it is useful to fix the contour range across multiple loadcases using a global or manual scale before creating the animation sequence. Setting the contour range will fix contour colours on each frame of the animation sequence relative to the others. The spread of stress, or other entity, can then be seen more readily. It is often useful to set an upper and lower value based upon the maximum and minimum values achieved across a range of loadcases.

- When animating deformed models it is recommend that the auto resize button  is switched off before creating the animation to prevent re-scaling during the animated sequence.

- Animation is carried out by capturing an entire set of pixels for each frame of the animation for the full view window and not just the extent of the model. As a result, for some models, any 'blank space' around a model will be also captured with each animation frame saved. To reduce the amount of unwanted area captured the view window can be re-sized to the proportions of the model width and height.

- The only model loadcases (that is, those loadcases that are saved with a model) available for inclusion in an animation are basic modelling loadcases and basic load combinations. Envelope and Smart Combination loadcases that are saved with a model cannot be animated.
- Increasing the number of included loadcases will require a proportionately longer amount of time to create an animation.
- When animating staged construction models the mesh layer display should be set to Show activated elements only in order to see the model building sequence.

Controlling the Animation

The buttons at the bottom of the animation window allow the animation to be speeded up or slowed down, stepped frame by frame, looped, and saved.

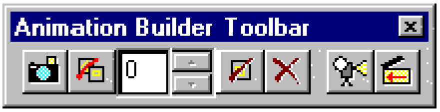
Saving Animation Files

Animation files are saved as Windows standard **.avi** files using the **File> Save As AVI** menu item. or by pressing the **Save As AVI** button on the Animation window. The **"Microsoft Video 1"** compression method is used by default with a 75% compression setting to reduce the file size. This produces a good reduction in file size without noticeable loss in quality and should be compatible with the majority of PCs.

Using the Animation Builder Toolbar

The animation tool builder tool bar can be found using the **View> Toolbars** menu item.

Animation sequences may be edited, or can be created frame by frame, using the animation builder toolbar.



Printing Results

Selected results values may be output to the screen in a tabular listing format for the active loadcase or for selected loadcases. Once listed the results can be printed or saved to a spreadsheet. The results for each loadcase are displayed on a separate tab in the print results window. A model info tab appears in all output windows and provides basic information about the model.

To print results to the screen his use the **Utilities> Print Results Wizard** menu item.

When more than one loadcase is present a choice can be made:

- ❑ **Active** outputs selected results from a single loadcase.
- ❑ **Selected** permits output of selected results for one or more loadcases.

Entity

The **Entity** chosen dictates the type of results printed. When smart combinations or envelopes are present the primary component can be set such that all other values are associated values which occur at the same time as the enveloped component. Only those components applicable for the elements used in the model will be displayed.

The entities and results types available (when applicable) are:

None

- **Eigenvalues** - eigenvalue, frequency and error norm.
- **Participation Factors -** participation factors in X, Y and Z directions.
- **Mass participation factors -** mass participation factors in X, Y and Z directions.
- **Sum of mass participation factors -** sum of mass participation factors in X, Y and Z directions. (This enables the % of active mass in each direction to be determined, as the sum of the mass participation factors in each direction should be unity).

Displacement, Residual, Reaction, Reaction Stress, Loading, Potential, Flux, Gradient

- **Component -** component results in tabular format.
- **Summary** - maximum and minimum visible values and their position on the model.

Stress/Strain

- **Component -** component results in tabular format.
- **Summary** - maximum and minimum visible values and their position on the model.
- **Principal** - principal values in tabular format.
- **Fatigue** or **Damage Results** - fatigue (Log Life) and damage results in tabular format.
- **Energy** - strain energy and plastic work results.
- **State variables** - extra nonlinear material parameters.

Slideline results

- **Component -** component results in tabular format.
- **Summary** - maximum and minimum visible values and their position on the model.
- **System forces** - forces in global or current transformed directions.
- **Gap forces** - forces required to reverse nodal penetration.
- **Contact forces** -forces generated across contact surfaces.
- **Contact stresses** - contact stresses computed as contact force/contact area in normal and tangential directions.
- **Section results** - generic contact results e.g. contact state, normal penetration etc.

For more details see **slideline results processing**.

Thermal surfaces results

- **Component** - underlying nodal results.
- **Summary** - maximum and minimum values (according to the extent specified) and their position on the model.
- **Flows** - flow components.
- **View factors** - summary of view factor sums across segments.

Transformation of results

- **Transformed** Printed results may be **transformed** to be relative to a specified local coordinate, according to element local directions for stresses, relative to the local element material directions, or to a specified angle in the XY plane.

Wood Armer

- **Wood-Armer** - Calculation of top and bottom reinforcement moments for plates and grillage elements and top and bottom reinforcement forces for shell elements.

Location

When applicable, printed results can be obtained for the following locations:

- **Averaged nodal -** Average nodal (smoothed) results from visible elements. Nodal results are extrapolated from the Gauss point values within each element before averaging.
- **Gauss Point** - Gauss point values internal to the visible elements. The most accurate results available from the analysis.
- **Element Nodal -** Unaveraged nodal results for visible elements. Nodal results are extrapolated from the Gauss point values within each element.
- **Coordinates** When the optional **Coordinates** checkbox is enabled the global X, Y and Z positions of the nodes or gauss points (as appropriate) are included as separate columns in the printed output. If a column heading is double-clicked the results will be sorted in ascending or descending order based upon coordinates. This enables sorting of nodal and gauss results data by coordinate.

Extent

The printed results for selected loadcases are governed by the following extent options:

- **Elements showing results** - default option which prints results only for those elements that are displaying them in the Modeller view window.
- **Visible model** - prints results all elements that are visible in the Modeller view window.

- **Full model** - prints results for all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.
- **Specified group** - prints results for a specified named group. Only active if group names have been defined.

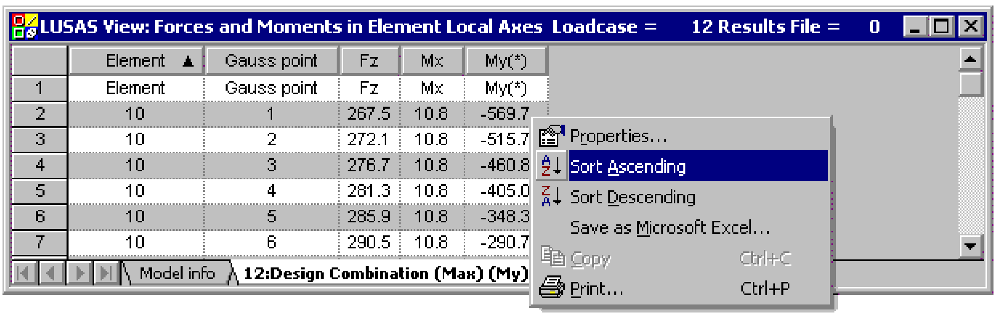
Printed results for automatically created slices

When section slices have been defined on a 3D model (with corresponding groups being automatically created in the Groups treeview) the results for just the automatically-defined groups/slices can be printed to the print results window by selecting the **Display for slice(s)** check box. Note that the automatically created group name 'Slices' is a collective name for the automatically created slice group names and does not contain results.

Results display and manipulation

The results for each selected loadcase are displayed on a separate tab in the print results window. A model info tab also appears in all output windows and provides basic information about the model. When the Printed Results window is displayed a context menu can be invoked which allows for the printed results data to be manipulated:

- **Properties** The number of significant figures or decimal places can be changed.
- **Sorting of data** Results data can be sorted in ascending or descending order. In addition, data sorting can be achieved by double-clicking on a header to sort by that column name. A second double-click on the same header will carry-out a reverse sort.
- **Saving to a spreadsheet** The contents of the current tab or all tabs can be saved to a spreadsheet or to a text file.
- **Copying to the clipboard** Selected cells or the whole grid can be copied to the clipboard.
- **Printing** The Print option sends the contents of the active tab to the printer.



Manipulating printed loadcase results

Printing results for Envelopes or Combinations

When the active loadcase is an envelope or smart combination the results printed will show the primary component (e.g. **Fx**) marked with an asterisk. Additionally, for Envelopes only, the loadcase in which the maximum or minimum value was extracted will be tabulated in the LCID (Loadcase ID) column.

When enveloping on **All** components, the loadcase from which the results are extracted cannot be tabulated because each individual results component may have come from a different loadcase.

LUSAS View: Averaged Forces and Moments...

	A	B	C	D	E	F
1	Node	Fx(*)	Fy	Mz	LCID	IRES
2	1	38.2381	-252.024	600.284	3	0
3	2	-68.4591	-48.6257	-103.858	3	0
4	3	868.661	0.0	59.092	1	0
5	4	-2.40769	-9.54964	-11.9554	3	0
6	5	839.402	-50.9965	-11.9912	1	0
7	6	-93.4626	-53.8285	-19.4126	3	0
8	7	-38.2381	-247.976	593.811	3	0
9	8	-96.8636	-48.9366	75.7912E-15	3	0
10	9	839.402	50.9965	-42.994	1	0
11	10	84.4685	-50.9805	-18.2945	3	0

Envelope results for primary component Fx showing Loadcase ID

LUSAS View: Averaged Forces and Moments...

	A	B	C	D	E	F
1	Node	Fx	Fy	Mz	LCID	IRES
2	1	38.2381	71.6582	600.284	0	0
3	2	-68.4591	-45.7704	91.9813	0	0
4	3	868.661	0.0	59.092	0	0
5	4	-2.40769	0.236848E-12	58.6137	0	0
6	5	839.402	-50.9965	20.0294	0	0
7	6	-93.4626	-51.2462	-19.4126	0	0
8	7	-38.2381	-71.6582	593.811	0	0
9	8	-96.8636	45.7704	75.7912E-15	0	0
10	9	839.402	50.9965	19.1477	0	0
11	10	84.4685	51.2462	-10.4339	0	0

Envelope results for primary component All (No Loadcase ID can be shown)

Notes

- Selected slice and slideline data and results may be printed from the group context menu.
- To print results and include model images in a report style format see **Generating Reports**

Printing and Saving Pictures

Views of the LUSAS model in the graphics area may be printed directly to the default printer from the graphics area using the print  button.

The **File> Print Preview** menu item is useful for visualising the document prior to printing taking place. Using the **File> Print** menu item allows alternative printer settings to be used.

Note. When a model is created the default paper size (for printing use) is set from the settings of the default printer installed on the local PC. This helps to ensure that regional paper sizes are used in preference to otherwise specified sizes.

Saving pictures for use in LUSAS reports and other applications

Views of the LUSAS model can be saved as BMP, JPG, or WMF files using the **File> Picture Save** menu item. BMP and JPEG files are saved to a fixed size of 1800 pixels in width with a height proportional to the size of the graphics window when the file was saved. JPEG files are the most efficient to save in terms of file size. Windows Meta Files now contain bitmaps instead of vectors for the modelling information with correspondingly smaller file sizes. Text and annotation layer information held in a WMF file is vector-based and is therefore scalable.

