LUSAS

Modeller
Reference Manual

Modeller Reference Manual

LUSAS Version 18.0 Issue 1

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.

LUSAS 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. Product options can be made available for use by the supply of an appropriate security key that 'unlocks' the product option, allowing immediate access to the extended facilities in your currently installed software. If your current license doesn't support these product options then please contact your LUSAS supplier to find out how you can gain access to them.

The following software options are available for all products:

- ❑ **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.
- ❑ **Nonlinear** - The Nonlinear option 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 and hygro-thermal analysis.. 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. Hygro-thermal analyses allow the determination of heat of hydration across a body of concrete, taking into account the availability of water for the exothermic reaction with time, based on the concrete mix, shape, exposed faces, insulated surfaces, environmental conditions etc.

❑ **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 three primary uses: Seismic response analysis of structures subjected to acceleration time histories of support motion; moving load analysis of structures, such as bridges, subjected to moving vehicle or train loads, where the magnitude and configuration of the loading remains constant throughout the analysis; and moving mass analysis of structures, such as bridges, subjected to moving vehicle or train loads, where spring-mass systems are used at the axle/bogie positions of a vehicle.

The following options are available for LUSAS Bridge only:

❑ **Traffic Load Optimisation** - The Traffic Load Optimisation facility complements and extends the vehicle loading capabilities of LUSAS Bridge and helps to simplify the evaluation of worst load position for various road and rail load configurations. Vehicle load optimisation is used to identify critical vehicle loading patterns on bridges and apply these loading patterns to analysis models. Rail load optimisation provides the means to define parameters for a particular design code, and generate the most critical rail track loading pattern for a selected track layout, for each influence shape under consideration. Both facilities reduce the amount of time spent generating models and leads to more efficient and economic design, assessment or load rating of bridge structures.

❑ **Steel and Composite Deck Designer** - This carries out comprehensive calculations for multiple sections on steel or steel/composite bridge decks to the Eurocodes.

❑ **Rail Track Analysis** - The Rail Track Analysis option permits track/bridge interaction analysis to the International Union of Railways Code UIC 774-3. It allows you to automatically build models from data defined in MS Excel spreadsheets, run an analysis, and quickly produce results in spreadsheet or results file formats.

The following options are available for LUSAS Bridge and LUSAS Civil & Structural only:

❑ **RC Frame Design** - Design checks due to bending with or without axial force can be carried out for regular or arbitrary shaped reinforced concrete sections at the Ultimate Limit State (ULS) and Serviceability limit states (SLS) for a specified RC design code.

❑ **Steel Frame Design** - Design code checking of steel members for a specified steel design code. Design check results are visualised as Utilisation ratios on a contour or values layer for a selected design code, and active load combination or design envelope. Results components for individual design checks can be chosen for viewing, and maximum utilisation factors can also be obtained.

What's new

When you run LUSAS Modeller you will be presented with information that links to a '**What's new**' page. Links on this page will tell you what has changed since the last major version and also cover 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.
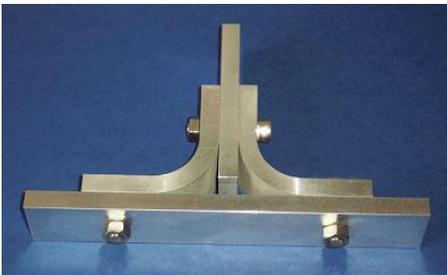
Engineering Problem   **Finite Element Model**
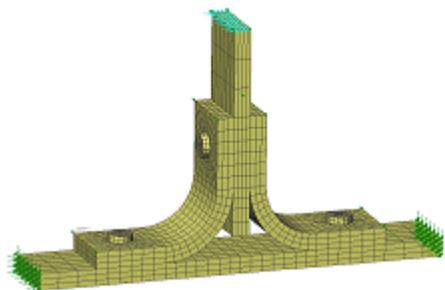
The real engineering problem responds in an infinite number of ways to external forces. The manner in which the Finite Element Model will react is given by the degrees of freedom, which are expressed at the nodes. For example, a three-dimensional solid element has three degrees of freedom at each node 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**
- ❑ **Worked Examples**
- ❑ **Application Manual (Bridge, Civil & Structural)**
- ❑ **IMDPlus User Manual**
- ❑ **Rail Track Analysis User Manual**
- ❑ **Element Reference Manual**
- ❑ **Solver Reference Manual**
- ❑ **Theory Manual (Volume 1 and 2)**
- ❑ **Verification Manual**
- ❑ **CAD Toolkit User Manual**
- ❑ **LUSAS Programmable Interface (LPI) Guides**
- ❑ **LUSAS Programmable Interface (LPI) Online Help**
- ❑ **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.

Worked Examples

- General and 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.

IMDPlus User Manual

Contains details of how to carry out multiple loading events with advanced loading conditions for three main uses:

- Seismic response analysis of structures subjected to acceleration time histories of support motion.
- Moving load analysis of structures, such as bridges, subjected to moving vehicle or train loads, where the magnitude and configuration of the loading remains constant throughout the analysis.
- Moving mass analysis of structures, such as bridges, subjected to moving vehicle or train loads, where spring-mass systems are used at the axle/bogie positions of a vehicle. The configuration of the systems remains constant throughout the analysis but as they move across the structure, the dynamic response of the unsprung and sprung masses affects the loading acting on the structure due to inertia effects.

- 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) Customisation and Developer Guides

- Provides information for users and application programmers wishing to customize the LUSAS environment or interface LUSAS with other applications.
- Available in PDF form only.

LUSAS Programmable Interface (LPI) Online Help

- Provides information for application programmers wishing to customize 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 only.

Chapter 2 : Using Modeller

Using LUSAS - An overview

Carrying out an analysis with LUSAS will involve 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'. Further details of these 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 are created by importing only those types of LUSAS model data, or third party data that are supported.

With very few exceptions the majority of LUSAS models are featured-based.

Modelling

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.

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.

Meshing attributes

Whilst points, lines, surfaces and volumes allow the geometry of the problem to be represented or imported, to solve the problem, the geometric features must be broken down into discrete 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, are created and assigned to relevant geometry features to define the type and number of nodes and elements that will be used to represent each part of the geometry.

A significant 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.

Other attributes

Other attributes are used to define aspects of the behaviour of parts of the model, or the external factors which are imposed on it. Within LUSAS Modeller, there are several types of attribute such as materials, loading and supports. Within each attribute type there may be 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 selected parts 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.

Analyses

Within a single model file any number of analyses may be defined. Each analysis will inherit most of the assignments, options and settings of what is called the base analysis for the model. 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.

Loadcases

In a linear analysis, it is convenient to gather certain types of applied loading together, if it is applied at the same. for example gravity and wind loading which are applied at the same time. A loadcase is the term for such a collection of loading. 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. Within each loadcase, the loading, supports, or type of analysis can be changed. Control parameters can be set for each loadcase.

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 settings made or entered to plot a graph, for instance, are all 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.

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 modeller for viewing and optional post-processing.

Viewing the Results

Results are available to be viewed in many different ways, and the 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**
- ❑ **Inspection locations**
- ❑ **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.**
- ❑ **Model reports**

Worked examples

A large selection of tutorial examples are available to help you to get up to speed with LUSAS and it is strongly recommended that you work your way through a selection of the examples to get a feel for the software before starting on your particular problem.

See Online help > Worked Examples for more information.

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.
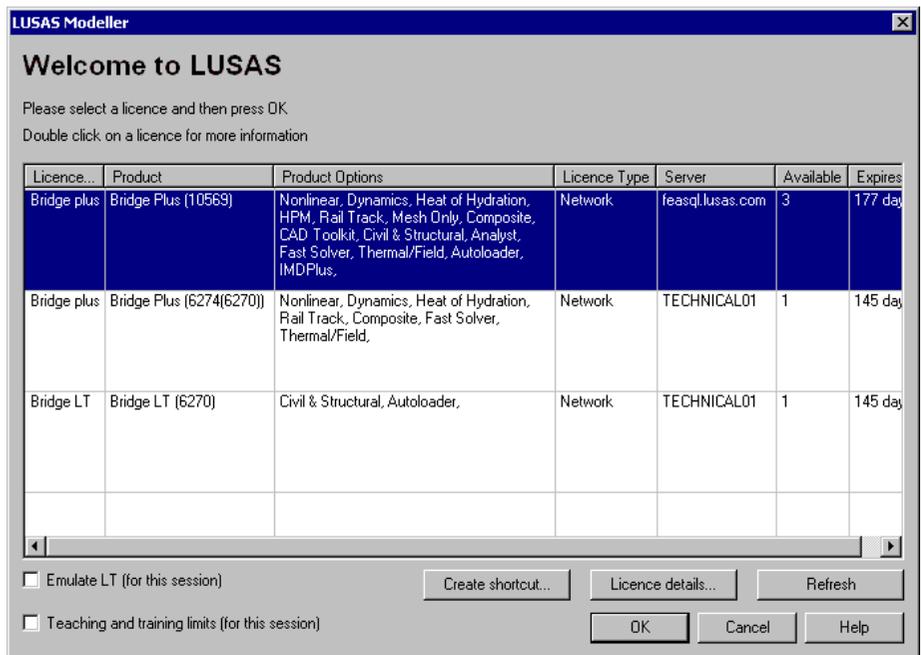
License order and usage

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.

The LUSAS Modeller User Interface

The principal regions and components of the Modeller user interface are described here.
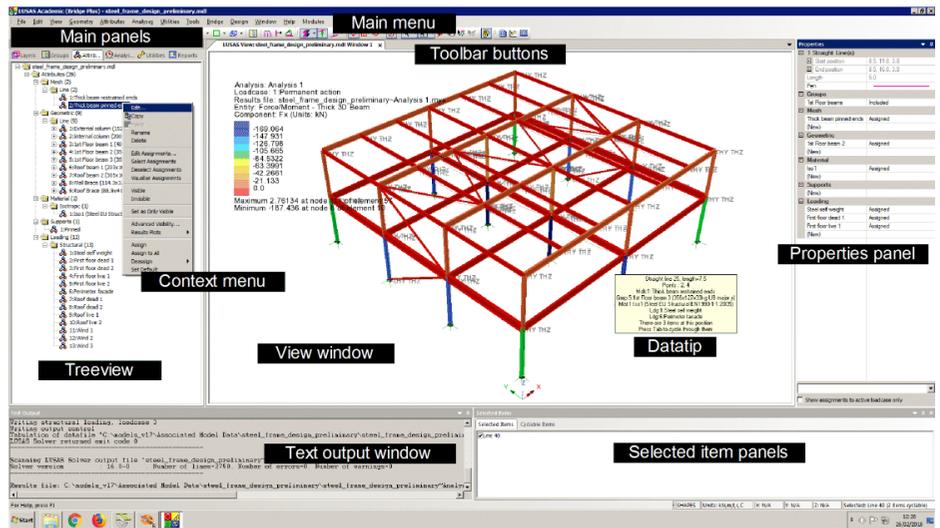
Main Menu and Toolbars

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> Toolbars menu item.

Choosing an analysis category when defining a new model modifies the user interface to generally show only those menu items, dialog options and settings that are appropriate to the analysis category stated

View window

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. Pre-defined view layers inside each window show/visualise/draw model information that can be generally manipulated in the current view window as required.

Modelling in LUSAS

A LUSAS model is graphically represented by geometry features (points, lines, surfaces, volumes) which are assigned attributes (mesh, geometric, material, support, loading etc.). Geometry is defined using a whole range of tools under the Geometry menu, or the buttons on the Toolbars. Attributes are defined from the Attributes menu. Once defined, attributes are listed in the relevant Treeview panel.

Main Panels

Six main panels each with its own **Treeview** provide access to model and results data and show a list of objects of a particular type. These panels are visible by default, but can hidden / made visible by selecting the **View > Main Panels** menu item. They are:

- ❑ **Layers** - controls the display of model data and results data to a view window. Each drawing layer is responsible for drawing one particluar aspect of this data such as geometry, mesh, and attributes layers for instance. See **Layers** for more information.