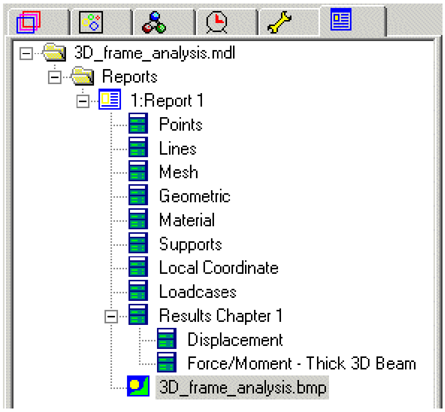
Note. The contents of the View window can also be copied (and subsequently pasted) for use in other applications by clicking the right mouse button and selecting **Copy** from the context menu.

Saving pictures for viewing in Expose

Pictures can also be saved as **LUSAS Picture Files** for viewing only in the LUSAS picture file utility program, Expose. Note that Graphs cannot be saved in LUSAS picture file format.

The contents of the graphics area can also be transferred to the Windows clipboard using the copy button.

Generating Reports



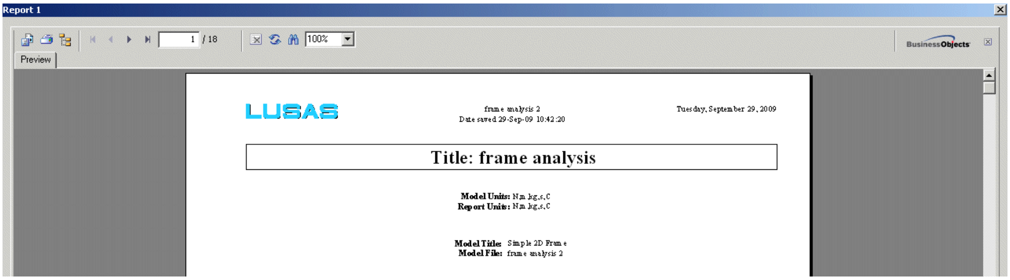
Report templates (which are created, modified and saved in the Reports Treeview for each model) hold the information required to generate reports for model or results data. Each time a report is to be viewed, the report details that are specified in the report template are extracted from the model and results files, and used to create the report.

A report is built by defining chapters that reference the model and results data that are to be included in the report. The modelling and results data selected can be restricted to particular model geometry, model attributes or loadcases, or for particular results entities, and data can be listed for all of, or just parts of a model. Additional user content such as screen captures, saved images or additional text can also be added. These user content items appear as separate chapters in the report.

Each report template can include any number of chapters that define the model attributes and analysis results to be viewed. Note that the creation of results chapters is only possible when results are loaded on top of a model. Any number of reports templates may be created and saved in the Reports Treeview of a model. The order of information in a report can be changed by dragging and dropping the chapter names up and down the Report Treeview.

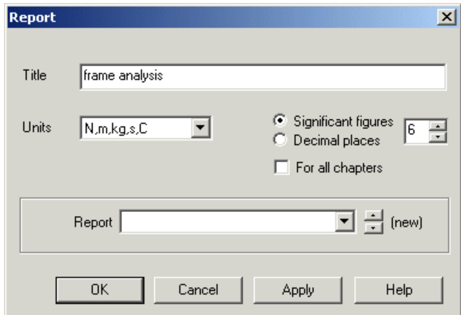
Reports are viewed in a third-party industry standard report viewer called BusinessObjects Crystal Reports (included with in a LUSAS software installation). Report data may be exported to Microsoft Excel spreadsheets for additional calculations to be carried out as well as being exported in PDF, RTF (for use in Microsoft Word), HTML and other formats.

If it is desired that only results values are to be viewed on screen or be selectively printed or output to spreadsheets the **Print Results Wizard** can be used.



Creating a Report

A new (empty) report template is created by selecting the **Utilties> Report** menu item or from right-clicking **New Report** from the **Reports** context menu in the Report Treeview. On the Report properties dialog:



- The report **Title** is optional and is used as a title in the exported report.
- **Units** for a report are, by default, the same as those of the model. However, it is possible to prepare a report in a different system of units, in which case all values seen in the report will be converted appropriately.
- It is also possible to control the number of **significant figures** and **decimal places** seen in the report. These can also be specified independently for each chapter
- If **For all chapters** is selected the values for significant figures or decimal places chosen on this dialog will be used throughout the report. This option overrides any different values set inside each chapter.
- The **Report** name is the name added to the Report Treeview. By default reports are named Report 1, Report 2 etc if no name is specified.

Once a New Report entry has been added to the Report Treeview, selecting the report name and using its context menu enables the adding of chapters to a report, modifying, viewing, renaming, or deleting of a report. Report templates are saved in the Report Treeview when a model is saved.

Adding a Report Chapter

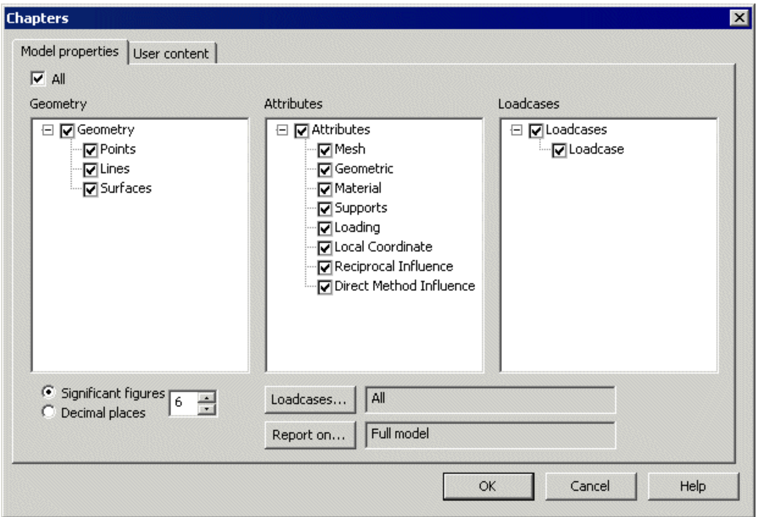
Model properties, loadcase and basic combinations, envelopes and smart combination results, eigenvalue results and user information such as images and text may be added to a report in the form of chapters. Each chapter represents one single aspect of the model. Multiple chapters may be added and re--ordered as necessary in the Report Treeview.

Chapters are added to a report by selecting the **Add Chapter** menu item from the Report name context menu (accessed by right-clicking the mouse button).

The following chapters can be added:

- **Model Properties**
- Loadcase/Basic Combination results
- **Envelope/Smart Combination results**
- **Eigenvalue results**
- **User Content**

Add or Edit a Model Properties Chapter



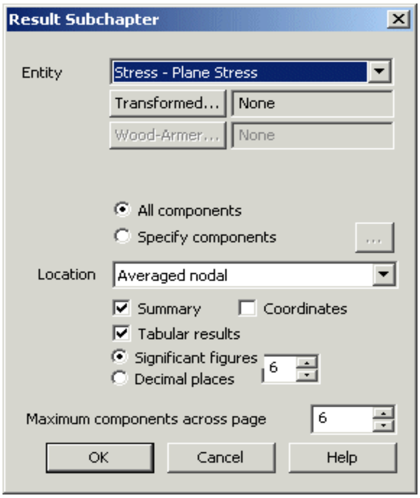
- The **Model Properties** tab of the Chapters dialog allows model geometry, attributes and loadcase/IMD loadcase information to be added to the report via the use of tick boxes.
- Use the **Loadcases…** button to display a dialog which restricts the chapter to display results for **All**, **Active** or **Specified** loadcases or combinations. Note that, by default, all loadcases are selected, but for some situations (notably when reporting on loading attributes for a model that have been generated by vehicle load optimisation software) a physical limit may be met because the generated report is too big for Windows to display. If this occurs, use the Loadcases button to reduce the number of loadcases selected so that a report can be generated, and generate a separate report for the remaining loadcases.
- The **Report on…** button displays a dialog which controls the scope of the chapters to be created. By default, the whole model is selected, but a report could also created for just the visible model or for a specified group.

It is also possible to control the number of significant figures or decimal places for this chapter as presented in the report.

Note: It is possible to visit this dialog several times to create multiple chapters, each of which can have different ordering, scope and loadcase choices. For example you can create one chapter describing the lines in group 1 and subsequently to create a different chapter describing the lines in group 2. Once created, the order of chapters in the report can be modified at any time by dragging and dropping them up and down the Report Treeview.

Add or Edit a Results Subchapter (for an Entity)

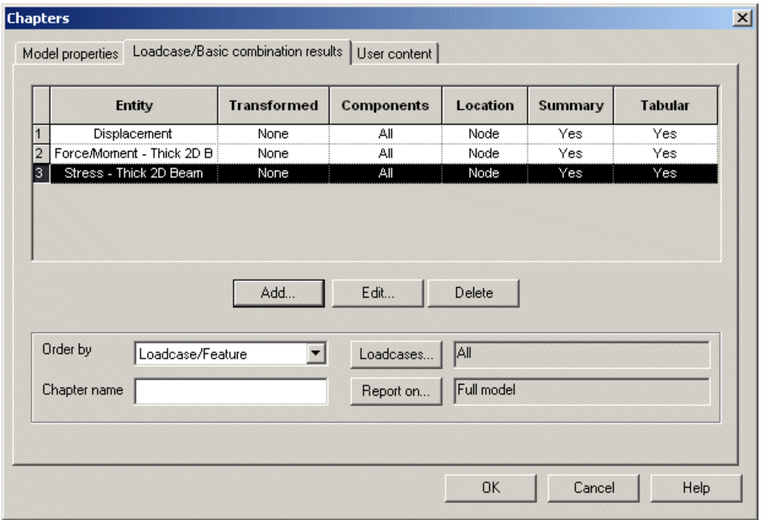
You can create or modify a section within a results chapter. Select the **Add...** button on the Chapters dialog to display this dialog.



- **Entity** chosen dictates the type of results. The component chosen is only appropriate to Envelopes and Smart Combinations and controls the primary component – equivalent to the component chosen when the Envelope or Combination is set active.
- **Transformed** Printed results may be **transformed** to be relative to a specified local coordinate, according to element local directions for stresses, relative to the local element material directions, or to a specified angle in the XY plane.
- **Wood Armer** Calculation of top and bottom reinforcement moments for plates and grillage elements and top and bottom reinforcement forces for shell elements
- Use the **All Components** or **Specify components** and **…** buttons to control which results component values will be added to the report.
- The **Location** drop-down specifies whether **Averaged nodel**, **Gauss Point** or **Element Nodal** results should be calculated.
- The **Summary** checkbox chooses whether or not to display a summary for each results component chosen. This summary consists of the maximum and minimum values encountered, along with their location. Note that sub-reports cannot be created from Summary results information when listed in a report.
- The **Tabular results** checkbox chooses whether or not to display a table of numerical results. If chosen, a value will be written to the report for each component, for each loadcase, for each node or gauss point chosen. Such tables can be very large. Note that sub-reports can be created from Tabular results information when listed in a report.
- Additionally, **Nodal coordinates** can be added to the report. These take the form of additional columns of data in the tabular results, one each for the X, Y, and Z coordinates of each node.
- The number of **Significant figures** or **Decimal places** can be specified for values written to the report for this chapter.

- ❑ **Maximum components across page** specifies how many columns of results components are expected to be able to be set-out across the width of the report page. If the number of results components available exceeds the number of components that can actually be accommodated across a page for a certain paper size and orientation a choice is offered of wrapping the results components or splitting the results sub-chapter into multiple sub-chapters to avoid wrapping. Note than when results components need to be wrapped a separate entry is created in the Add/Edit Chapter dialog grid for each set of wrapped data. The Components column of the grid shows All for results data that is not force-ably wrapped, and Specified for results data that is force-ably wrapped.

Add or Edit a Loadcase Results Chapter



The Loadcase/Basic Combination results tab of the Chapters dialog is only shown if a results file is loaded.

The **Envelope/Smart Combination results** tab (not shown) is only added to the dialog if a model contains envelope or smart combination results.

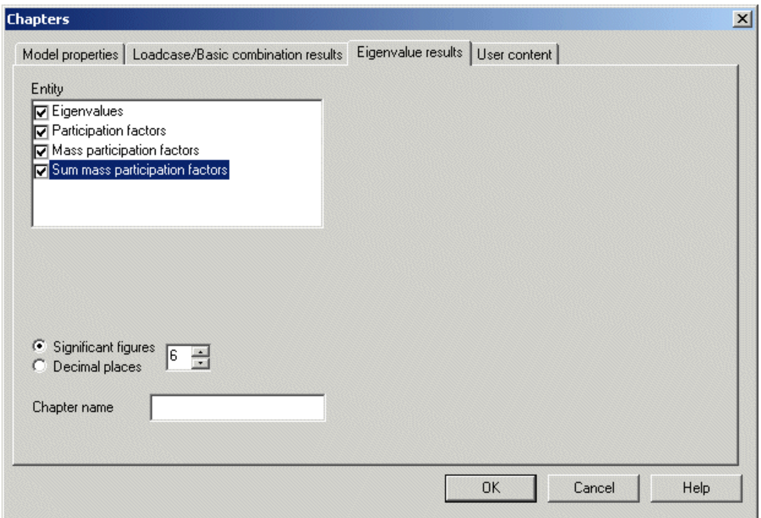
- Use the **Add...** button to add entities for inclusion in the report as results sub-chapters. When added, results sub-chapters are shown in a grid form at the top of the dialog. By selecting the Entity name in the grid you can subsequently **Edit** or **Delete** existing content.
- Use **Order by...** to dictate the order in which the tabular results sub-headers are presented. This is similar to the concept of sorting data by column header in a spreadsheet program. The order by options are: **Loadcase / Features**, **Features / Loadcase** and **Loadcase / Mesh**. The use of the Order by facility is of particular

importance when exporting results to a spreadsheet format where the use of Order by Loadcase / Mesh is recommended because of the reduced number of blank lines it creates in the output file.

- Use the **Loadcases…** button to display a dialog which restricts the chapter to display results for **All**, **Active** or **Specified** loadcases or combinations. By default, all loadcases are selected.
- Use the **Report on...** button to display a dialog which controls the scope of the chapter to be created, for example to restrict the chapter to display results for the **Full model**, the **Visible model** or a **Specified group** only. By default, the whole model is selected.
- **Chapter name** can be edited if the default or previously entered name is to be altered.

Note: It is possible to visit this dialog several times to create multiple chapters, each of which can have different ordering, scope and loadcase choices. For example you can create one chapter describing the displacements for the whole of a model and subsequently to create a different chapter describing the stress in a particular group of elements. Once created, the order of chapters in the report can be modified at any time by dragging and dropping them up and down the Report Treeview.

Add or Edit an Eigenvalue Results Chapter



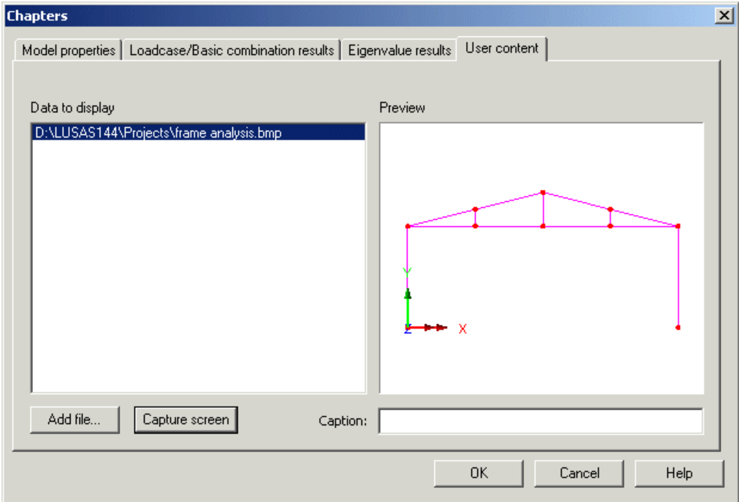
The **Eigenvalue results** tab of the Chapters dialog is only shown if a model contains eigenvalue results. The following eigenvalue results can be selected for listing:

- ❑ **Eigenvalues**
- ❑ **Participation factors**
- ❑ **Mass participation factors**
- ❑ **Sum mass participation factors**

The number of **significant figures** or **decimal places** for this chapter can be specified.

The **Chapter name** can be specified or edited if the default or previously entered name is to be changed.

Add or Edit a User Content Chapter



The **User Content** tab of the Chapters dialog allows images or text files to be added to the report using the **Add file...** button.

The **Capture screen** button takes a snap-shot of the current view window and allows it to be saved as a fixed-size BMP, JPG or WMF file to the working folder (by default) or to any other specified folder. Note that the captured image will include the full view window and not just the extent of the model. As a result for some models 'blank space' will appear around each image saved. To avoid this, the view window can be re-sized to the proportions of the model width and height in order to reduce the amount of unwanted area captured.

The **Caption** field provides the means to associate a title with each image and have it written beneath each image in the report.

Clicking the **OK** button adds this image to the report as a separate chapter. Only BMP files currently appear in the preview pane. Each image added to the report is added as a separate chapter.

Note: It is possible to visit this dialog several times to create multiple chapters. Once created, the order of chapters in the report can be modified at any time by dragging and dropping them up and down the Report  Treeview.

Chapter Extent

The extent of data that the chapter is to report on can be specified by selecting either:

- **Elements showing results** - prints results only for those elements that are displaying them in the Modeller view window.

- **Visible model** - prints results all elements that are visible in the Modeller view window.
- **Full model** - defaults option which prints results for all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.
- **Specified group** - prints results for a specified named group.

For example, if the selected chapter describes Materials, by selecting a Specified group only the Material assignments used in that group will be present in that chapter.

Loadcase Selection

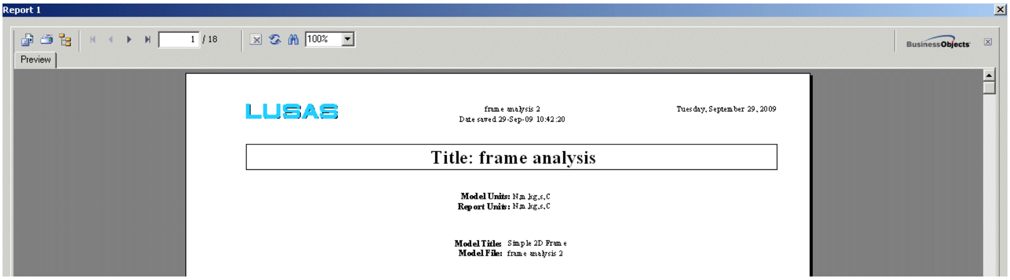
The loadcases, combinations or envelopes that the chapter is to report on can be specified by selecting either:

- **All**
- **Active**
- **Specified**.

If multiple loadcases are specified, multiple entries will appear in the report.

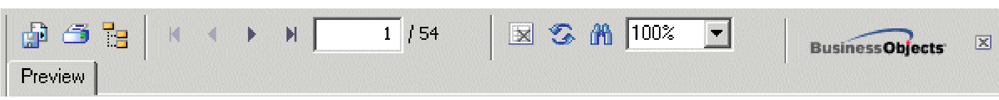
Viewing a Report

Reports are viewed from the Report Treeview by double-clicking on the report name or by choosing the **View Report...** menu item from the report name context menu. After a short delay whilst the report data is assembled and formatted, a report consisting of all the selected chapters will be displayed inside the third-party BusinessObjects Crystal Reports viewer.

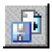


BusinessObjects Crystal Reports viewer

The BusinessObjects Crystal Reports viewer is a linked-in third-party application that is widely used in industry to present and manipulate report data. It has a toolbar that provides the following buttons / facilities for viewing, manipulating, printing and exporting the selected LUSAS model and results data:



Report Viewer Toolbar Buttons

Button	Name and usage
	**Export Report** permits reports or sub-reports to be exported to a variety of formats to any of the following destinations: • To a file on your disk (**Disk file**). • To an **Application** on your computer that will open once the file has been created. • To a folder in your mail client (Exchange).
	**Print Report** prints the current report or sub-report view only to a specified printer.
	**Toggle Group Tree** permits the viewing of loadcase/feature numbers in sub-reports in a treeview style format.
	**Page selection** These options provide the means of moving or jumping to a specific page.
	**Stop Loading** stops the loading of large files
	**Refresh** refreshes the view contents
	**Search Text** provides the means to find and jump to particular words in the report or sub-report.
	**Scaling** adjusts the size of the page view
	**Close Current View** closes the sub-report view leaving other views visible until they are closed. To go back to the LUSAS Modeller window the report viewer must be closed by using the report window's main Close button.

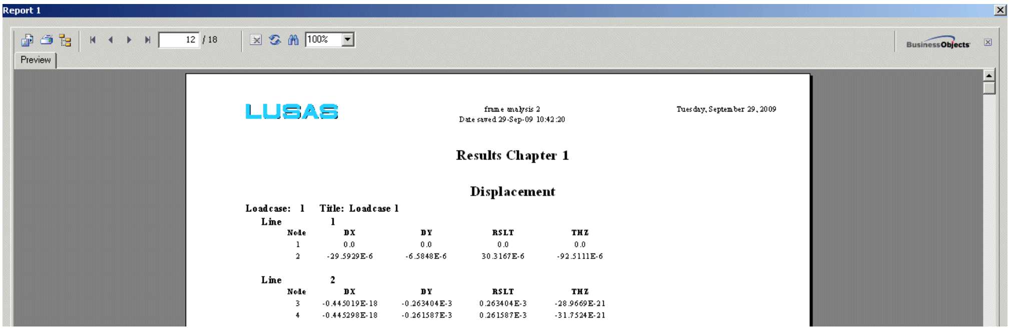
Notes

- When viewing a report using the Crystal Reports viewer no changes can be made in LUSAS Modeller. The only way a LUSAS model or the report listing (as held in the LUSAS Report Treeview) can be re-edited is to close the report view. This is to ensure that the report data always matches the model from which it is created.
- When a report is first loaded into the report viewer the Page Down key will not work until the view has acquired focus. Simply click anywhere in the view to set the focus.
- When saving to disk the default export directory is a temporary directory specified by the report viewing software that is used. Browse to your LUSAS project directory if

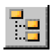
you wish to save your report file with your model. The number of pages to be exported or saved can be specified.