- ❑ **Groups** - used to store selected user-defined collections of objects (geometry, nodes or elements) under a collective name. See **Groups** for more information.

- ❑ **Attributes** is used to store named collections of data that may be attached (assigned) to the model geometry, such as 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. See **Attributes** for more information

- ❑ **Analyses** - shows all analyses that have been defined including loadcases, eigenvalues, results, and control chapters. Also in this treeview are post-procressing concepts such as combinations and envelopes. See **Analyses Treeview** for more information.

- ❑ **Utilities** - 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. See **Model Utilities** for more information.

- ❑ **Reports** - contains named reports, each of which is made up of multiple chapters. Chapters may be defined to view modelling information, results, screenshots and similar. See **Generating reports** for more information.

Two other main panels provide details of properties and associativity for selected objects. The panels are hidden until accessed by a selecting a feature/object and choosing the relevant context menu item for that feature/object. They can also be made visible by selecting the **View > Main Panels** menu item. The panels can be docked, and separate tabs are provided for each panel when done. Double-clicking a docked window will isolate it again. The panels are:

- ❑ **Properties** - displays the current properties of selected geometry features (points, lines, surface, volumes) and selected mesh objects (nodes, edges, elements, faces). See **Properties Panel** for more information.

- ❑ **Hierarchy** - shows the **merge** status, and lists the higher and lower order features/objects of a single selected geometry feature or mesh object. The type and feature number of the selected feature is shown. Selection of a feature

within the hierarchy panel will update the panel to show the hierarchy for that feature. The following setting and listing options are provided:

- **Is mergable** (only present for Geometry features) Sets the **merge** status.
- **Lower order** (not applicable to Points) Lists the lower order features that define the selected feature.
- **Higher order** (not applicable to Volumes) Lists the higher order features that the selected feature defines.

Manipulation of data in a Treeview

Treeviews use drag and drop functionality. For example, an attribute in the Attributes Treeview can be assigned to pre-selected model geometry by dragging the attribute into the model View window, or by copying and pasting an attribute onto another valid Treeview item as for instance, a group name, as held in the Groups Treeview.

All Treeviews (with the exception of the Layers Treeview) support multiple selection of objects in that treeview in order to carry-out a subsequent operation on the selected items, such as a deletion or a copy, for instance. Selection-specific context menus provide additional options. When selecting treeview objects the following applies:

- Left-clicking on an item clears the previous selection and sets this item as the only one selected.
- Pressing the **Ctrl** key and left-clicking on an item toggles the items selected status between on and off
- Pressing the **Shift** key and left-clicking on two items allows a range to be selected between the first item and the last one selected.
- A multiple selection is also available by clicking and dragging to define a rectangular area inside the Treeview control.
- Right-clicking following a selection displays a context menu specific to the selection made.

Only appropriate context menu options are provided for multiple selected items. For instance a 'Deassign' option may be provided because it would be valid for all objects, but an 'Assign' option will be disabled because confusion could occur over the additional data required, for example loadcase or analysis.

See **Treeview icons** for details of treeview content.

Selection Panels

These panels are not displayed by default but can be viewed by choosing the appropriate the **View> Selection Panels** menu item, or by right-clicking in a view

window and selecting the appropriate context menu item. The windows can be docked, and separate tabs are provided for each window when done. Double-clicking a docked window will isolate it again.

- ❑ **Selected Items** This panel can be viewed using the View> Selection Panels > Selection menu item. Once visible it will contain a list of all currently selected items which may then be individually deselected.
- ❑ **Selection Memory** This panel can be viewed using the View> Selection Panels > Memory menu item. Once visible it will contain a list of all items currently in Selection Memory which may then be individually deselected.
- ❑ **Cyclable Items** This panel can be viewed using the View> Selection Panels > Cyclable Items menu item. Once visible it will contain a list of all cyclable items. Where multiple items could possibly be selected by the same mouse click, cycling is the process of stepping through those items one at a time. An item can then be individually selected or deselected.
- ❑ **Visible items** This panel can be viewed using the View> Selection Panels > Visible Items menu item. Once opened it contains details of all visible items selected in the view window.

LPI Command Bar

The LUSAS Programmable Interface (LPI) Command Bar window can be added to the user interface by using the View > LPI Command Bar menu item. For advanced use, it allows typing of commands that would otherwise be chosen or selected within the Modeller user interface. See the *LUSAS Programmable Interface Customisation and Automation Guide* for more details.

Text Output Window

The Text Output Window displays messages and warnings during a modelling session. When an error message or warning relating to a particular object is written to the text output window, extra information is sometimes available by double-clicking on that line of text.

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. X, Y and Z panel entries, when clicked, will orientate the model to align with those axes. The View> Status Bar menu item may be used to hide or show the Status Bar.

Datatips and Tooltips

Datatips and Tooltips provide basic information about whatever is under the cursor. Datatips generally report on information within a Modeller view window such as

model geometry, attributes and assignments etc. Tool tips report on uses of toolbar buttons or expected input for grid cells etc.

Context Menus

Although commands can be accessed from the main menu, pressing the right-hand mouse button with an object selected usually displays a context menu which provides access to relevant operations. Treeview panel items 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. Right-clicking in most Treeview panels provides a sorting capability.

Property Dialogs

General data relating to a model are presented in property dialogs. Properties may relate to the whole model or the current window, or a single geometric feature. Most objects have properties. To view an object's properties, select it, press the right mouse button, then choose Properties from the context menu. Geometric properties of features or mesh objects are shown in a **Properties panel**.

Online Help

LUSAS contains a comprehensive online help system.

- Most dialogs include a Help button which provides information on that dialog.
- Selecting **Help > Help Topics** from the main menu provides access to all the Help files.

If the Help Contents, the Help Index and the Search facility are not shown when a help page is first displayed pressing the Show button will show these tabs in the HTML Help Window.

Treeview Panels and Icons

Treeviews

The **Tree Frame** of the **Modeller User Interface** 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.

🔵 Default group icon.

🔵 The addition of a black dot denotes the current group into which all new geometry will be added when created.

When 'Show visibility feedback in treeview' is selected the following group icons are provided:

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

✓ The addition of a black dot denotes the current group into which all new geometry will be added when created.

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

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

When 'Show visibility feedback in treeview' is selected and a results file is loaded the following icons are provided:

☑ 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)

Note that visibility feedback is intended for relatively few numbers of groups.

Attributes Treeview ⚛

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

⚛ An attribute has been assigned to the model (coloured image)

⚛ An attribute has yet to be assigned to the model or used in the definition of another attribute (grey image)

⚛ A black dot on a coloured attribute icon shows that an attribute, which has been assigned to the model, has been set active (for example setting active a local coordinate assignment).

⚛ A black dot on a grey attribute icon shows that an attribute, which has yet to be assigned to the model, has been set active.

▬ Fibre in a structural analysis, or active lamina in a composite analysis.

▬ Active fibre in a structural analysis, or active lamina in a composite analysis.

⚛ An attribute icon (coloured or grey) 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.

⊘ A 'Not valid' symbol shows that an attribute assignment is pending or invalid in some way, requiring investigation and correction.

⚠ A warning symbol will appear in the following situations:

- If any attributes that were previously created have been superceded by a change to a model. In this case a model must be re-solved to re-create those attributes..

- If attributes defined are not compatible with a stated **analysis category**.

⚛ 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

Vehicle Load Optimisation results folder

Rail Load Optimisation results folder

Onerous results table

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 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.

Active RC Frame Design loadcase (coloured loadcase image with a stylised symbol of a reinforced member)

Inactive RC Frame Design loadcase (greyed-out loadcase image with a stylised symbol of a reinforced member)

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

Various to show influence point defined.

Other

Model properties options and controls; Coupled analysis options; Solver options; Combination and Envelope options

Utilities Treeview

Icons used in the Utilities Treeview:

Graph created by the Graph Wizard Utility

Graph set to not auto-update on screen or if added to a report

Saved view

Print Results Wizard table entry

Used for all other general utility entries such as graph datasets, variation data, tendon data etc.

Reports Treeview

Icons used in the Report Treeview:

Main report entry

Model or results chapter

Image entry (chapter) - note that manual inserted image data does not automatically update each time a report is generated.

Saved view entry (chapter) - automatically updates each time a report is updated

One-click report entry (chapter)

Graph entry (chapter)

Creating a New Model

A new model is created by using the **File> New** menu item.

File name

Creating a new model requires a model file name to be entered. 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.

Working folder

A working folder must 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.

Model properties

Analysis Type

Specifying an analysis type modifies the interface for the type of analysis to be carried out and also defines the class of model to be created. Analysis type are:

- ❑ **Structural**
- ❑ **Thermal**
- ❑ **Hygro-thermal**
- ❑ **Coupled thermal/structural**
- ❑ **Coupled hygro-thermal structural**

Analysis Category

Specifying an analysis category modifies the user interface to generally show only those menu items, dialog options and settings that are appropriate to the analysis category stated. Selecting an analysis category as part of creating a new model defines which startup templates are available for selection. It also determines the initial vertical axis for the model, and the orientation of element types and library items as displayed on the Geometric Line dialog. Options available are:

- ❑ **2D Grillage/Plate**
- ❑ **2D Inplane**

❑ **2D Axisymmetric**

❑ **3D**

Models created prior to Version 16 will have an analysis category assigned to them automatically by LUSAS when they are opened in a later release (but only where a suitable analysis category can be established based upon the elements present within the model). If an analysis category cannot be assigned by LUSAS automatically (perhaps because a range of conflicting elements is defined within the model) an analysis category will need to be stated.

Basic settings and permissible uses for each analysis category are shown in the following table:

Analysis Category	Default vertical axis	Permissible freedoms	Permissible geometric usage types
2D Grillage/Plate (structural)	Z	Z, THX, THY	Grillage
2D Inplane	Y	X,Y,THZ	Bar / link, cross section beam, 2D thick beam, 2D thin beam, plane strain beam, ASPC thickness
2D Axisymmetric	Y	X,Y,THZ	Axisymmetric membrane / shell
3D	Z	X,Y,Z,THX,THY,THZ	Bar / link, cross section beam, 3D thick beam, 3D thin beam

Notes:

- A model's analysis category can be changed on the **Model Properties** dialog, but this should be used with caution since subsequent dialog choices made may not be supported in certain levels of the software, or may be inappropriate for a particular type of analysis.

- Changing the Analysis category only changes the vertical axis if it cannot remain the same because it would be invalid for the analysis category chosen.

Model units and timescale units

A set of consistent modelling units must be specified when starting a new model. Timescale units must also be chosen. Note that changing model units after they have been used does not modify the numerical value of any existing inputs. For example, for model units, an input of 1m will become 1mm (not 1000mm). Note that changing the timescale units does not modify the duration of any existing time inputs. For example, an input of 12 hours will become 0.5 days (not 12 days). Changing model units may require any previously defined material or geometric properties to be re-created and re-assigned to the model. Contact LUSAS technical support for more information.

Optional settings

Startup template

Dependent upon the analysis category selected, a selection of startup templates are listed for selection. These pre-load the Attributes ⚙ Treeview with selected mesh, geometric, support, and loading attributes, and other settings suitable for that analysis type. **User-defined startup templates** can also be created and accessed via the button alongside the Startup template drop-down list.

Layout grid

A regular or user-specified set of construction grid lines to aid with defining geometric model features can be defined as part of creating a model. See **layout grid** for more details.

Job no.

Optional text input field.

Model title

The model titles is used during an analysis and annotated to the plot border. To subsequently change the title specified use the File> Model Properties menu item.

When a model is created

A "Startup Script Files" on page 89 automatically runs after a new model is created. This allows for user-defined combinations of commands to be run to set preferred defaults, colour schemes, other preferences.