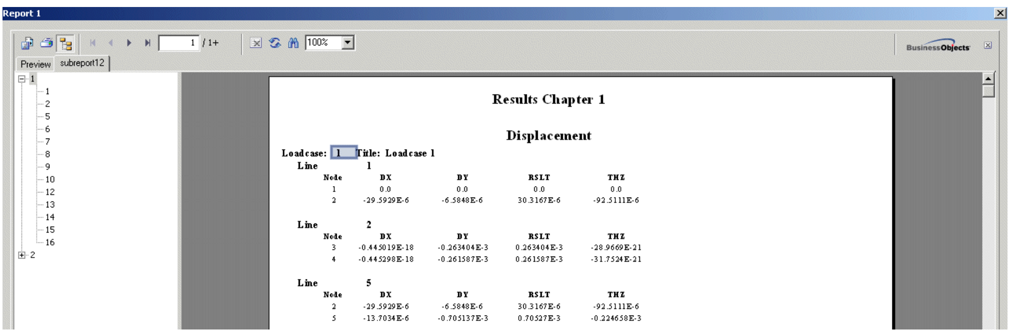
Creating and Viewing Sub-Reports

When viewing a report, selected sections of data may be viewed in a sub-report. This can be particularly useful when wanting to export selected data to another application as, for instance, when exporting report results data to a spreadsheet. It also provides an easy way of visually printing selected pages of a main report.



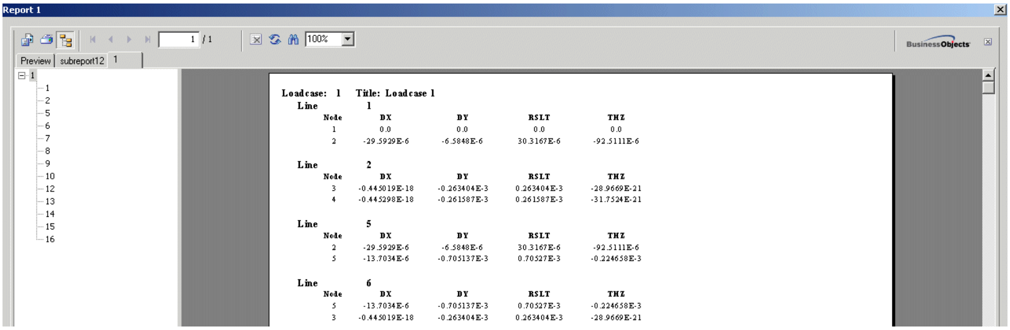
- To create a sub-report from a section of a normal report double-click on the main body of the data where you wish a sub-report to be generated, or double-click on a feature or loadcase name. A new tab will appear in the report viewer next to the Preview tab.

- Selecting the **Toggle Group Tree**  button will permit the viewing of the loadcase/feature numbers in the sub-reports in a treeview style format as shown on the following image.



Note that it is also possible to create a sub-report from a sub-report. For example, when a sub-report containing results for a set of loadcases is being viewed, a sub-report for a particular loadcase, or for a particular feature such as a line, surface or volume, could also be created.

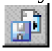
- To create a sub-report from a sub-report view double-click on the loadcase name (as shown in the previous image) or feature for which the sub-report should be created. A new tab will appear in the report viewer next to the previous sub-report tab as shown on the following image.



Notes

- Not all sections of a report can be selected to make a sub-report. The cursor will change to a magnifying glass when it is over data that can be selected for a sub-report.
- The toolbar buttons such as Export Report, Print Report, etc., act on the currently selected preview tab (and hence, the currently selected report view). This means that only those pages in the particular report or sub-report that is being viewed will be exported or printed.
- To delete the visible sub-report select the **Close Current View** button at the top-right of the report viewer toolbar.

Exporting Report Data

When viewing a report, report data may be exported to a variety of formats to any of the following destinations by use of the **Export Report** button 

- ❑ To an **Application** on your computer that will open once the file has been created
- ❑ To a file on your disk (**Disk file**)
- ❑ To a folder in your mail client (**Exchange**)

In each case the number of pages to be opened or saved can be specified.

Note that when saving to disk the default export directory is a temporary directory specified by the report creation software that is used. Browse to your LUSAS project directory to save your report file with your model if required.

When the number of columns in a report become too large for a portrait view use Page Setup (accessed from the Report Name context menu) to either change the report page margins, or to change the report page orientation to Landscape.

Exporting Report Data to a Spreadsheet

When viewing a report, report data may be exported to a spreadsheet such as Excel by use of the **Export Report** button

- ❑ Select a Format of: **Microsoft Excel 97-2000 - Data only (.XLS)**
- ❑ Select a Destination of: **Application**
- ❑ Select **Custom** format options

Note that use of the Results chapter **Order by...** option to dictate the order in which the tabular results sub-headers are presented is of particular importance when outputting results data to a spreadsheet. The option to order results data according to **Loadcase / Mesh** is recommended because of the reduced number of blank lines it creates in the output file.

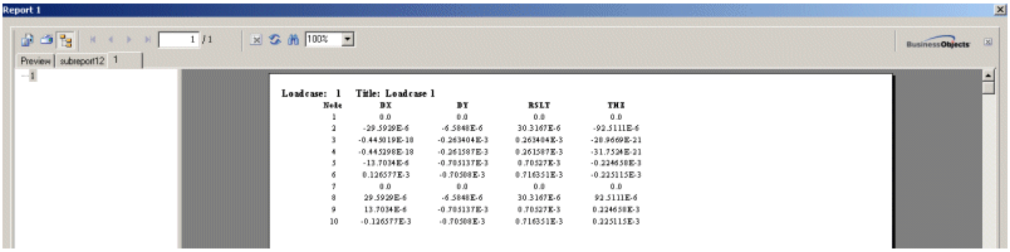
The images that follow show an example of a sub-report created with Loadcase / Mesh order and the corresponding results exported to an MS Excel spreadsheet.

This **Order by** option should be set prior to exporting the data as follows:

1. In the Report Treeview, select the Results Chapter name and use its context menu to the select **Modify** menu item which, in turn, will display the Edit Chapter dialog.
2. Use the **Order by...** drop-down to select **Loadcase / Mesh** and click **OK**
3. In the Report Treeview, select the report name containing data to be exported and use its context menu to select **View Report**
4. After the report is displayed in the report viewer, find the Results Chapter containing data to be exported and double-click in the body of the data to create a sub-report
5. If results for a particular loadcase is to be exported, double-click on that loadcase data to create a further sub-report
6. Lastly, use the **Export Report** button to allow selection of the Format and Destination of the results data to be created.

Example output

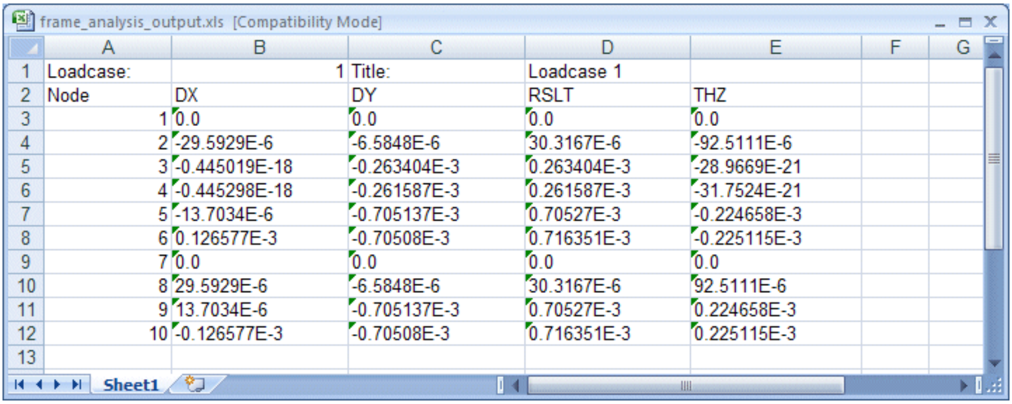
The images that follow show an example of a sub-report created by LUSAS with Loadcase / Mesh order and the corresponding results exported to a Microsoft Excel spreadsheet.



Loadcase: 1 Title: Loadcase 1

Node	DX	DY	RSLT	THZ
1	0.0	0.0	0.0	0.0
2	-29.5929E-6	-6.5848E-6	30.3167E-6	-92.5111E-6
3	-0.445019E-18	-0.263404E-3	0.263404E-3	-28.9669E-21
4	-0.445298E-18	-0.261587E-3	0.261587E-3	-31.7524E-21
5	-13.7034E-6	-0.705137E-3	0.70527E-3	-0.224658E-3
6	0.126577E-3	-0.70508E-3	0.716351E-3	-0.225115E-3
7	0.0	0.0	0.0	0.0
8	29.5929E-6	-6.5848E-6	30.3167E-6	92.5111E-6
9	13.7034E-6	-0.705137E-3	0.70527E-3	0.224658E-3
10	-0.126577E-3	-0.70508E-3	0.716351E-3	0.225115E-3

Sub-report listing ordered by Loadcase / Mesh



	A	B	C	D	E	F	G
1	Loadcase:	1	Title:	Loadcase 1			
2	Node	DX	DY	RSLT	THZ		
3	1	0.0	0.0	0.0	0.0		
4	2	-29.5929E-6	-6.5848E-6	30.3167E-6	-92.5111E-6		
5	3	-0.445019E-18	-0.263404E-3	0.263404E-3	-28.9669E-21		
6	4	-0.445298E-18	-0.261587E-3	0.261587E-3	-31.7524E-21		
7	5	-13.7034E-6	-0.705137E-3	0.70527E-3	-0.224658E-3		
8	6	0.126577E-3	-0.70508E-3	0.716351E-3	-0.225115E-3		
9	7	0.0	0.0	0.0	0.0		
10	8	29.5929E-6	-6.5848E-6	30.3167E-6	92.5111E-6		
11	9	13.7034E-6	-0.705137E-3	0.70527E-3	0.224658E-3		
12	10	-0.126577E-3	-0.70508E-3	0.716351E-3	0.225115E-3		
13							

Sub-report data exported to a Microsoft Excel spreadsheet.

Exporting Report Data to a Word Document

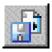
When viewing a report, report data may be exported to a Word Document by use of the **Export Report** button This will require you to:

- ❑ Select a Format of: **Microsoft Word - Editable (RTF)**
- ❑ Select a Destination of: **Application**
- ❑ Select a page range.

Note that the Results chapter **Order by...** option which dictates the order in which the tabular results sub-headers are written is particular use when exporting results data. If you do not wish to group the results per feature (e.g. per line) which is the default, then the option to order by **Loadcase / Mesh** should be used.

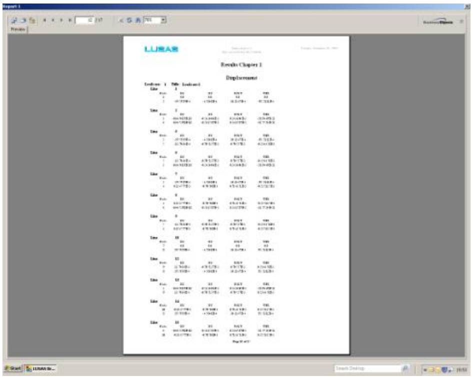
This **Order by...** option should be set prior to exporting the data as follows:

1. In the Report Treeview, select the Results Chapter name and use its context menu to the select **Modify** menu item which, in turn, will display the Edit Chapter dialog.