An alternative to adding combinations of commands to a pre-defined script file is to create and run your own script to make settings after a model has been created. See "Script Files" on page 89 for details.

Changing initial settings

The **Model Properties** dialog allows some of the initial settings to be changed, if appropriate, after the new model dialog has been exited. It is recommended that modelling units are not changed after the model is created.

New model creation from command line input

A new model can also be created by running LUSAS Modeller from a command line, specifying the location of a script (to set any preferences) and stating a name for the model as follows:

```
"C:\Program Files(x86)\LUSAS<version
number>\Programs\lusas_m.exe"
```

```
run=C:<folder_name>myScript.vbs
C:<folder_name>model_name.mdl
```

Model 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'.

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.

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 **Expressions and Functions Supported** 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.
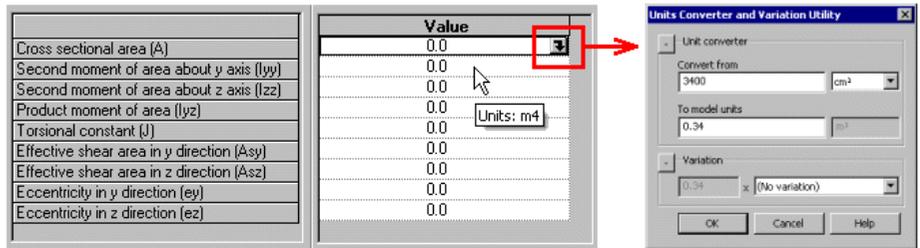
Reporting of expected model units and triggering of the Units Convertor dialog

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 model units in use

Model 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, and 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**. Changing model units may require any previously defined material or geometric properties to be re-created and re-assigned to the model. Contact LUSAS technical support for more information.

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 displayed 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.

❑ **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.

The Modeller user interface presents different menu and context menu items according to the type of model and analysis category 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 **Analyses > IMDPlus > Moving load** and **Analyses > IMDPlus > Moving Mass** menu items are not currently supported

- The **Bridge > Moving load** menu item is not currently supported.

- Prestress modelling is 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 Layers 🔲 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) Options**
- ❑ **Defaults**
- ❑ **Solver System Variables**

General

- ❑ **Title** Brief description of the current model as entered in the New Model startup dialog.

- ❑ **Analysis category** An analysis category must be chosen when a model is first created. It defines the category of analysis that will be carried out with this model. Analysis category choices are 2D Grillage/Plate, 2D Inplane, 2D Axisymmetric, and 3D. It can be changed on the Model Properties dialog, but this should be used with caution since subsequent dialog choices made may not be supported in certain levels of the software, or may be inappropriate for a particular type of analysis. Models created prior to Version 16 will have an analysis category assigned to them automatically by LUSAS when they are opened in Version 16 (but only when a suitable analysis category can be established based upon the elements present within the model). If an analysis category cannot be assigned by LUSAS automatically (perhaps because a range of now conflicting elements is defined within the model) a dialog will appear asking for an analysis category to be chosen.

- ❑ **Only show compatible options** When checked, shows only those menu items, dialog options and settings that are relevant for the stated analysis category. When unchecked, will show all dialog options and settings irrelevant of the analysis category set, and permits changing the analysis category on attribute dialogs that support this facility on a case-by-case basis. It should be used with caution.

- ❑ **Model units** Specifies the modelling units. These are used to preset the units seen on many of the units dependent dialogs, and to annotate the LUSAS output. A set of consistent modelling units must be specified when starting a new model. Timescale units must also be chosen. Note that changing model units after they have been used does not modify the numerical value of any existing inputs. Changing model units may require any previously defined material or geometric properties to be re-created and re-assigned to the model. Contact LUSAS technical support for more information.

- ❑ **Output in 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, days or increments.

- ❑ **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.

- ❑ **Vertical axis** Set automatically by setting the Analysis category for new models. Used to determine the direction that gravity loading will be applied if added using the Bridge or Civil > Gravity menu item. It is also used to define 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 changing the Analysis category (see above) only changes the vertical axis if it cannot remain the same because it would be invalid for the analysis category chosen.

- ❑ **Gravity** - Value set for the acceleration of free fall, as used when creating a Gravity loading from the Bridge of Civil menu item, and in a number of attribute definitions.

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 a Notes panel accessed via the **Utitilites > Notes** menu item. The Notes placeholder on the Model Properties dialog is retained to direct users to the new location and method of adding notes.

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 active then all subsequent geometry definition is carried out in transformed coordinates.

Advanced Geometry 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. Splitting of features is achieved by use of the **Geometry > (Feature) > By Splitting** menu items.

❑ **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. A value for Hollow volume tolerance may be set.

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. See **Creating Holes in Surfaces** for more information.

❑ **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. See **Merging and Unmerging Features** for more information.

Meshing

❑ **Equivalence** Defines the default nodal equivalence tolerance used in a equivalence attribute.

Tolerance A default value of 1e-3 is set.

Automatic (if selected) carries out equivalencing automatically for all nodes in the model and no equivalence attribute assignment is required. See **Nodal Equivalencing** for details.

❑ **Line Mesh Defaults**

- **Default line divisions** Sets the default number of mesh divisions on a line and the maximum subtended angle per element for an arc or splines.
- **Default angle subtended** If an element exceeds the max subtended angle the number of divisions on the arc or spline will be increased.

❑ **Default beam orientation method**

- **Orientate beams and columns using model vertical axis** When selected, identifies the alignment of the beams present in a supported third-party import file, or for beams within the model, and sets the beam element axes for each beam to a suitable orientation to represent either a vertical column or a horizontal beam, as typically used in modelling building frames. Unselected, the beam element axes for each beam are set to the default LUSAS orientation.

❑ **Miscellaneous**

- **Create a group of objects that failed to mesh after meshing** 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 line / surface element edges** Moves the position of the mid-side nodes on quadratic elements to lie on a straight line connecting the corner nodes.

❑ **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.

❑ **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.

❑ **Script to run after meshing** allows running of a user-defined script after meshing as, for example, if a quality check was desired to be run on mesh quality.

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).

- **Activate spring supports in undeformed location** This option is turned off by default.

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

❑ **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.

(Solution) Options

Options can be specified to be common to all analyses or be set individually for each analysis. 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.

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**.

- ❑ **Platform** If 32 and 64 bit versions of LUSAS Solver have been installed, you can specify which version of LUSAS Solver should be used with your current instance of LUSAS Modeller:

  - **Follow Modeller** - use the same version of LUSAS Solver as being used by LUSAS Modeller
  - **32 Bit** - use the 32 bit version of Solver
  - **64 bit** - use the 64 bit version of Solver

- ❑ 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.

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).

❑ **Axisymmetric about the global X-axi**s 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 (structural)** Formulate lumped mass matrix instead of consistent mass matrix for elements. (LUSAS Solver option 105).

❑ **Lumped mass matrix (thermal)** 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.

❑ **Improved transverse shear for thick shells** When selected, and only if equivalent stress is computed, a parabolic distribution of shear stress is calculated, having a mid-shell maximum value of 1.5 x $Fy_{average}$. Top and bottom shell shears will report as zero. When not checked, constant shear is obtained.

❑ **Show results for rigid zone elements** By default, the elements representing rigid ends of beams do not present element-based results when results are viewed, displayed or printed. By selecting this option these may be obtained.

❑ **Internal displacements and deformations for beams** By default this option is off. When set, it uses the LUSAS Solver calculated deformations at each internal calculation point in each beam element. Setting this option 'on' allows for plotting of displacement contours for beam elements.

❑ **Residual bending flexibility correction** When selected, improves the bending performance of thin shell (TTS3) elements.

Integration options

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

❑ **Fine integration for stiffness and mass (thermal)** 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). See the *Element Reference Manual* for more details.

❑ **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). See the *Element Reference Manual* for more details.

❑ **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). See the *Element Reference Manual* for more details.

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 object 🐝 in the Analyses 📿 Treeview. Draping options can only be specified to be common for all analyses.

Averaging options

Averaging options are accessed via the Model Properties dialog or the Model Properties control object 🐝 in the Analyses 📿 Treeview.

The Averaging Options dialog allows averaging rules to be applied to state when results averaging should and should not take place between adjacent pairs of elements in line, surface and volume features. For each pair of elements that is considered, if all the averaging rules "pass" then averaging of results across those elements is permitted.

- ❑ **Feature** Checked 'off' for all elements by default - meaning that averaging of results across each pair of elements being considered will be done if the other averaging rules chosen permit.
- ❑ **Different assignments** Checked 'on' by default - meaning do not average results across adjacent elements with different attribute assignments.
- ❑ **Common LOF assignment** Checked 'on' by default - meaning do not average results across adjacent elements in common lower-order features, such as when a line is shared by a surface, or a point is shared by two lines. Note that the assignment if a null mesh does not count as a discontinuity. For general assignments no attempt is made to check whether the assignment(s) apply in the active loadcase.
- ❑ **Visibility** Checked 'off' by default - meaning average results across invisible adjacent elements if other averaging rules chosen permit. When checked, emulates the averaging behaviour for visible / invisible features/elements from Version 15.2.
- ❑ **Junction** Checked 'on' by default - meaning do not average results at a junction (where three or more elements meet at a common lower order feature - such as where elements form a 'T' or a more complex 'framed' junction.
- ❑ **Angle exceeds** a specified number of degrees (12.0 by default). Checked 'on' by default - meaning do not average results across adjacent elements beyond that angular tolerance. A special case applies when transforming to material, local coordinate, path or global directions where for beams, only the x axis is compared, and for shells, only the z axis is compared. For other elements, and other transformations, all three axes are compared.
- ❑ **Defaults** when pressed, returns the dialog to the settings stated above.

Averaging options can only be specified to be common for all analyses. See **Results Files and Results Processing** for more information.

Defaults

Sets the defaults for Pens that are used to draw Points, Lines, Surfaces and Volumes, and symbols, arrows and text, which are used when visualising attributes. Actual settings are controlled from the **Attribute Visualisation layer** for the current window.

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, and tiled using the **Window > Tile Horizontally** or **Tile Vertically** options. Closing the only window displayed 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.

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.

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.

Saving views

A view of a model may be saved by name into the Utilities treeview using the **Window> Save View** menu item. Saved views can be applied to new or existing views, and can also be inserted into reports. See **Saved Views** for more information.

Saved Views

A view of a model may be saved by name into the Utilities Treeview using the **Window> Save View** menu item. Views saved in this way are always retained and available when a model is saved, or when a model containing them is opened in a higher version of LUSAS. Saved views can be applied to other views and can be inserted into reports.

The following settings can be optionally saved for use when the saved view is applied to an existing or new view:

❑ **Rotation** Saves the current rotation setting for the view (which consists of a vector and an origin as specified on the View tab of the View properties)

❑ **Resize to fit window / page** Forces the saved view contents to fit the view window / report page when the saved view is applied to another or new window (Working mode only), or added to a report. The initial setting of this option is governed by any setting of the Resize toolbar button. This option is not available in **Page layout** mode because page layout mode always resizes to fit page.

❑ **Zoom and pan** Saves the current zoom and pan setting in use for the view (Working mode only). This option is not available in **Page layout** mode, because zoom and pan is meaningless when page layout mode always resizes to fit page.

❑ **Projection** Saves the perspective view, if set.

❑ **Layers** Saves the window layer names (and their settings) present in the Treeview for the view.

❑ **Colours** See 'Saved View Colours' heading below.

❑ **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.

Notes

- The use of meaningful saved view names is recommended, especially for results and report uses.

- To save a view with the same name as an existing Saved view name, over-type the name on the Saved view dialog.

- A Saved view entry is identified in the Utilities treeview by this icon . The entry can be edited (to see the view settings it contains), be renamed or deleted by the means of its context menu

Uses

Updating of view windows

A Saved View entry in the Utilities treeview can be dragged and dropped onto any View window to automatically update the view settings and the display in that View window to use those settings contained in the saved view. A Saved view entry can also be assigned to a New window or the Current window by using the respective **Apply to>** context menu items. Using these methods a number of standard model or results views can be set-up for easy and quick viewing of the predefined views of the model and of selected results plots.

Reports

Saved views can be inserted into **reports** and when done will ensure that the report, when generated, will always contain up-to-date images of any model or results plots. This is because the image data contained in a saved view is re-generated each time a report is created. A saved view name is automatically used as the title for the inserted image of the view in a generated report. This title is comprised of the view name followed by a loadcase name in parenthesis. For a default view name and loadcase name the inserted image of a saved view in a report would be titled: 'View1 (Loadcase 1)'. As a result it makes sense to name a saved view explicitly according to the contents of the view, for example, 'Isometric view', or 'Maximum displacement', or 'Contours of Mz'.

Working mode (**View > Working Mode**) should be used if zoomed-in views of a model are required to be added to a report, because Page layout mode always re-sizes a model to fit a page size.

Saved View Colours

The selection of 'Colours' in the context of saving and reloading a saved view is to reinstate the following options:

- Whether to use Windows colours or manually specified background and selection colours.

- If manually specified, the choice of background and selection colours.

- The seed colours used for contouring (as shown on the contour dialog).

- Whether or not the coordinate system is visualised.

- All the display settings for the coordinate system visualisations - origin, pen, size, etc. (as shown on the view properties dialog).

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**
- ❑ Influence shape
- ❑ **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:
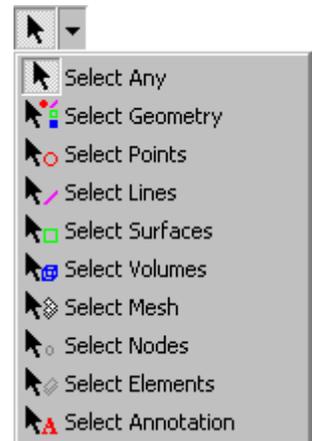
G – Geometry selection filter

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:
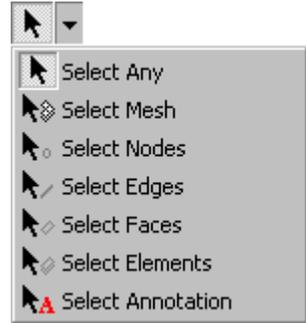
M – Mesh selection filter

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.

Planar selection filters

These can be used to define planes in which, or axes along which features can be selected. When used in isolation all features/objects in a chosen axis or plane will be selected within a cursor selection. When a planar option is set prior to using the previously mentioned Feature/Mesh Object Selection option individual points or lines or elements etc can be selected within the axis or plane chosen.

Axes and planes are defined by model **view axes** and the model **vertical axis**.

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 description of these selection filters.
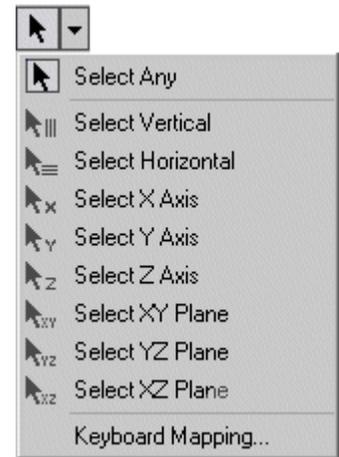
Area selections

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.

Cumulative selection

Note that cursor selection options are cumulative, so it is possible, for example, to make a rectangular selection of only all the vertical surfaces within a model. This would be achieved by selecting the Select Vertical only toolbar button, prior to either selecting the Surface selection toolbar button, or selecting the

Rectangle Select toolbar button whilst holding down the **S** key to limit the selection to just Surfaces.

Browsing Selected Features

Items in the current selection may be viewed in the Selected items panel, which can be displayed from the **View> Selected Items** menu item. This panel 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.

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 entering 1T5I2 (representing 'one to five in increments of 2') will select items numbered 1,3, and 5. Similarly, entering 4T7,10,12 will select items 'four to seven, ten, and twelve. 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 Cyclable Items panel, which is available from the **View> Selection Panels> Cyclable Items** menu, or 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.

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.

Properties panel

The Properties panel displays the current properties of any number of selected geometry features (points, lines, surface, volumes) or selected mesh objects (nodes, edges, elements, faces) and displays the attributes which are assigned to those selected items. It allows for changing the properties of selected features/objects and for assigning or deassigning attributes to and from those items.

The panel is initially hidden until accessed by a selecting any number of features/objects and choosing the **Properties** context menu item for that object. It can also be made visible by selecting the **View > Main Panels** menu item. The panel can be docked, in which case a tab is provided to re-access it, or it can be set to 'float'. Double-clicking a docked window will isolate it again.

For the remainder of this topic when referencing features, the term objects also applies.

- When a property of the selected feature(s) is the same for all the features then the value of the property is displayed in the properties grid .

- When a property of the selected feature is not the same for all the features then the text 'Different Properties' is displayed as the value of the property. Depending on other parameters the property can be disabled or enabled.

Selection of multiple features/objects

When multiple features/objects are selected the properties grid can only display properties of the same feature type simultaneously. So, for example, if a point and a line are selected, the grid cannot display the properties of both at the same time. Instead the grid shows the properties of the highest order of the selected feature, and populates a combo box at the bottom of the panel with the selected feature types (for example 'Line' and 'Point'). By selecting a feature type in this combo box the properties for each feature type in the selection can be displayed separately.

When objects of different sub-types are selected, only common properties may be viewed and modified. So, for example, if a straight line and an arc line are selected, the first section of the grid that usually displays coordinates is omitted, and only the groups and the assignments sections will be available.

Assignments

The properties grid also displays the attributes which are assigned to the selected features. The assignments are categorised by their type. A type is only present in the grid if at least one attribute of this type exists in the model. That means for example that if no material attributes exist in the model, the material category will not exist in the grid. Depending upon the attribute, the following assignment options may be listed:

❑ **Assigned** If an attribute is assigned to all the selected features then its name is displayed at the left grid column and the word "Assigned" is displayed as its value. If the attribute is assigned to only some of the selected features the text "Different Properties" will be displayed, consistent with other non-attribute properties. If an attribute is not assigned to any of the selected features then it does not appear in the grid.

❑ **Deassign** The attribute will be deassigned from the feature type selected.

❑ **Edit** displays the corresponding attribute definition dialog so the attribute can be edited

❑ **Details** This option is only available when the assignments have additional data and that data is the same for all of them. For example if all the selected features have attribute 'A' not only assigned but also assigned in the same analysis then the 'Details' option will be available, and by selecting this a dialog will display the assignment's data.

❑ **More** This is similar to the Details option; when the assignments have additional data and that data is different, the action 'More' is available. As an example: if attribute 'B' is defined, and there are four lines in a model, and two analyses called 'Analysis 1' and 'Analysis 2'. If attribute 'B' is assigned to one of the lines in 'Analysis 1', and to two of the lines in 'Analysis 2', and for one line it is not assigned at all, the 'More' option will be available. By selecting this option a dialog will display the assignments categorized by their data.

❑ **(New)**This option (at the end of each attribute type's section) allows a new attribute assignment to be made to the selected features.

Groups

Features may or may not belong to any number of groups. 'Included', or 'Not included' will be seen within the grid accordingly.

Showing assignments for the active loadcase only

❑ **Show assignments to active loadcase only** If this check box at the bottom of the Properties panel is checked only those assignments for the current active loadcase will be listed in the grid.

Hierarchy Panel

The Hierarchy panel lists the higher and lower order features/objects of the currently selected <u>single</u> geometry feature or mesh object

The panel is initially hidden until accessed by a selecting a feature/object and choosing the **Hierarchy** context menu item for that feature/object. It can also be made visible by selecting the **View > Main Panels** menu item. The panel can be docked, in which case a tab is provided to re-access it, or it can be set to 'float'.

For the remainder of this topic when referencing features, the term objects also applies.

The type and feature number of the selected feature is shown.

❑ **Is mergable** (only present for Geometry features). Sets the **merge** status.

❑ **Lower order** (not applicable to Points). Lists the lower order features that define the selected feature.

❑ **Higher order** (not applicable to Volumes). Lists the higher order features that the selected feature defines)

Selection of a feature within the hierarchy panel will update the panel to show the hierarchy for that feature.

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

Groups can be defined by pressing the Group ![icon] button on the main toolbar or by selecting the **Geometry> Group> New Group** menu item. Groups can be defined for selected items, visible items, or an empty group can be created ready for adding items to it at a later stage

All current group definitions are listed in the Group panel of the ![icon] 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 ![icon] 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.
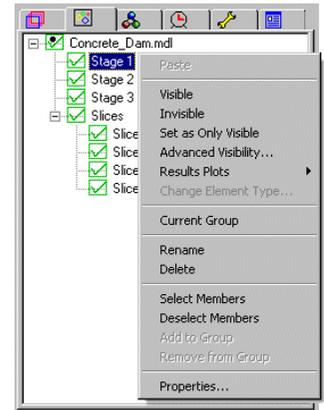
When the group defines a slice section the otherwise generally available Rename, Delete, Add to group, and Remove from group options are unavailable but additional slice-related options are include on the context menu:

- ❑ **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 names are added to the Groups treeview in creation order, but can be re-ordered by using the **Sort by name** context menu item accessed from the Groups tab.

Group symbols explained

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

 Default group icon.

 The addition of a black dot denotes the current group into which all new geometry will be added when created.

When modelling:

When 'Show visibility feedback in treeview' is selected the following group icons are provided:

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

When 'Show visibility feedback in treeview' is selected the following group icons are provided:

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

 (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:

When 'Show visibility feedback in treeview' is selected and a results file is loaded the following icons are provided:
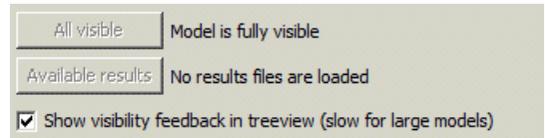
 (green tick, green border) All of the objects in this group are showing results.

 (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 **'Show visibility feedback in treeview**' option 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.

All visible	Model is fully visible
Available results	No results files are loaded

☑ Show visibility feedback in treeview (slow for large models)

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.

Manipulating the View of a Model

A number of rotational, pan and zoom tools, and pre-defined projections are available to manipulate the view of the model.

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 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 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 screen Y axis.
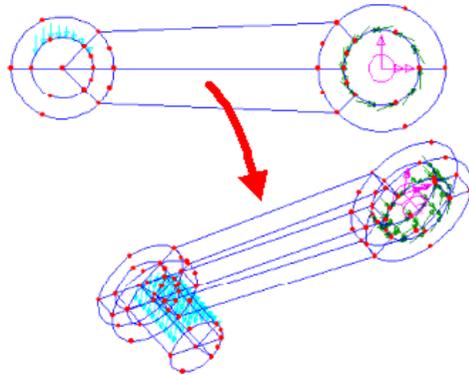
Note. Models are dynamically rotated using the 'model ball' method. With this method, a model can be imagined to be surrounded by a sphere such that a mouse-click and a drag of the cursor on the screen represents clicking on the surface of the sphere and dragging to rotate it to a new position. In doing 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

Vertical axis

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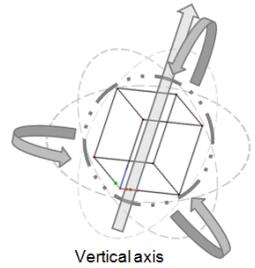
Incremental rotate Allows the model to be rotated by an specified rotation about a chosen global axis. The specified rotation may be modified by adjusting the rotation increment on the window's properties.