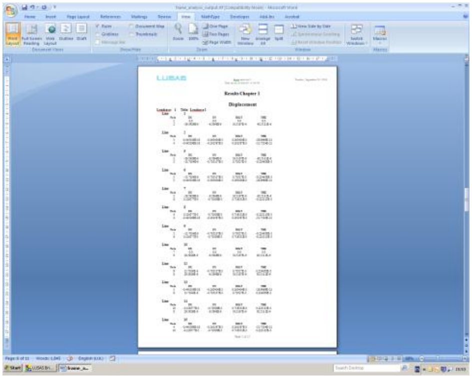
2. Use the **Order by...** drop-down to select the Order required e.g. **Loadcase / Mesh** and click **OK**
3. In the Report  Treeview, select the report name containing data to be exported and use its context menu to select **View Report**
4. After the report is displayed in the report viewer, find the Results Chapter containing data to be exported and double-click in the body of the data to create a sub-report.
5. If results for a particular loadcase is to be exported, double-click on that loadcase data to create a further sub-report.
6. Lastly, use the **Export Report** button  to allow selection of the Format and Destination of the results data to be created.

Example output

The images that follow show an example of a report created by LUSAS and the same data exported to an Microsoft Word document.



Report listing in the report viewer



Report data exported to a Microsoft Word document.

Printing Report Data

When viewing a report, report data may be printed by selecting the **Print Report** button  in the BusinessObjects Crystal Reports toolbar.

Deleting Report Data

To delete the visible report or sub-report select the **Close Current View**  button at the top-right of the report viewer toolbar.

Appendix A : Smart Combination Examples

Smart Combination Examples

The following examples demonstrate how the different factors and settings can be used in smart combinations. For the purposes of these examples the results at a single node are going to be considered.

- **Case 1** - Considers a node where long term load effects are all negative.
- **Case 2** - Considers a node where short-term load effects are of mixed sign.
- **Case 3** - Considers a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four.
- **Case 4** - Considers a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four and the "Variable loadcases" set to four.
- **Case 5** - Considers a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four and the "Variable loadcases" set to one.

Smart Combination - Case 1

Consider a node where long term load effects are all negative.

In this instance the permanent and variable load factors are considered and will be added together based on the nodal result being adverse.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for positive load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Dead load	-20	1.0	0.15	1.0	-20 x 1.0 = -20
Deck surfacing	-10	1.0	0.75	1.0	-10 x 1.0 = -10
Superimposed load	-15	1.0	0.2	1.0	-15 x 1.0 = -15
				Smart combination (Max)	**-45**

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Dead load	-20	1.0	0.15	1.0 + 0.15	-20 x 1.15 = -23
Deck surfacing	-10	1.0	0.75	1.0 + 0.75	-10 x 1.75 = -17.5
Superimposed load	-15	1.0	0.2	1.0 + 0.2	-15 x 1.20 = -18
				Smart combination (Min)	**-58.5**

Smart Combination - Case 2

Consider a node where short-term load effects are of mixed sign.

In this instance the permanent and variable load factors are considered and will be added together based on the nodal result being adverse. However as the permanent effects have been set to zero, this will only combine the results that are adverse.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for positive load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Temperature	-5	0	1.3	0	-5 x 0 = 0
Wind	-5	0	1.4	0	-5 x 0 = 0
Settlement	-10	0	1.2	0	-10 x 0 = 0
Live load 1	-20	0	1.5	0	-20 x 0 = 0
Live load 2	-15	0	1.5	0	-15 x 0 = 0
Live load 3	10	0	1.5	0 + 1.5	10 x 1.5 = 15
Live load 4	-5	0	1.5	0	-5 x 0 = 0
				Smart combination (Max) =	**15**

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Temperature	-5	0	1.3	0 + 1.3	-5 x 1.3 = -6.5
Wind	-5	0	1.4	0 + 1.4	-5 x 1.4 = -7
Settlement	-10	0	1.2	0 + 1.2	-10 x 1.2 = -12
Live load 1	-20	0	1.5	0 + 1.5	-20 x 1.5 = -30
Live load 2	-15	0	1.5	0 + 1.5	-15 x 1.5 = -22.5
Live load 3	10	0	1.5	0	10 x 0 = 0
Live load 4	-5	0	1.5	0 + 1.5	-5 x 1.5 = -5
				Smart combination (Min) =	**-83**

Within the smart combination facility there are also two check boxes marked "Loadcases to consider" and Variable loadcases". These additional options are used for a number of bridge design codes that require the loadcases in the combination to be filtered.

Smart Combination - Case 3

Consider a node where short-term load effects are of mixed sign with the “Loadcases to consider” set to four.

In this instance the permanent and variable load factors are considered and will be added together based on the nodal result being adverse. With the number of “Loadcases to consider” set to four, only the four most positive resultants will be combined for the Max combination and the four most negative resultants will be combined for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for positive load effects. The number of load effects summed is restricted to the number of loadcases specified. The loadcases used are the most adverse, for example the most positive for max combination and all other load effects assembled are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	0.8	0.7	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
				Smart combination (Max) =	**4.5**	

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects. The number of load effects summed is restricted to the number of loadcases specified. The loadcases used are the most adverse, for example the most negative for min combination and all other load effects assembled are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Used
Wind	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Not used
Settlement	-10	0.7	0.8	0.7 + 0.8	-10 x 1.5 = -15	Used
Live load 1	-20	0.7	0.8	0.7 + 0.8	-20 x 1.5 = -30	Used
Live load 2	-15	0.7	0.8	0.7 + 0.8	-15 x 1.5 = -22.5	Used
Live load 3	10	0.7	0.8	0.7	10 x 0.7 = 7	Not used
Live load 4	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Not used
				Smart combination (Min) =	**-75**	

Smart Combination - Case 4

Consider a node where short-term load effects are of mixed sign with the “Loadcases to consider” set to four and the “Variable loadcases” set to four.

In this instance the permanent and variable load factors will only be considered for the number of loadcases specified as the number of Variable loadcases to consider. The factors will be added together based on the nodal result being adverse. The remaining loadcases are considered using the permanent factor. With the number of “loadcases to consider” set to four, only the four most positive resultants will be combined for the Max combination and the four most negative resultants will be combined for the Min combination. However by setting the “variable loadcases” to four, only positive results will be considered for the Max combination and negative results for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for the *number of positive load effects* specified by the *number of Variable loadcases* to consider. The remaining positive load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are discarded. The loadcases used are the most adverse, for example, the most positive are used for a maximum combination and all other load effects assembled are discarded. Also with the variable loadcases set to four the max combination will include only positive load effects, all negative load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	0.8	0.7	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
				Smart combination (Max) =	**15**	

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects for number of negative load effects specified by the number of Variable loadcases to consider the remaining negative load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are also discarded. The loadcases used are the most adverse, for example the most negative for min combination and all other load effects assembled are discarded. Also with the variable loadcases set to four the min combination will include only negative load effects, all positive load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -.5	Not used
Settlement	-10	0.7	0.8	0.7 + 0.8	-10 x 1.5 = -15	Used
Live load 1	-20	0.7	0.8	0.7 + 0.8	-20 x 1.5 = -30	Used
Live load 2	-15	0.7	0.8	0.7 + 0.8	-15 x 1.5 = -22.5	Used
Live load 3	10	0.7	0.8	0.7	10 x 0.7 = 7	Not used
Live load 4	-5	0.7	0.8	0.7	-5 x 1.5 = -3.5	Not used
				Smart combination (Min) =	**-75**	

Smart Combination - Case 5

Consider a node where short-term load effects are of mixed sign with the “Loadcases to consider” set to four and the “Variable loadcases” set to one. The permanent and variable load factors will only be considered for the number of loadcases specified as the number of Variable loadcases to consider. The factors will be added together based on the nodal result being adverse. The remaining three loadcases are considered using the permanent factor. With the number of “loadcases to consider” set to four, only the four most positive resultants will be combined for the Max combination and the four most negative resultants will be combined for the Min combination. However by setting the “variable loadcases” only positive results will be considered for the Max combination and negative results for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for number of positive load effects specified by the number of Variable loadcases to consider the remaining positive load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are also the discarded. The loadcases used are the most adverse, for example the most positive for max combination and all other load effects assembled are discarded. Also with the variable loadcases set to one the max combination will include only positive load effects, all negative load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	0.8	0.7	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects for number of negative load effects specified by the number of Variable loadcases to consider the remaining negative load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are discarded. The loadcases used are the most adverse, for example the most negative for min combination and all other load effects assembled are discarded. Also with the variable loadcases set to one the min combination will include only negative load effects, all positive load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Used
Live load 1	-20	0.7	0.8	0.7 + 0.8	-20 x 1.5 = -30	Used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Not used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
				Smart combination (Min) =	**-51**	

Appendix B : LUSAS Solver Trouble Shooting

LUSAS Solver Troubleshooting

During an analysis, warning and error messages may appear in the LUSAS Solver output file. An error message will terminate the solution immediately. A warning message will attempt to continue the analysis. The most common warning and error messages are:

- **Negative Jacobian** (Error)
- **Diagonal Decay** (Warning)
- **Small Pivot** (Warning)
- **Negative Pivot** (Warning)
- **Zero Pivot** (Error)

A description of these Warning and Error messages follows.

Negative Jacobian Errors

A Jacobian determinant is a measure used to give an accurate value of the current area or volume of an element. A magnitude of less than or equal to zero will automatically invoke this message and may be a result of one of the following:

- **Incorrect definition of the 2D continuum and plate elements**

  By design, LUSAS requires these element types to have an anti-clockwise node numbering sequence. The order is controlled from the underlying surface feature in which the element resides. If this message is output, the solution is to **reverse the ordering** of the surfaces for the elements having these warning messages output. Do this in LUSAS Modeller with the **Geometry**> **Surface**> **Reverse** menu item.

  Note that, in LUSAS Modeller, the **local axis system of the surface may be viewed** prior to tabulation - the xy axis system displayed on each surface represents a right handed axis system, from which the anti-clockwise (or positive qz definition may be checked).

- **Too large a loading increment causing massive deformation of one or more elements**

  This means that the elements are inverting. Note that this is only applicable for nonlinear analyses

Diagonal Decay Warnings

The stiffness matrix is a crucial component in a finite element analysis, but it can be poorly conditioned. Poor conditioning may result in round-off error, which is a loss of accuracy in the evaluation of the terms during the reduction process of the solution. This in turn leads to inaccuracies in the predicted displacements and stresses.

LUSAS monitors the round-off error by evaluating the amount of diagonal decay present during the Gaussian reduction process. This criterion is based on the assumption that initially large diagonal terms accumulate errors proportional to their size. As reduction progresses, the diagonal term is reduced, amplifying the errors until they become a maximum when the diagonal term is the pivot. An indication of probable errors may be obtained by examining the change in magnitude of the diagonal term.

The tolerance threshold above which a diagonal decay warning is output is actually quite conservative (controlled by a system variable DECAYL, default = 0.1E5). Although a check would always be recommended for any Warning of this description, significant effects are not generally expected until the decay reaches a value of 0.1E8 or greater.

Poor conditioning of the stiffness matrix occurs because of large variations in the magnitude of diagonal stiffness terms and may be due to:

- **Large stiff elements being connected to small less stiff elements**. An example may be where a stiff beam element is being used to transfer load into the structure. The stiffness of the beam would need to be reduced - typically, the beam would only need to be 1000 times the stiffness of the local elements.
- **Elements with highly disparate stiffnesses**, e.g. a beam element may have a bending stiffness that is orders of magnitude less than it's axial stiffness.