- Anti-clockwise Y
- Clockwise Y
- Anti-clockwise X
- Clockwise X
- Anti-clockwise Z
- Clockwise Z
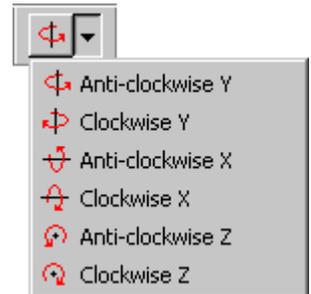
View along axis Views the model along a chosen global axis. Note that this toolbar button is not provided as part of the standard user interface but it can be added by selecting the View > Toolbars > Customise > menu item. This facility is also available by clicking in the X, Y or Z boxes in the Status Bar (holding-down the Shift key, if needed, to obtain the views along the negative screen axes.)

- Show XY (+Z)
- Show XY (-Z)
- Show ZY (+X)
- Show ZY (-X)
- Show XZ (+Y)
- Show XZ (-Y)
- Show Isometric
- Show Dimetric
- Show Trimetric
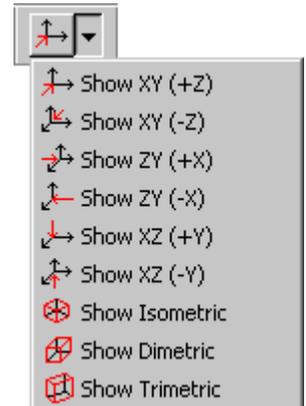
Zooming in and out

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.

Note that this facility is deactivated when a perspective projection is in use.

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, similar to dynamic rotate. Otherwise the zoom takes place with respect to the centre of the view window. Using the cursor the model can be visually enlarged or reduced in size.

Note that when a perspective projection is in use and it is required to zoom in to a particular part of the model, select a feature(s) of interest first prior to selecting the dynamic zoom button.

Panning (Dragging) the Model

Dynamic Pan is carried out by selecting the Dyanimic 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.

View projections

Isometric, Dimetric and Trimetric standard views of the model can be selected, as well as a perspective projection.

Isometric Sets an isometric projection for the view

Dimetric Sets an dimetric projection for the view

Trimetric Sets a trimetric projection for the view

Perspective Sets a perspective projection according to properties specified in the **View properties** dialog.

Orthogonal Model Views
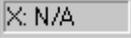
These buttons are located in the status bar.

X: N/A   = View along the +X axis towards the origin

Y: N/A   = View along the +Y axis towards the origin

Z: N/A   = 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 + X: N/A   = View along the -X axis towards the origin
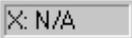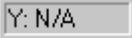
Ctrl key + Y: N/A   = View along the -Y axis towards the origin

Ctrl + Shift key + X: N/A   = 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.

Resetting 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 model save, that have resulted in editing of the model data, 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. Note that the undo button does not undo any model rotation or zoom operations. Instead specific toolbar buttons can be used to undo these operations, if required. See the notes below.

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

Notes:

- Accidentally deleting an attribute (such a material attribute) is treated as a 'simple' undo resulting in that attribute simply being reinstated if the undo button is pressed. For a more complex operation, 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.

- **Undo rotate** and **Undo zoom** buttons can be added to an appropriate toolbar as follows:

    1. Select the **View > Toolbars** menu item

    2. Select the **Customize** button

    3. Select the **Customize** tab and choose to view the complete set of Rotate and Zoom toolbar buttons.

    4. Drag and drop the buttons onto an appropriate toolbar.

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 **Tools> 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.

```
SCALE 1/    10.00
EYE X-COORD =    .0000
EYE Y-COORD =    .0000
EYE Z-COORD =   1000.
```

❑ **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 customized.

❑ **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.

❑ **Startup scripts** automatically run after a new model is created, an existing model is opened, or a model is solved. These files allow user-defined combinations of commands to be inserted to set preferred defaults, colour schemes, or results contour preferences.

❑ **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.

❑ **Screen** When connecting a laptop or PC to an external screen or projector it is possible to specify which screen should be used to ensure that fonts and other rendered information can be seen at an appropriate resolution.

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.

- ❑ **Perspective projection** Switches from isometric to perspective viewing of the model. When set, an option to set a **Fixed camera position** is also available. This controls whether there is relative motion between the camera and the model when pan and zoom are used.

    - **View angle** determines the rate at which the perspective distortion occurs. A suggested initial setting is 45.

    - **Close depth** This parameter determines how far in front of the view position objects are removed from the visualisation. If it is set to zero, an object will grow to infinite size on the screen before it disappears from view. If it is greater than zero, then objects will disappear from view as the depth is reduced. This is input as a percentage of the view depth at 100% zoom, and is defaulted to 1%.

- ❑ **Show only activated elements** Controls whether only the elements activated in the active loadcase are displayed or not.

- ❑ **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.

- ❑ **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.

- ❑ **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.

- ❑ **Save View** Saves the current view settings in a named entry in the Utilities 🔧 treeview. See **Saved views** for more information.

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.

Initial deformations

❑ **Exaggerate initial deformations** (Off by default) Initial deformations may be inherited from a previous analysis, as for example with an initial imperfection in a girder system generated from an eigenvalue buckling mode, used for a full nonlinear buckling analysis. See Specifying an Initial State for an Analysis for details. In such a case, usually the initial deformation is not exaggerated (displayed at a scale of 1:1), so that it is not distracting when examining deformations in the current (e.g. nonlinear buckling) analysis. However the initial deformation can be exaggerated by turning on this check-box.

Screen

❑ **Automatic** Contains a dropdown containing an entry for each connected screen for which physical size information can be determined. Selecting an entry and pressing apply will rescale all the objects within LUSAS Modeller to a size appropriate for the selected screen. If no screen sizes can be detected, then the manual field will be selected by default and populated using a screen pixel-based calculation.

❑ **Manual size** Allows a screen size to be specified (measured in inches between diagonal corners of the screen).

Customize 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 Customize 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. The Customise dialog presented shows available toolbar groups, allows toolbar groups to be turned on or off, allows new toolbar groups to be defined, customised toolbar groups to be created, or user-defined toolbar buttons to be 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 Microsoft and is incorporated into LUSAS Modeller for general use.
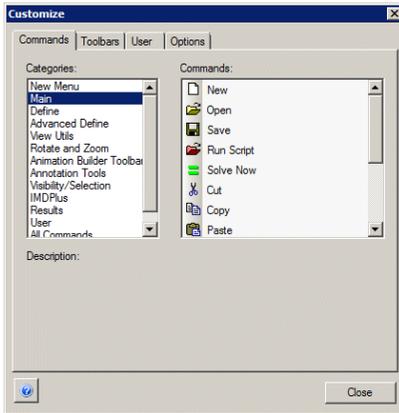
Main category toolbar group buttons available, showing text labels	The toolbars tab shows the status of each toolbar group
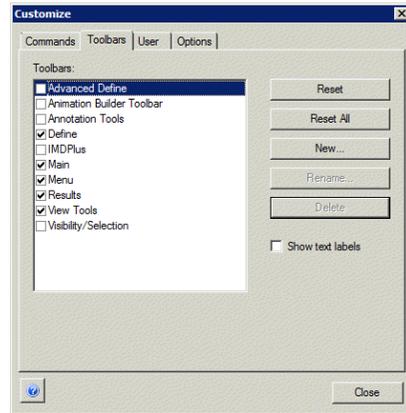
Turning toolbar groups on and off

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 text labels** displays the text label beneath each button in the toolbar group.

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.

Creating customized toolbar groups

Toolbar buttons can be added to existing toolbar groups or placed on a new toolbar button group.

• Use the **Commands** tab to select a toolbar category and then, from the arrangement of buttons shown for that category, drag and drop a toolbar button into the existing or newly named toolbar group on the user interface. Note that toolbar 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 **Commands** 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:\Scripts\Bending_Mz.vbs"`
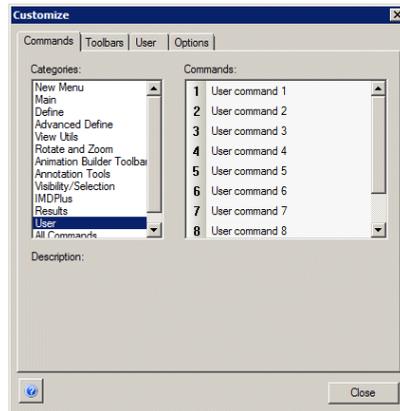
User-definable buttons available
showing text labels
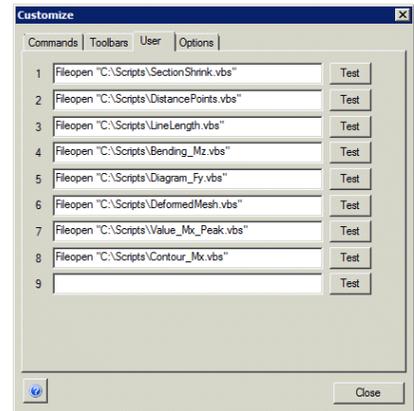
A script can be assigned to each
numbered user button

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.

Options

Use the Options tab to show:

❑ **Show Screen Tips on toolbars** shows a temporary description of the toolbar button when the cursor is moved over the button.

❑ **Large Buttons** shows an enlarged view of the original size toolbar button.

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 failed, or if it has been chosen to stop an analysis during a solve, as well as re-start an analysis from a stated increment or loadcase
- ❑ **HistoryFiles** (.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 and others) allow graphical and related 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 for use with third-party applications.

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-LUSAS version 15.0 model files

Opening models created prior to LUSAS 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 the following for more details:

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

- **Opening pre-LUSAS Version 15.0 models that reference multiple results files**

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-LUSAS Version 15.0 models that do not reference multiple results files

From LUSAS Version 15.0 onwards 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:
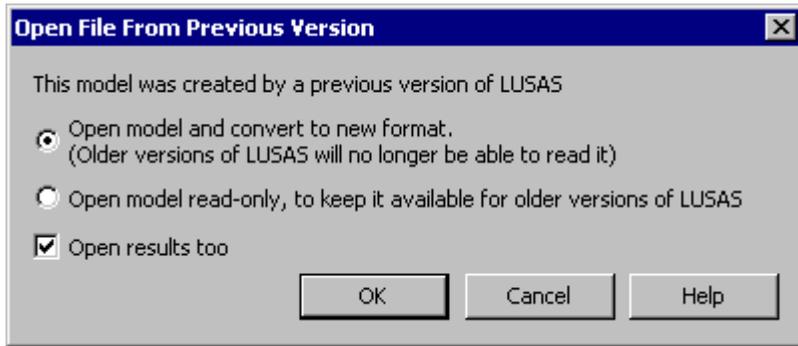
On the File open dialog displayed:

❑ **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-LUSAS Version 15.0 models that reference multiple results files

From LUSAS Version 15.0 onwards 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:
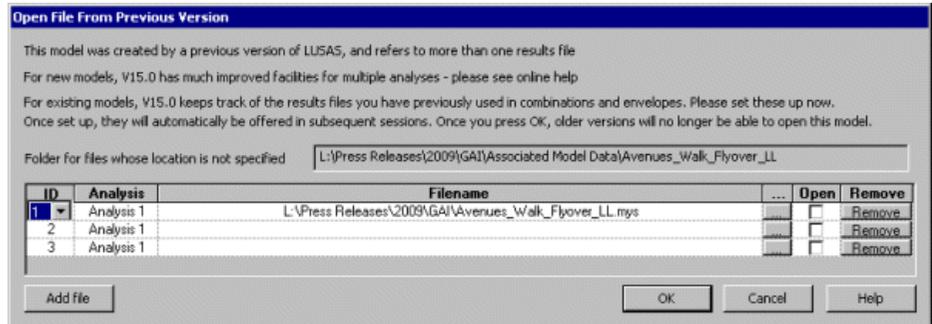
On this example dialog the model opened has three results files associated with it: results file **ID** 1, 2 and 3. The location of results file 1 is assumed to be the same folder as the model file location because prior to Version 15.0 a results file was (by default) written to the same folder and with 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 LUSAS, 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-LUSAS 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**.

- Files written to and present in Associated Model Data folders can be re-created from model files. As a result, whilst entire folder structures of folders containing LUSAS model data can be chosen to be backed up by system administrators for record and security purposes, there is no need to back-up Associated Model Data folders if it is chosen to not do so for diskspace, storage capacity or other reasons.

Model Merge

Model merge allows two or more people to create separate models of specific parts of a structure (as for example for different storeys of a building, or different elements of a stadium) and then combine them at a later date into one master model. To do this a previously saved model can be merged with a currently-loaded model using the **File> Model merge...** menu item. The resulting dialog presented provides options to specify how the geometry, attributes, loadcases and groups from the imported model are to be merged with equivalent objects in the currently open model.

Merge attributes

The following choices are available for merging model data

- ❑ **Off** The attributes of imported model and currently open model will not be merged.
- ❑ **By name** The imported model attributes of the same type will be merged with those of the currently open model if they have the same name.
- ❑ **By name and data** The imported model attributes of the same type will be merged with those of the currently open model if they have the same name and the same definition data.

Merge geometry

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 an assignment of that type.

Notes

- 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.

Other options

❑ **Merge groups by name** The groups in the imported model will be merged with those in the current model if the groups have the same name. The groups are merged by adding the members of the imported groups to those groups in the currently open model. This option is on by default.

❑ **Merge loadcases** The source and destination analyses will be merged if the source analysis has the same name as an existing destination analysis and the two analyses have the same number of loadcases with the same names in the same order. This option is off by default.

Naming convention for merged attributes and analysis names

When the names of imported objects have the same name as objects in the currently open model, and are <u>not</u> to be merged, the following takes place:

- **For attributes, loadcases and groups** a number in parentheses is appended to the original name, so an imported model attribute name of "Lmsh1" would become "Lmsh1 (2)" in the currently open model.

- **For analyses** the word 'Imported' in parentheses is appended to the imported analysis name, so an imported model analysis name of "Wind loading" would become the destination analysis "Wind loading (Imported)". Note all inheritance is removed from the imported analyses as they are copied into the current model because the base analysis in the current model will most likely have different assignments to that in the imported model, and there can be only one base analysis in the final model.

- **For geometry objects** the number and name of geometry objects will be retained, if possible, in the current model. Where this is not possible a number in parentheses will be appended to the object name if it has one.

- The object number used to identify each item in the Attributes 🔗 Treeview will be changed to give some indication of its origins. For example, if Point 23 in the source database clashes with point 23 in the destination database, then rather than just using the next available number say point will use the number 123 (23 + 100) in the current model, rather than just using the next available.

Notes

- Mesh-only models cannot be imported / merged.

- The two models must have the same units because geometry or attributes are not scaled during the import/merge process.

- When a model has been imported and merged with the current model the File > Save menu item acts as if it were the File Save As option, requiring a filename to be specified, protecting the current model from being accidentally overwritten..

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 contains a record 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.

Solver restart files are not created by default. Their creation must be specified in LUSAS Modeller prior to running an analysis by choosing the **File> Export Solver datafile...** menu item, accessing the **Advanced Solution Options** dialog and selecting **Generate** option.

The restart facility enables failed or terminated analyses to be restarted from the last saved or chosen increment, or loadcase. 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 solving that may otherwise mean starting the analysis again 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.

Notes

- Restarts from a Solver restart file are not directly supported by LUSAS Modeller, so must be run using LUSAS Solver. See the Solver Manual for more details.

- Restarts from a Solver restart file are indirectly supported by LUSAS Modeller. For more information see "Specifying an initial state for an analysis" on page 536

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

From Version 15.0 of LUSAS onwards 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.
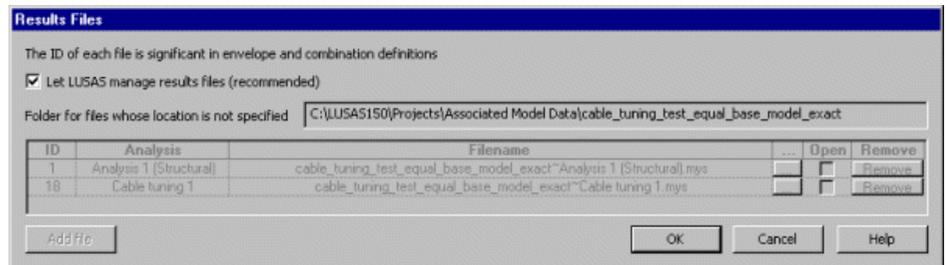
It is recommended to always let LUSAS manage results files.

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 was introduced in 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).

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, or by setting Advanced Output Options that are set as part of defining a Structural Analysis (or Thermal Analysis). 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.

Startup script files are provided which automatically run after a new model is created, an existing model is opened, or a model is solved. These files are created in Visual Basic Script (.VBS) format and are supplied blank for user-defined combinations of commands to be inserted to set preferred defaults, colour schemes, or results contour preferences.

Startup Script Files

Startup script files are provided which automatically run after a new model is created, an existing model is opened, or a model is solved. These files are created in Visual Basic Script (.VBS) format and are supplied blank for user-defined combinations of commands to be inserted to set preferred defaults, colour schemes, or results contour preferences.

The files provided are:

- **AfterSolve.vbs**

- **AfterNewModel.vbs**

- **AfterOpenModel.vbs**

For use by the current user

To carry out actions in one of these files that will affect the current user only, create the file you want to alter here:

%LOCALAPPDATA%\LUSAS 16.0\x86

For example on window 7 for AfterSolve.vbs you would create:

C:\Users\jamesh\AppData\Local\LUSAS 16.0\x86\AfterSolve.vbs

For use by all users

To carry out the actions in one of these files that will affect all users you would create your file here:

%PROGRAMDATA%\LUSAS 16.0\x86\config\AfterSolve.vbs

For example on window 7 for AfterSolve you would create:

C:\ProgramData\LUSAS 16.0\x86\config\AfterSolve.vbs

 Note: The easiest way to add vbs commands to a startup script file is to record a series of operations in LUSAS and save the resulting file. See **Script Files** for details.

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 are a historical proprietary image format that 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.

Notes

- 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**

- Picture files do not support the perspective viewing facility.

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.

Page Setup

Accessed via the **File > Page Setup** menu item, it allows the paper size and orientation to be specified when printing the contents of the Modeller window to a printer or a file. Print margins may also be set to control the position of the model data on the page to allow for binding edges or for the inclusion of contour key information so that it does not lie on top of a model.

Notes

- Margin values specified control the position of the model data on screen for both Page Layout Mode and also for Working Mode.

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.

Currently supported interface file formats:

❑ **ABAQUS input files (.inp)** (used to import node and element data)

❑ **ANSYS cdb files (.cdb)** (used to import node and element data)

❑ **BIM/BrIM** (*.ifc) (used to import and export BIM/BrIM file data)

❑ **CMD** (**.cmd**) Format for import of LUSAS Modeller model files saved as command (CMD) files in previous versions of LUSAS.

❑ **DXF** (**.dxf**) AutoCAD Drawing eXchange Format.

❑ **ETABS input files** (**.e2k**) (used to import geometry and property/attribute data)

❑ **IGES** (**.igs**) Initial Graphics Exchange Specification. Format for import and export of geometry data.

❑ **MIDAS input files** (**.mct, .mgt**) (used to import geometry and property/attribute data)

❑ **NASTRAN Bulk Data files** (**.bdf,.dat**) (used to import node and element data)

❑ **PATRAN** (**.def**) Neutral file format for inputting phase I geometry information and outputting phase II mesh information

❑ **SAP input files** (**.s2k**) (used to import geometry and property/attribute data)

❑ **Search area topology** (**.inf**) contain details of search areas used by Autoloader vehicle load optimisation software. These when imported into LUSAS are converted into geometric line and surface data for graphical cross-checking.

❑ **Solver Data Files** (**.dat**) LUSAS Solver data files (used to import or export node and element data)

❑ **STEP** (**.stp**) STandard for the Exchange of Product data.

❑ **STL** (**.stl**) Stereolithography data files.

Summary of Interface File Import / Export Capability

Interface file name and extension	Import file into LUSAS Modeller?	Export file from LUSAS Modeller?
ABAQUS (.input)	YES	NO
ANSYS (.cdb)	YES	NO
BIM/BrIM (.ifc)	YES	YES
CMD (.cmd)	YES	YES
DXF (.dxf)	YES	YES
ETABS (.e2k)	YES	NO
IGES (.igs)	YES	YES
MIDAS Input File (.mgt)	YES	NO
NASTRAN Bulk Data Files (.bdf, .dat)	YES	NO
PATRAN (.def)	YES	NO
SAP (.s2k)	YES	NO

Interface file name and extension	Import file into LUSAS Modeller?	Export file from LUSAS Modeller?
LUSAS Solver data Files	YES	YES
STEP (.step, .stp)	YES	YES
STL (.stl)	YES	YES

Notes

- See **Importing Interface 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.

- In addition to interface files that can be imported or exported from within LUSAS Modeller, other standalone programs are available for particular applications and are documented in the CAD Toolkit User manual.

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.

Checkboxes are disabled (as shown by ▣) according to whether import or export of data has been chosen.

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.

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.

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 a Solver data file

The **File> Export Solver datafile...** menu item can be used 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, <u>without any solving taking place</u>.

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

When a solution takes place, an output file is generated. Advanced (solution) output options can be defined for each solution. These options can be accessed from the **Advanced** button on the Export Solver datafile dialog, or by setting Advanced Output Options that are set as part of defining a **Structural Analysis** (or **Thermal Analysis**).

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 also be written to a history file if required.
- ❑ **History** when checked outputs the named variables, and selected node and element results from LUSAS Solver to a **History file**
- ❑ **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

- • 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.

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
Coalesce 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
Use dimensions from file geometry post import	If selected uses the dimensions and units as specified in the imported file	True
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%

Option	Description	Default
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

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.

BIM / BrIM File Import/Export

Import

Basic geometry data from third-party BIM/BrIM files can be imported to create a feature-based geometry model in LUSAS using the **File> Import...** menu item, or a mesh-only model by using the **File > Import Mesh** menu item. The type of feature that is created within LUSAS Modeller will depend upon which entities have been created in the CAD system from which the BIM/BrIM file was created. For example:

- An extruded area solid (*IfcExtrudedAreaSolid*) entity will create line features with assigned sections within LUSAS Modeller.

- A closed shell (*IfcClosedShell*) will create a volume (comprised of surface and line features) within LUSAS Modeller.

Prior to carrying out a BIM/BrIM file import, the use of extruded area solid entity in the CAD system model should be used in preference to a closed shell entity since this will produce a more efficient LUSAS beam element model that will solve faster than a volume model, which requires a greater number of elements to be assigned to achieve a solution.

When a file is selected the **Advanced** button can be used to specify import parameters. See **Model Types** for more information.

Notes

- Any attributes and assignments that were previously exported from LUSAS to a BIM/BrIM file will also be imported if the BIM/BrIM file containing those items is imported.

- Currently all BIM/BrIM file data that cannot be translated into data that can be used by LUSAS will be disregarded for the purposes of the import.

Export

Both BIM/BrIM Structural domain files (*.ifc) and BIM/BrIM Architectural domain files (*.ifc) are supported for export. Either LUSAS model geometry only, or model geometry and attributes can be exported.

Notes

- When **Geometry only** is selected for export standard BIM/BrIM file entities are written to represent that data. LUSAS feature information is also written to the export file using a private format to enable LUSAS Modeller to import a BIM/BrIM file and recreate correctly named features of a model from a previous export. All details of loadcases, attributes, assignments, and similar are missing in this type of file.

- When **Geometry and attributes** are selected for export, LUSAS analyses, loadcases, attributes and assignments data are also written to the export file using a private LUSAS format to enable LUSAS Modeller to import a BIM/BrIM file and re-create a model from a previous export.