  For instance, the cantilever beam problem is notoriously problematic with respect to ill-conditioning because of the potential for large differences between the axial and shear/rotational stiffness components. A typical stiffness matrix might be

$$K = \begin{bmatrix} EA/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 12EI/L^3 \end{bmatrix} \quad \begin{array}{c} (u) \\ (v) \\ (\theta) \end{array}$$

The longer the beam, the greater the difference between EA/L and $12EI/L^3$.

Potential data input mistakes leading to poor conditioning

Poor conditioning may be as a result of deliberate modelling strategy but, more usually, an error in one or more of the following data input areas:

- **Mesh description** - typical mistakes:

    - The aspect ratio of some elements are greater than the recommended limits (see the corresponding element section in the *Element Reference Manual* for further information). An ideal value is 1:1. however, values up to 1:10 are reasonable. Depending on the results required, this value may be increased still further (a test run would be recommended first). This problem is indicated by the WARNING message: **"Unreasonably distorted element... "** The only exception are explicit dynamic elements which really do require aspect ratios of 1:1.
    - Some element shapes are too distorted (see the corresponding element section in the *Element Reference Manual* for further information).

- **Geometric properties -** typical mistakes:

    - Omission of values for any shear area parameters in the geometric properties for beams
    - Omission of values for other important properties, such as the torsional constant or thickness
    - Defining incompatible 1st and 2nd moment section properties for beams

- **Material properties** - typical mistakes:

    - Different units used to define the nodal coordinates and the material properties.
    - Incorrect nonlinear material parameters (yield stress and hardening values particularly)
    - Inconsistent units throughout the model. This would only be of concern for dynamic analyses, where SI units are recommended.
    - Incorrect definition of orthotropic properties. The inequalities given in the appropriate element section of the theory manual need to be adhered to. Numerical instabilities may result when the material characterisations approach their limits (see **Notes on material properties orthotropic** for a list of these inequalities).

- **Support nodes -** typical mistakes:

    - The structure has not been restrained against free body translation and rotation.

Each of the above suggestions are of interest because they make a contribution to the stiffness matrix.

A further possibility is that the LUSAS Modeller model geometry is invalid because the element mesh contains gaps or has discontinuities in the connection of the elements. Such modelling problems may be found in LUSAS Modeller by:

- Using the Mesh layer to view only the outline of the mesh. The view will show lines wherever a discontinuity occurs.
- Using the Labels layer to draw the node numbers onto the mesh to see if any node numbering is overwriting at any point (indicating two nodes at the same point). Correction would normally require either a **merging** or an **equivalencing** operation.

The diagonal decay message is closely related to the small pivot WARNING message (see below). See also the additional notes in the *Theory Manual* regarding the Gaussian solution method.

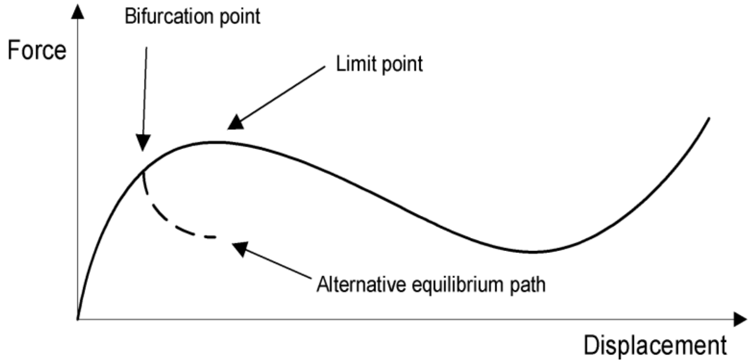
Small Pivot Warnings

See the section titled **Diagonal decay warnings**.

Negative Pivot Warnings And Errors

A negative pivot could be the result of poor conditioning so make sure you have seen the section titled **Diagonal decay warnings**. However, a well conditioned stiffness matrix can produce a negative pivot if:

- ❑ **The system is unstable**- an unstable structure could be passing through a bifurcation or limit point, as shown in the following diagram:



Such a bifurcation point could permit another, non-physical, solution path to be followed because, numerically, it requires less energy.

Every negative pivot warning occurring in the LUSAS output file represents a bifurcation point. A negative `CSTIF` value, together with a negative `PIVMIN` value corresponds to a limit point but a positive `CSTIF` and a negative `PIVMIN` correspond to a bifurcation point (although this is only the first one located in each case since limit points are detected by a **change** in sign of the slope of the force displacement curve). See **The nonlinear logfile**.

A negative pivots sometimes occurs during the iterative solution (indicating that the load step may be too big) but disappear when the solution has converged. If negative pivots occur and the solution will not converge then first try reducing the load step.

If the solution still does not converge, a limit or bifurcation point may have been encountered and the solution procedure may need to be changed. Running the problem under arc-length control gives the best chance of negotiating a limit or bifurcation point. A load limit point can also be overcome by using prescribed displacement loading.

- ❑ **The system is not adequately restrained** - for example when using a 3D beam in a 2D analysis.
- ❑ **Mechanism has been excited** - This is a further possibility when reduced integration is used. The use of Option 18 will normally solve this problem. If the problem persists, continue with the use of the option but refine the mesh further.

A count of the number of negative pivots is given in the LUSAS log file (parameter `NSCH`). Initially `NSCH` = 0 since, initially, a stable path is assumed. When `NSCH` = 1, an unstable point (limit or bifurcation) has been reached, `PIVMN` will give the value of the minimum pivot at this point.

Notes

- The use of LUSAS Option 62 is not recommended until all other checks have been carried out to ensure model integrity.
- Before modifying the solution procedure to arc-length, the checklist given in the section above on small pivots should be checked.

Zero Pivot Errors

LUSAS uses a Gaussian reduction solution technique to solve the finite element equations. This technique requires the structure stiffness matrix to be non-singular. This means that for static analyses the structure, or any components of the structure, must not permit any rigid body displacements or rotations. Failure to comply with this criterion will result in a zero pivot message.

The error message includes the node and variable number that may be affected by the poor conditioning - these variables should be investigated in the model. Typical mistakes can include:

- Omission of a support condition in one or more of the rigid body directions for the structure.

- Insufficient additional restraint when connecting a beam element to a continuum element. In this case a rigid body torsional spin about the axis of the beam may occur.
- Six degrees of freedom have been specified for a thick shell element, but the drilling rotation has not been correspondingly restrained.
- Insufficiently large slideline interface stiffness coefficients allowing the two bodies to pass through each other as rigid bodies. The load increment may also be too large.
- Incorrect nonlinear material parameters, such as a zero yield stress.
- Joint elements may require investigation as the stiffnesses operate in local directions and can be easily defined incorrectly - as a result, the joint stiffnesses will not be providing support in the required directions.
- There may be totally or partially unconnected elements within the structure as a result of incomplete merging or equivalencing of the model.

Other Warnings

Other warnings that may be found in the LUSAS output file include:

- ❑ **Aspect ratios warnings** - See the appendix on element restrictions in the *Element Reference Manual* for more information.
- ❑ **Excessive curvature for beams warnings** - See the appendix on element restrictions in the *Element Reference Manual* for more information.

Notes On Orthotropic Material Properties

For orthotropic material models the D matrix must be symmetric and a number of further relations must also be satisfied:

Material properties orthotropic (e.g., QPM4)

To maintain symmetry:

$$nyx = nxy * Ey/Ex$$

and to obtain a valid material:

$$nxy < (Ex/Ey)1/2$$

This applies to Fourier elements as a special case to simulate a bladed structure.

Material properties orthotropic plane strain (e.g. QPN4).

$$Ey * (nxy*Ez + nyz*nxz*Ex) = Ex * (nxy*Ez + nxz*nyz*Ey)$$

Material properties orthotropic axisymmetric (e.g. QAX4).

To maintain symmetry:

$$nyx = nxy * Ey/Ex$$

$$nzx = nzx * Ez/Ex$$

$$nzy = nyz * Ez/Ey$$

and to obtain a valid material:

nxy < (Ex/Ey)½

nxz < (Ex/Ez)½

nyz < (Ey/Ez)½

Material properties orthotropic solid (e.g. HX8, QSL8).

To maintain symmetry:

nyx = nxy * Ey/Ex

nzx = nxz * Ex/Ez

nzy = nyz * Ez/Ey

and to obtain a valid material:

nxy < (Ex/Ey)½

nxz < (Ex/Ez)½

nyz < (Ey/Ez)½

Material properties orthotropic thick (e.g. QSC4).

To maintain symmetry:

nyx = nxy * Ey/Ex

and to obtain a valid material:

nxy < (Ex/Ey)½

Notes

- Option 16 can be used to override non-convergence as a result of poor conditioning. When Option 16 is specified and an increment has failed to converge within the maximum number of iterations allowed, LUSAS assumes convergence, writes the output/plot file results, and then continues with the next increment. Load step reductions can also be suppressed via the STEP REDUCTION section under the NONLINEAR CONTROL data chapter. Using these procedures may help to locate the source of the problem when investigating an unconverged configuration in the LUSAS Modeller post-processor.
- A pivot refers to the diagonal element of the upper triangular matrix that is formed **after** elimination has been completed. Note that in the frontal solution these pivots are computed as soon as all the relevant equations have been assembled.
- Computation of det(K) as part of a nonlinear solution scheme is not necessary since a count of the number of negative pivots (`NSCH` in the log file) together with the value of `PIVMN` gives all the information required.
- A zero pivot implies that det(K)=0.
- If `NSCH`=2 then another unstable point has been reached and implies that det(K)>0.

Eigenvalue Analysis: Troubleshooting

It is good practice to perform a linear static analysis prior to the eigenvalue analysis. This eliminates the added complexities of the dynamic variables and will enable a check on the basic stiffness matrix for the structure. Any warning or error messages in the LUSAS output file (such as zero, negative or small pivots) should be investigated.

In the event of problems occurring after completing the linear static and the eigenvalue analysis consider some of the more common queries and their typical solutions as listed below.

Eigenvalues are missing

A Sturm sequence check is performed by default to indicate the number of modes which may be missing.

- **Check for Warnings or Errors** Check the output file for any other warnings or errors. There may be diagonal decay or pivot warnings that will indicate node and element numbers and help to identify any suspect areas of the mesh.
- **Iteration Vectors** Increase the number of starting iteration vectors. If any of the modes are close together, the default magnitude for this parameter may not be sufficient to allow accurate resolution in their extraction.
- **Convergence Tolerance** Tighten the convergence tolerance since, again, some modes may be close together. This would normally also require an increase in the number of iterations permitted.
- **Convergence Achieved** Ensure the solution converged correctly. If not, then increase the number of iterations permitted.
- **Mesh Refinement** Increase the mesh refinement of the model in order to increase the number of degrees of freedom in the structure to simulate all the modes expected.
- **Symmetry** Taking advantage of symmetry in an eigenvalue analysis may cause the inadvertent omission of several eigenvalues as a result of the corresponding symmetry supports restraining certain non-symmetric eigenmodes.
- **Increase Shift** If a shift has been used to eliminate rigid body motions when analysing unsupported structures, then it may be that the value used is insufficient. The solution is typically not overly sensitive to changes in this parameter and, therefore, any changes tried should be in terms of orders of magnitude.
- **Constraint Equations** If constraint equations have been defined in the problem the Sturm sequence check may prove unreliable. This is a limitation of the Lagrange multiplier technique used in LUSAS.