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 following table:

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)	POLYFACE 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.

ETABS Input File Import

Basic data from ETABS Input Files (.e2k) 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.

Notes

- **Meshing**: ETABS does not use the same default beam orientation algorithm as LUSAS Modeller. As a result, when an ETABS input file is imported the Model Properties meshing option 'Orientate as columns and beams' is automatically switched on to identify the alignment of the beams present, and the beam element axes for each beam are set to a suitable orientation to represent either a vertical column or a horizontal beam, as typically used in modelling building frames.

- **Materials**: These are directly available from the ETABS input file. A material attribute is created in LUSAS Modeller for each material present.

- **Geometric properties**: These are directly available from the ETABS input file. Geometric properties may be stated in the input file either by reference to the ETABS section library, for a section description with section dimensions, or for a simple rectangular section dimensions with properties. For all these methods a geometric section property is created in the Attributes treeview within LUSAS Modeller for each beam present. A section library containing sections listed in the ETABS section library is provided on a LUSAS installation kit.

- **Supports**: Fixed and spring supports are imported.

- **Loading**: Concentrated, face, beam, temperature and self weight load types are imported. Wind and seismic loading cannot be imported.

- ETABS storey / grid line data is not currently imported.

- Importing an ETABS input file sets the **vertical axis** within LUSAS Modeller to be the Z axis. Imported beam sections are orientated accordingly.

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

Entity	Description
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.

MIDAS Input File Import

Basic data from MIDAS Input Files (.mct, .mgt) 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.

Notes

- **Mesh properties**: MIDAS does not use the same default beam orientation algorithm as LUSAS Modeller. As a result, when a MIDAS input file is imported the Model Properties option 'Orientate as columns and beams' is automatically switched on to identify the alignment of the beams present, and the beam element axes for each beam are set to a suitable orientation to represent either a vertical column or a horizontal beam, as typically used in modelling building frames.

- **Material properties**: These are not directly available to LUSAS from the MIDAS input file. Often, materials are listed in the input file with reference to an internal material within MIDAS; sometimes some material values are present in the input file, but not all values. As a result, a material attribute is created within LUSAS Modeller for each material name that is present in the input file, and values of zero are present when no properties for a parameter can be derived. The Text Output window within Modeller will list all issues with an import. To update the properties it will be necessary for users to locate the material library data in the MIDAS installation directory and enter or update corresponding values for each material name within LUSAS Modeller.

- **Geometric properties**: These are not directly available to LUSAS from the MIDAS Input File. The MIDAS file defines beam section properties either by a name that references an internal library, or by section dimensions, where it is not clear what part of the section the dimensions relate to. As a result a dummy

attribute with zero values is created. To update the properties it will be necessary for users to locate the relevant geometric property data in the MIDAS installation directory and enter corresponding values for each geometric property name within LUSAS Modeller.

- **Supports**: Fixed and spring supports are imported.

- **Loading**: Concentrated, face, beam, temperature and self weight load types are imported. Wind and seismic loading cannot be imported.

- MIDAS grid line definitions are not currently imported .

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.

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

SAP Input File Import

Basic data from SAP Input Files (.s2k) 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.

Notes

- **Meshing**: SAP does not use the same default beam orientation algorithm as LUSAS Modeller. As a result, when a SAP input file is imported the Model Properties option 'Orientate as columns and beams' is automatically switched on to identify the alignment of the beams present, and the beam element axes for each beam are set to a suitable orientation to represent either a vertical column or a horizontal beam, as typically used in modelling building frames.

- **Material properties**: These are directly available from the SAP Input File. A material attribute is created in the Attributes treeview within LUSAS Modeller for each material present.

- **Geometric properties**: The SAP input file defines both beam section dimensions and the section properties. A geometric section property is created in the Attributes treeview within LUSAS Modeller for each beam present.

- **Supports**: Fixed and spring supports are imported

- **Loading**: Concentrated, face, beam, temperature and self weight load types are imported. Wind and seismic loading cannot be imported.

- SAP grid line definitions are not currently imported .

- Importing a SAP input file sets the **vertical axis** within LUSAS Modeller to be the Z axis. Imported beam sections are orientated accordingly.

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, mesh or node information may be exported to STL format from the **File> Export...** menu item.

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.

Command Files

V13 Command files are used to import and export models from and to version 13 of LUSAS Modeller.

Models are imported using the **File> Import...** menu by selecting the **V13 CMD Files (*.cmd)** entry in the **File of type** combo.

Models may also be exported in cmd format using the **File> Export...** menu and selecting the **V13 CMD Files (*.cmd)** entry from the **Save as** type combo.

Notes

- When exporting to a cmd file an analysis type must be chosen. All of a base analysis is written to the cmd file, but if another analysis references a base analysis that data will not be written to the cmd file. To overcome this, the attributes referenced in the base analysis should be assigned directly to the other analysis prior to the File> Export menu item being chosen.
- Only geometry features supported by version 13 Modeller will be exported.

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.

Associative modelling

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 selecting a relevant Geometry menu item for each geometric feature type, for example **Geometry> Line**, or be imported from other systems. **See Importing Geometry Data**.

Notes

- 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.

- 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.

- 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 cannot 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:
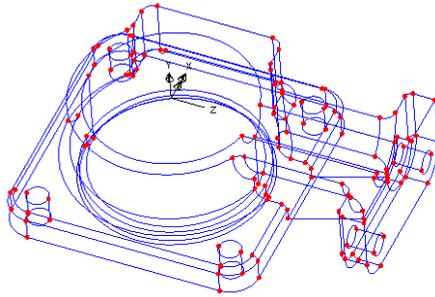
Default wireframe geometry visualisation
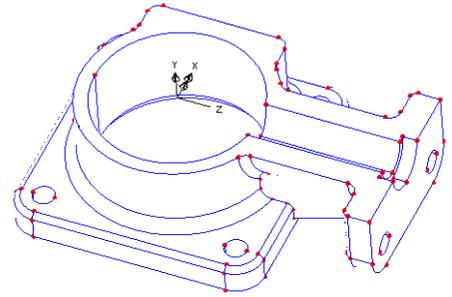
Wireframe with hidden Parts removed
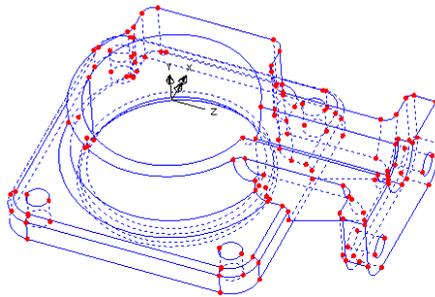
Wireframe with hidden parts draw dashed
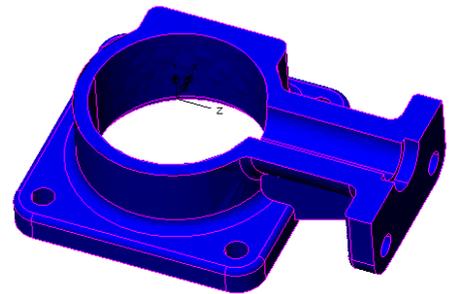
Solid fill with wireframe and hidden parts removed
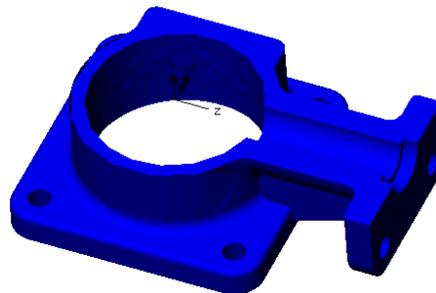
Solid fill with wireframe (hidden parts removed) and Points hidden

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.
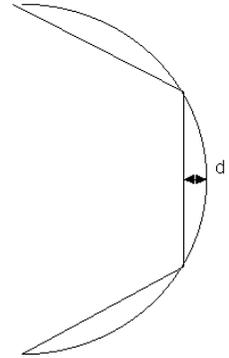
Notes

- 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.

Mesh Key
——— Surface Mesh 1
——— Divisions=1
——— Divisions=3
——— Divisions=4
——— divs12

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** or **Line axes** can be drawn to indicate the local x direction of Line.
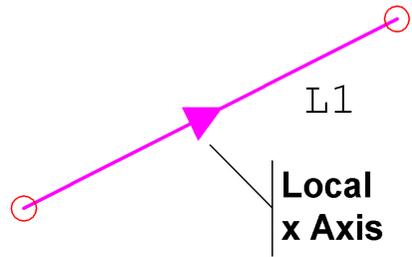
❑ **Surface axes** or **Surface normals** can be displayed. Axes are positioned adjacent to the first Line in the Surface definition. In the example shown the axes are orthogonal but viewed from an angle to show the z axis orientation
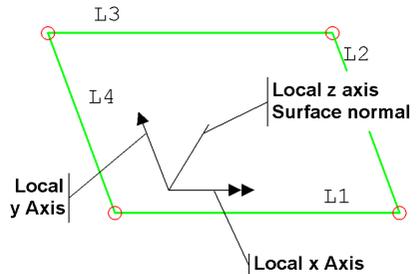
❑ **Volume axes** can be displayed. The origin for the axis is closest to the first point in the first Surface of the Volume definition.
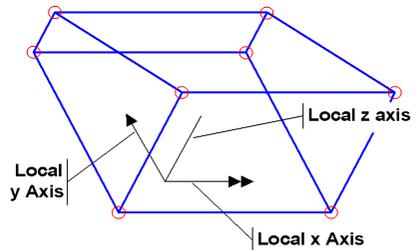
Tip: To aid axes visualisation on larger models, choose the **Orientations only if selected** option. This will display axes only on selected features.

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

❑ Thermal surfaces
❑ Retained freedoms
❑ Damping
❑ Activate
❑ Deactivate

- ❑ Supports
- ❑ Loading
- ❑ Transformed freedoms
- ❑ Composite
- ❑ Slideline
- ❑ Constraints

- ❑ 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.

Defining points

Points can be defined by the following methods:

- ❑ **By 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.

- ❑ **By Grid** Allows points to be defined with respect to a geometry definition grid (which displays a regularly spaced set of points) or a **layout grid**. On each cursor click, a point is created at the nearest grid intersection.

- ❑ **By 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.
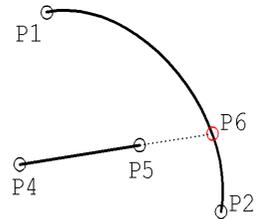
- ▪ **All point pairs** creates Points at all possible intersections.
- ▪ **Nearest point pair only** creates a Point on each line where the projection of the Lines is at its nearest.
- ▪ **Nearest pair to reference position** defines Points on each line at the intersection nearest to the defined reference 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.
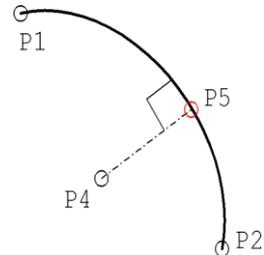
❑ **By Extension** Defines a point at the extension of a selected Line. The extension may be defined as a parametric or actual length.

❑ **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 directions are defined by the order of the points defining the line. 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)

❑ **Elliptical arc / Ellipse** defined by point coordinates and specification of major and minor axes directions, and major and minor radii.

❑ **Spline** defined by two or more Points.

❑ Lines may also be created by **Splitting** lines, and **Intersecting** with other lines

Defining lines

Lines can be defined by the following menthods:

❑ **By 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.

❑ **By Grid** Allows lines to be defined singly with respect to a geometry definition grid (which displays a regularly spaced set of points) or a **layout grid**, or by creating multiple lines in the horizontal or vertical directions, again with respect to a **layout grid**. See **Defining lines and surfaces with respect to a layout grid** for more details.

❑ **By 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.