The Solution Did Not Converge

- **Check for Warnings or Errors** Check the output file for any other warnings or errors. There may be diagonal decay or pivot warnings that will contain node and element numbers, and help identify any suspect areas of the mesh.

- **Iteration Vectors** Increase the number of starting iteration vectors. If any of the modes are close together, the default magnitude for this parameter may not be sufficient to allow accurate resolution in their extraction. Increasing this parameter is also essential if only requesting a small number of eigenvalues (1-2). Ten iteration vectors would be a reasonable starting value for such a situation. This parameter is not used for Lanczos extraction.
- **Convergence Tolerance** The convergence tolerance criteria may be too tight - try slackening this criteria. This would normally also require an increase in the number of iterations permitted.
- **Increase Shift** If a shift has been used to eliminate rigid body problems when analysing unsupported structures, then it may be that the value used is not sufficient. The solution is typically not overly sensitive to changes in this parameter and, therefore, any changes tried may be in terms of orders of magnitude.

Negative Eigenvalues Are Calculated

- **Check for Warnings or Errors** Check the output file for any other warnings or errors. There may be diagonal decay or pivot warnings that will indicate node and element numbers and help to identify any suspect areas of the mesh.
- **Iteration Vectors** Increase the number of starting iteration vectors. If any of the modes are close together the default magnitude for this parameter may not be sufficient to allow accurate resolution in their extraction. Increasing this parameter is also essential if only requesting a small number of eigenvalues (1-2). Ten iteration vectors would be a reasonable starting value for such a situation.
- **Convergence Achieved** Ensure that the solution converged correctly. If not, then increase the number of iterations permitted.
- **Convergence Tolerance** Tighten the convergence tolerance, since some modes may be close together and require greater numerical resolution. This may also require an increase in the number of iterations permitted.
- **Reduce Load Level** Reduce the load applied to ensure that it is below the lowest expected buckling mode of the structure. A negative eigenvalue in a buckling analysis could also simply mean that the applied loading is in the opposite direction to that which would cause buckling; e.g. a strut subjected to tensile load instead of compression.
- **Alternative Buckling** QSL8 elements have given negative eigenvalues for thin structures and resulted in negative projected mass errors. The remedy is to use the alternative buckling algorithm where positive eigenvalues should be obtained. This is always the case except when the buckling load factor is less than unity. Adjust the load level to ensure that all the load factors are greater than 1 if this occurs. Additionally, use Option 18 (fine integration rule for the element) to overcome the excitation of any element mechanisms.

Why Loading is Ignored in an Eigenvalue Analysis

In a standard eigenvalue analysis, the loading will be ignored completely. For a linear analysis, once the eigen-pairs have been obtained, the stress distribution s for each mode shape F is evaluated using:

> s = D * B * F
>
> where D and B are the elastic constitutive and strain-displacement matrices respectively.

The reason that the forces have no effect on the vibrational behaviour is because the loading conditions in a linear analysis do not affect the D or B matrices. To include their effects, a nonlinear D and B matrix must be evaluated prior to the eigen analysis. This is achieved by performing a static nonlinear analysis (with a geometrically nonlinear option) followed by an eigenvalue analysis. In this way the effects of the loads are included via the updated B matrix (and D matrix if a nonlinear material has been specified).

Appendix C : Keyboard Shortcuts

Keyboard Shortcuts

Selecting Model Features

Features displayed in the View window may be selected using either specific cursors or by using normal cursor mode in conjunction with specific keys:

Feature / Mesh Object Selection Options

Hold the key shown down when using the left mouse button.

Specific cursor	= Normal cursor + key
All geometry selection	= + **G** key
Point selection	= + **P** key
Line selection	= + **L** key
Surface selection	= + **S** key
Volume selection	= + **V** key
Mesh selection	= + **M** key
Node selection	= + **N** key
Edge selection	= + **B** key

Face selection = + **F** key

Element selection = + **E** key

Annotation selection = + **A** key

Note. Key shortcuts can be used to override specific cursor selections.

Area Selection Options

Rectangular, circular, or polygonal areas can be selected by using specific area toolbar buttons or by using normal cursor mode with a specific key:

Specific cursor	= Normal cursor + key
Click and drag the cursor to the opposite diagonal corner.	=
Click the centre of the circle and drag the cursor to the required radius.	= + **C** key
Click each corner of a polygon and either double click to close the polygon or select Close Polygon from the context menu.	= + **X** key

Memory shortcuts

After features have been selected:

Ctrl + M key = Set selected items into Selection Memory

Selection Modifiers for All Cursors

Features displayed in the View window may be added to, or removed from, any initial selection using these selection modifers:

Shift key = Add to current selection

Ctrl key = Toggle (include /exclude selection)

Ctrl + Shift key = Remove from current selection

Tab key = Cycle (items at the same location)

Shift + Tab key = Cycle previous

Alt key = Intersect mode. By default all items completely enclosed in a selected area will be selected. By holding down the **Alt** key, items intersecting the selection perimeter will also be selected. The **Alt** key may be used with, or independently from, the **Shift** or **Ctrl** keys. The **Alt** key can also be used with feature selection shortcuts e.g. **Alt + Shft + L** adds lines to the current selection.

Alt + Return key = Display properties of item

'Datatip' + **Return** key = Adds current item to selection

Mapping of Keyboard Modifiers for Cursor and Area Selection

The Cursor drop-down menu contains a **Keyboard Mapping...** menu item which allows the default Windows-based cursor and area selection keyboard modifiers to be changed to suit those that are generally used with CAD modelling systems or be user-defined. For more information see **Selection of model features**.

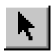
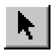
Browsing Selected Features

Items in the current selection may be viewed in the Browse Selection window which can be displayed from the **View> Browse Selection** menu item. This window can also be triggered by a right mouse button click in the **Selected** area of the status bar at the bottom of the Modeller user interface, or by right-clicking in a View window. Selected features can be deselected and reselected as necessary from those listed.

Model Viewing Shortcuts

The model can be rotated, zoomed, panned, and viewed at predefined orthogonal and non-orthogonal views using specific cursors or view buttons, or by using normal cursor mode in conjunction with specific keys. Rotation, zoom and pan can also be carried out in other cursor input modes such as when defining lines by cursor or section slicing for example.

Dynamic Pan (Drag)

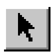
Specific cursor	= Normal cursor + key
 Hold down the left mouse button to pan the model.	=  + **D** key =  + **Middle** mouse button

Note. Hold down the key(s) to restrain the pan for either specific or normal cursor mode about the axis stated:

X or **Shift** key = Restrain in the screen X axis

Y or **Ctrl** key = Restrain in the screen Y axis

Dynamic Rotation

Specific cursor	= Normal cursor + key
 Rotates the model around various multiple axes.	=  + **D** key =  + **Middle** + **Left** or **Right** mouse

button

Note. The model is rotated about its centre unless any part of the model is selected in which case the model is rotated about the centre of the selection. Hold down the key(s) to restrain rotation about the axis stated:

X or **Shift** key = Restrain in the screen X axis

Y or **Ctrl** + **Shift** keys = Restrain in the screen Y axis

Z or **Ctrl** key = Restrain in the screen Z axis

Dynamic Zoom

Specific cursor	= Normal cursor + key
Hold down the left mouse button and move the mouse.	= + **Scroll mouse wheel** = + **Z** key

Note. If any part of the model is selected it is used as the centre of the zoom.

Zoom

Drag a box around the region to be enlarged or click the left mouse button to zoom in progressively with each click.

Ctrl key = Zoom out (when held at same time)

Note. The cursor position dictates the centre of the zoom.

Orthogonal Model Views

These buttons are located in the status bar.

= View along the +X axis towards the origin

= View along the +Y axis towards the origin

= View along the +Z axis towards the origin

Any of the views along these axes can be modified by using these key sequences which select an alternative view orientation:

Shift key + = View along the -X axis towards the origin

Ctrl key + = View along the -Y axis towards the origin

Ctrl + Shift key + = View along the -Z axis towards the origin

Equivalent toolbar buttons for these view shortcuts can be found on the customisable toolbar dialog or by clicking the right mouse button on the orthogonal model view buttons.

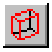
Non-Orthogonal Model Views

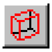
 = Isometric view

Ctrl key +  = Reverse isometric view

 = Dimetric view

Ctrl key +  = Reverse dimetric view

 = Trimetric view

Ctrl key +  = Reverse trimetric view

Useful Windows Shortcuts

The following standard Windows shortcuts are useful when creating and printing models in LUSAS Modeller:

Ctrl key + **N** key = New

Ctrl key + **O** key = Open

Ctrl key + **S** key = Save

Ctrl key + **P** key = Print

Ctrl key + **A** key = Select all items

Ctrl key + **C** key = Copy

Ctrl key + **X** key = Cut (for text only)

Ctrl key + **V** key = Paste

Ctrl key + **Z** key = Undo

F2 key = Rename (when name is selected)

F5 key = Redraw

Appendix D : Tip of the Day

Tip of the Day

When starting LUSAS Modeller useful tips can be optionally displayed. This is done by selecting the **Help > Tip of the Day...** menu item and ensuring that **Show tips at Startup** is selected. Next and Previous tips can be browsed.

The following is the list of tips supplied in the current release version:

- If you haven't already done so, please read the 'Getting Started' leaflet.
- The 'Keyboard Shortcut Guide' will help you use LUSAS in an efficient manner
- If you lose unsaved edits because of a power, hardware or software failure, restart LUSAS, open the model again and chose 'yes' when prompted to recover.
- Saving your work frequently prevents data loss and makes undo/redo work faster.
- Help is available even for greyed out buttons and menus. First click on the Help button on the main toolbar, then click on any menu item or toolbar button.
- Right-clicking in a window displays a context menu relevant to both that window and any selection within it.
- Toolbar buttons with a small triangle to the right hand side are menu buttons. Press the left hand side to use the top button, or the right hand side to display other buttons.Holding down the Shift key while selecting will add to the current selection.
- Holding down the Ctrl key while selecting will toggle an object's selection state.
- Holding down the Alt key while selecting will additionally select items that intersect (cross) the selection perimeter.
- Ctrl-A selects everything visible in the current window.
- You may filter your selection to include only Volumes, Surfaces, Lines, Points, Elements, Nodes or Annotation by respectively holding down the V, S, L, P, E, N or A key while selecting.
- Alt-Enter displays the properties of the current selection.
- If you the hover the cursor over an object, a data tip will appear providing details of that object.
- Pressing Enter whilst a data tip is showing will select the object described.

- To Zoom, Drag or Rotate the model hold the Z, D or R key down whilst moving the mouse.
- Using the dynamic rotation while holding down the Ctrl key rotates the model in the plane of the screen.
- Using the dynamic rotation while holding down the Shift key rotates the model about the screen X axis.
- Using the dynamic rotation while holding down the Ctrl & Shift keys rotates the model about the screen Y axis.
- Using the drag tool while holding down the Ctrl key drags the model up and down.
- Using the drag tool while holding down the Shift key drags the model left and right.
- If you have a 3 button wheel mouse, the wheel will zoom, holding the middle button will drag and holding the middle button and either of the other two buttons will rotate.
- To change the rotation increment, visit the 'View' tab on the View Properties dialog (right click in the current view)
- To change the style of the XYZ axes arrows, visit the 'View Axes' tab on the View Properties dialog (right click in the current view)
- To change the selection or background colours, visit the 'General' tab on the View Properties dialog (right click in the current view)
- Most context menu functionality for attributes is also available for groups of attributes. This helps you check your model.
- Dragging an attribute from the treeview and dropping it on a window assigns it to anything selected in that window.
- Dragging and dropping layers in the layers treeview changes the order in which they are drawn. This is useful if, for example, your solid model is eclipsing your labels.
- Dragging and dropping loadcases in the Analyses treeview changes the order in which they are analysed.
- Dragging and dropping controls, attributes, or groups of attributes, in the Analyses treeview assigns them to a different loadcase.
- The 'Selected' box in the status bar shows the number of objects are currently selected.
- Pressing 'Tab' or clicking in the 'Selected' box in the status bar cycles through all the objects which could have been selected at the last mouse click.
- Right-clicking in the 'Selected' box in the status bar allows access to a menu of selected objects. Choose 'Previous' when you have cycled the selection once too many!
- The 'browse selection' window gives useful feedback on which items are currently selected.
- The 'browse cyclable items' window shows all items which could possibly be selected at the chosen position.
- To select an item with a known name, use the 'Advanced Selection' dialog.
- To show the location of a named item, select it using 'Advanced Selection', right click on it in the 'Browse Selection' window, and chose 'find' from the menu.
- If a message in the text output window refers to a particular object by name, double click on the message for help finding that object.

- Clicking in the 'Z' box in the status bar views the model from the positive end of the Z axis - and similarly with X and Y. (Hold down Ctrl while clicking to view from the reverse side).
- Clicking the 'advanced' button on the LPI command bar lets you define macros for frequently used commands
- The members of a group can be examined from the group's property page
- The 'jump to' button on the 'hierarchy' tab of an object's property page can be used to select and show the properties of connected objects.
- The 'assigned in active loadcase' option on an object's property page allows you to view either assignments in the active loadcase, or in any loadcase.
- Treeviews can be moved to different tree frames by drag and drop. This is useful if you prefer to see more than one treeview at the same time.
- Attributes can be assigned to the contents of groups by copying the attribute and pasting it onto a group in the treeview.
- By saving a view using the Window menu, and naming it 'default', every new window will use the saved view.
- Pressing the Ctrl and Break keys together will interrupt the current process.
- For larger models save time by setting the manual redraw option (Window menu) and manually redraw (F5 key) at any time.
- For larger models save time by setting the manual resize option (Window menu) and manually resize at any time.
- For larger models save time by locking the mesh whilst making several geometry or mesh changes. Either manually remesh, or unlock once finished to reinstate automatic meshing
- It is usually much quicker to undo several events all at once than to undo them individually
- Many additional toolbar buttons are available via the View>Toolbars>Customise menu item
- Selection is possible using several different criteria (e.g. connectivity, element type) - see the 'Advanced Selection' dialog
- To retain a record of the commands used in a session, use the File>Script>Start Recording menu item.
- You may use the Esc key to close any dialog.
- Holding down the Shift and Control keys while selecting will remove from the current selection. For example, the current selection could be trimmed from a perpendicular view orientation to achieve a 3D selection.
- A selection may be filtered to include only Geometry, Volumes, Surfaces, Lines, Points, Mesh, Elements, Nodes or Annotation by respectively holding down the G, V, S, L, P, M, E, N or A key while selecting.
- Element faces may be selected by holding down the F key while selecting (LUSAS HPM users only)
- LUSAS Solver can be paused by pressing the Pause key on your keyboard to temporarily free up PC resources if required. Press the Esc key to resume Solver.

- Errors and warnings reported in the Text Output window can be double-clicked to open the Identify Object facility which can be used to locate the referenced items.

Editing and adding your own tips

You can add your own tips by editing the text file named **tips.txt** which is held in the <LUSAS installation folder>\Programs\Config directory.

Appendix E : Real Numbers and Expressions in LUSAS

Input and Output of Real Numbers in LUSAS

The precision of user-entered real numbers is preserved both internally within LUSAS Modeller, and when re-displayed on dialogs.

All real numbers are displayed in engineering-style notation (that is in 3,6,9 etc, powers of 10; as in 89.6E3) rather than 8.96E4 or 0.896E5. This applies to all text entry fields, including the grids used in many places throughout the Modeller user interface.

For cosmetic numbers that are system generated, like the current zoom factor, only a sensible number of significant figures is shown. This does not alter the precision stored, which will always be the maximum allowed by the operating system.

Expressions and Functions Supported

Expressions may be entered anywhere in LUSAS Modeller where numbers may be input. For example, in point definition, it is possible to enter **3+10**, **4*6**, and **5-1** as valid co-ordinates. All arithmetic operators including braces are available, as well as the standard trigonometry functions sin, cos, log etc. This facility is available in all text entry fields throughout the Modeller user interface.

The following functions are also supported:

Arithmetic

```
(A), -A, A+B, A-B, A*B, A/B, A^B, ceil(A), floor(A), abs(A),
max(A, B), min(A, B), pow(val, exp), mod(val, div)
```

Trigonometric

sin, cos, tan, asin, acos, atan, sinh, cosh, tanh, sind, cosd, tand, asind, acosd, atand, atan2, atan2d

Mathematical

exp, log, log10, sqrt

Logical

A > B, A < B, A = B, not(A), and(A, B), or(A, B), boolEq(A, B), boolNE(A, B), gt(A, B), ge(A, B), lt(A, B), le(A, B), eq(A, B), ne(A, B), if (condition, then, else)

Other

Radians(angle) Converts an angle entered in degrees into an angle in radians

Degrees(angle) Converts an angle entered in radians into an angle in degrees

Appendix F : Treeview Icons

Treeview Icons

The **Tree Frame** has six tabbed panels each with its own **Treeview**. The Treeviews provide access to model and results data and show a list of objects of a particular type. The Treeview panels comprise:

- **Layers** 
- **Groups** 
- **Attributes** 
- **Analyses** 
- **Utilities** 
- **Reports** 

Whilst the icons seen in the various Treeview panels are generally described in relevant sections of the Modeller User Manual for completeness they are also summarised here.

Layers Treeview

A symbol adjacent to each layer name in the Layers  Treeview shows the display status of each layer:

 The display of a layer has been turned 'on' (coloured layers image)

 The display of a layer has been turned 'off' (greyed-out layers image)

 No results are loaded or currently available for this layer, or inappropriate settings have been defined (red circle with a line through it)

 View properties: options and settings

Groups Treeview

A symbol adjacent to each group name in the Groups Treeview shows the visibility and status of each group.

When modelling:

A black dot next to a symbol denotes the current group into which all new geometry will be added when created.

All of the objects in this group are visible (green tick)

Some of the objects in this group are visible (blue tick)

None of the objects in this group are visible (red cross)

When a results file is loaded:

All of the objects in this group are showing results (green tick, green border)

Some of the objects in this group are visible and some are showing results (blue tick, blue border)

All of the objects in this group are visible but only some are showing results (green tick, blue border)

Attributes Treeview

A symbol adjacent to each attribute in the Attributes Treeview shows the status of each attribute present.

An attribute has been assigned to the model or used in the definition of another attribute (coloured image)

An attribute has yet to be assigned to the model or used in the definition of another attribute (greyed-out image)

Active fibre in a structural analysis, or active lamina in a composite analysis

An attribute has been set as default, meaning it will automatically be assigned to features of the model as they are generated. (coloured or greyed-out attribute with a surrounding red box)

If any attributes that were previously created by a Multiple tendon Prestress Wizard are edited outside of the wizard, or some other aspect of the model has been changed (for example the geometry or mesh) then the attributes will be marked with a warning symbol to show that the wizard must be re-run to recalculate new values.

Influence assignments are pending

Direction definition options

Analyses Treeview

A symbol adjacent to each analysis or loadcase related entry in the Analyses Treeview shows the status of each entry.

A 'base' analysis on which any other analyses present may be dependent upon (green 'solve' symbol)

An analysis that may be dependent upon a 'base' analysis (cyan 'solve' symbol)

A black dot on an analysis icon (green or cyan) indicates the analysis contains the active loadcase

Load curves

Active load curve (coloured load curve image)

Inactive load curve (greyed-out image)

Loading assignments should be moved to the load curves

Loadcases

Active loadcase (coloured loadcase image)

Inactive loadcase (greyed-out image)

Active loadcase with gravity assigned as a property of the loadcase (coloured loadcase image with a loading arrow)

Inactive loadcase with gravity assigned as a property of the loadcase (greyed loadcase and loading arrow)

Active influence prior to being solved

Inactive influence prior to being solved

Active loadcase has results available (coloured results contour image)

Inactive loadcase has results available (coloured results contour image)

Active loadcase has results available and gravity was assigned to the loadcase as a property of the loadcase (coloured results contour image with a loading arrow)

Inactive loadcase has results available and gravity was assigned to the loadcase as a property of the loadcase (greyed-out results contour image with a loading arrow)

Active loadcase has gravity assigned to the loadcase as a property of the loadcase but the loadcase is turned off for solving (coloured loading arrow with a red cross)

Inactive loadcase has gravity assigned to the loadcase as a property of the loadcase but the loadcase is turned off for solving (greyed-out loading arrow with a red cross)

Active loadcase does not have gravity assigned to it as a property of the loadcase and the loadcase has been turned off for solving. (red cross)

Inactive loadcase does not have gravity assigned to it as a property of the loadcase and the loadcase has been turned off for solving. (grey cross)

Combination and envelope results options; Nonlinear and Transient options; Nonlinear options

Post processing

Active load combination (coloured loadcase image with a plus symbol)

Inactive load combination (greyed-out loadcase image and plus symbol)

Active load combination containing a combination of data that warrants a warning (coloured loadcase image with a plus symbol and warning triangle). Hovering over the icon will show a datatip stating why the warning has been issued.

Inactive load combination containing a combination of data that warrants a warning (coloured loadcase image with a plus symbol and warning triangle) Hovering over the icon will show a datatip stating why the warning has been issued.

Active envelope (coloured loadcase image with an envelope symbol)

Inactive envelope (greyed-out loadcase image and an envelope symbol)

Active load envelope containing a combination of data that warrants a warning (coloured loadcase image with an envelope and warning triangle). Hovering over the icon will show a datatip stating why the warning has been issued.

Inactive load envelope containing a combination of data that warrants a warning (greyed-out loadcase, envelope and warning triangle). Hovering over the icon will show a datatip stating why the warning has been issued.

Cable tuning

Active cable tuning results loadcase (coloured cable stay image)

Inactive cable tuning results loadcase (greyed-out cable stay image)

Inappropriate settings have been defined (red circle with a line through it)

Target values

Active target values loadcase (coloured target image)

Inactive target values loadcase (greyed-out target image)

Inappropriate settings have been defined (red circle with a line through it)

Influence analysis

Influence point defined

Other

Model properties options and controls; Coupled analysis options; Solver options; Combination and Envelope options

Utilities Treeview

Symbols used in the Utilities Treeview:

Used for all entries in the Utilities Treeview

Reports Treeview

Symbols used in the Report Treeview:

Report entry

Chapter entry in a report

Image entry in a report

Index

A

B

C

F

G

H

N

O

P

R

S

T

U

V

W

Y

Z