❑ **By 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.
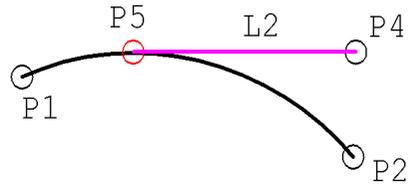
❑ **Tangent > Line between Arcs** Defines a Line which is tangent to two coplanar Arcs. The new Line can be defined as an inside or outside tangent. Options are available to split the Arcs at the new Points, and then delete the original Arcs.
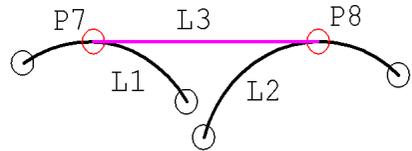
❑ **By Sweeping** Defines a Line by sweeping a Point through a transformation (translation, rotation, mirror or scale). In this example Line 1 is created by sweeping Point 1 through a *translation* in X and Y.
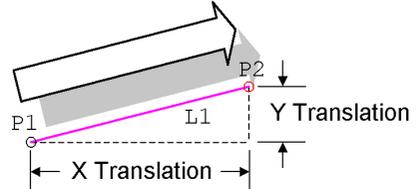
❑ **By Offsetting** Defines a Line or series of Lines offset from a selected Line or series of Lines. If multiple Lines are selected a new set of Lines can be offset in one operation. The direction of the generated lines can be controlled by selecting a point in the direction of the new lines required, 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.
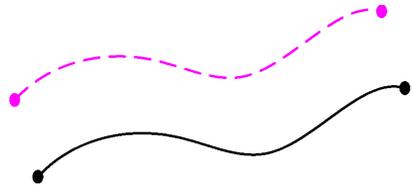
❑ **By Joining** Defines a number of Lines by joining two sets of Points. Points put in selection memory define the start of each Line, and Points selected in a subsequent 'normal' selection define the end of each Line. The Points must pair up equally. Lines are joined according to the order in which 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.
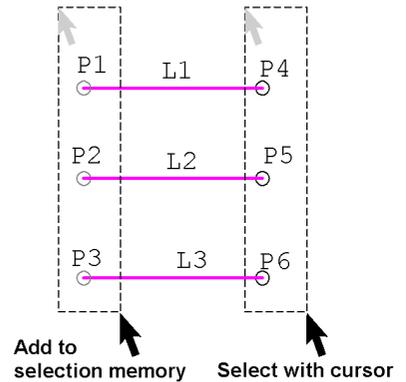
Note also that a fan of lines from one point to many others can be created by setting one point only in selection memory prior to selecting the other set of points in a 'normal' selection. This is useful for creating lines to represent the cables radiating from the top of a tower to the deck on a cable 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.

Line by intersection
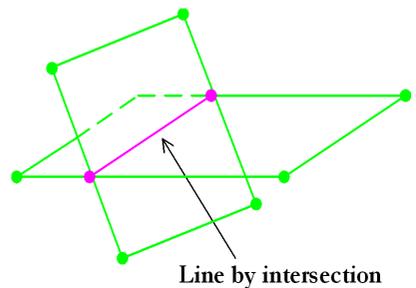
❑ **By Manifolding** (via projection) Existing Lines may be projected or laid onto an existing Surface. A Surface to be projected onto is selected, followed by the Line to be projected and the **Geometry> Line> By Manifolding** menu item is used to

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.
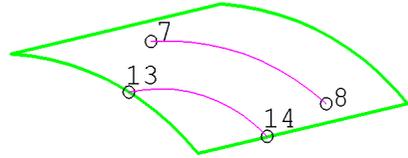
Defining Arcs and Circles

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 a number of 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)
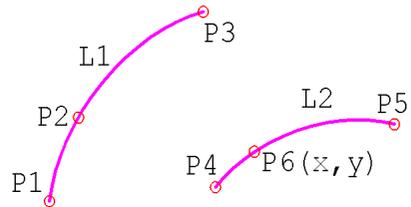
- **Start Point, Directional Point, End Point, and Radius** Defines an Arc or Circle between two coordinate points, to a specified radius. A third coordinate point is required to indicate the direction in which the arc or circle bulges. In this example arcs are created from Point 1 to Point 2 with a 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.
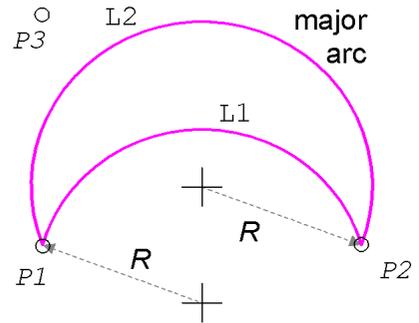
- **Start Point, Centre Point and End Point** Defines an Arc or Circle between two coordinates, with a third coordinate defining the centre of the arc or circle. The centre must be equidistant from the start and end points. In this example arcs are created by selecting Points 2, 3, 4 with Point 3 as 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.

- **In Plane** (seen when defining a circle) is a point (which does not have to be on the circumference of the circle) that will define the plane in
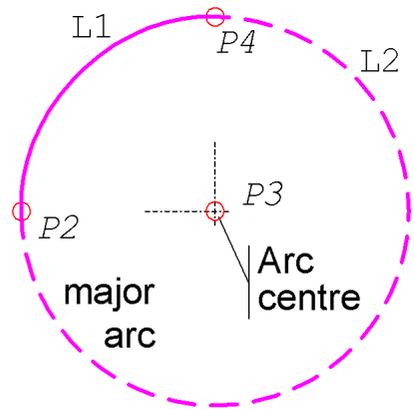
which the circle is drawn from the defined centre, and start and end point of the arc defining the circle.

- **Radius** If a radius is specified, the start and end point locations of the drawn arc are adjusted to match the radius. If the radius is not specified. then it is defined by the distance between the centre and start point.

❑ **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.
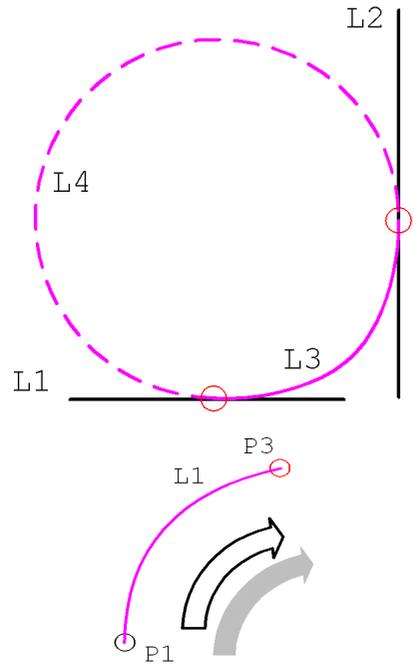
Defining a circle

The following values and settings would define a circle of radius 3, about an origin of 0,0: Centre 0,0 / Radius 3 / Start/End -3,0 / In Plane 1,1

Note that when a radius is specified the Start / End coordinate does not have to lie at the actual start or end of the arc because it defines the direction in which the point defining the arc lies. When no radius is specified the Start / End coordinate is used to defined the radius of the circle about the defined centre. The In Plane value of 1,1 simply defines the plane in which the circle lies - and because no z value is defined the circle will lie in the x-y plane.
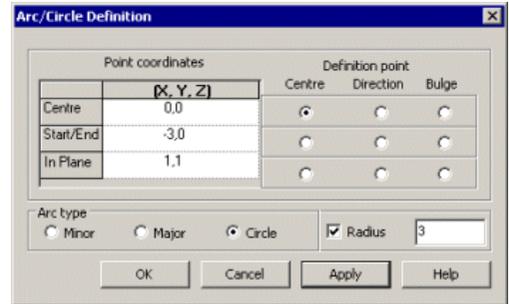
Elliptical Arc and Ellipse Definition

Elliptical arcs and ellipses are defined from the **Geometry> Line> Ellipse** menu item. Coordinates can be entered manually or taken from selected points.

Ellipse point coordinates and settings are used to define the arc/ellipse in the following ways:

❑ **Start Point, Centre Point and End Point** Defines an arc between the start and end coordinates, with the other specified coordinate defining the centre of the ellipse about which the arc is generated..

❑ **In Plane** (only seen when defining an ellipse) is a point (which does not have to be on the circumference of the ellipse) that will define the plane in which the ellipse is drawn from the defined centre, and start and end point of the arc defining the ellipse.

❑ The Axis direction is specified by selecting either a **Major axis** or **Minor axis** along with stating an **x, y, z** direction vector to define the direction in which the major or minor axis resides. For example entering values of '0 1 0' would define an axis in the Y direction.

❑ **Minor part**, **Major part** or **Full ellipse** options draw either an elliptical arc of the shortest length between the start and end points, an arc of the longest length between two points, or a full ellipse based upon all the associated data.

Defining an ellipse

The following values and settings would define an ellipse of major diameter 20, minor diameter 10, about an origin of 0,0,0: Centre 0,0,0 / Start/End 10,0,0 / In Plane 0,5,0 / Major axis with a direction vector of '1 0 0' / Full ellipse / Major radius 10 / Minor radius 5.

Alternative method

As an alternative to the above stated method, ellipses can also be created by drawing a circle, dividing it into (say) 4 arc lines, selecting these lines, and then by selecting the context menu option Move and choosing the Scale option, scale factors can be specified for each direction relative to the centre of the circle. Note that if the circle is defined by a single line, it will not be possible to scale it in this way; the circle must be divided up such that each arc line can become a spline line that takes the correct curvature.

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)
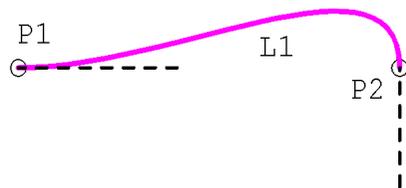
respectively.

❑ **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.
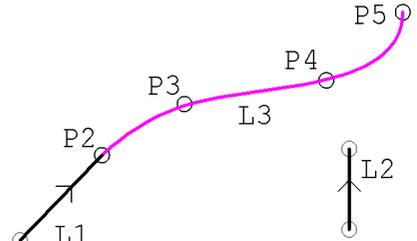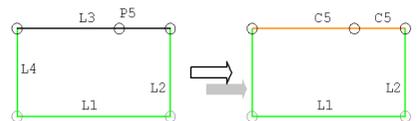
Splitting Lines

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.
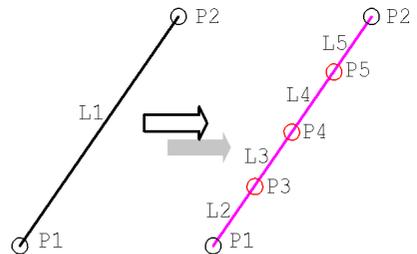
❑ **Parametric or Actual Distance**
Splits an existing Line into a number
of divisions based on specification of
parametric distance values along the
Line. The direction of the existing line
is used to calculate the splitting
positions. A new line is created at
each division. In this example Line 1
is 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.
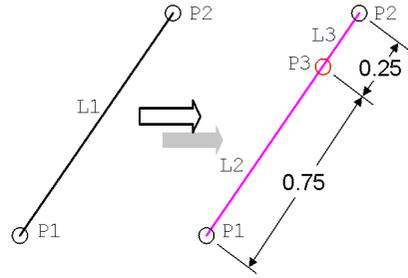
Combined Lines

A Combined Line is a Line which is composed of several continuous individual Lines.
Combined Lines may be used in exactly the same ways 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 must be used when carrying out a steel design code check whenever a
series of lines are used to represent a single steel member, such as when one beam is
supporting a number of cross-beams along its length. The series of lines representing
the supporting beam will need to be specified as a combined line.

Combined Lines are defined from the **Geometry> Line> Combined Line** menu item
after pre-selecting the lines required.

❑ **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.
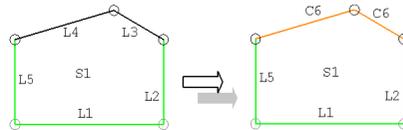
Notes

- Automatic numbering uses the next highest available number for Lines or Combined Lines.

- Mesh attributes cannot be assigned directly 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.
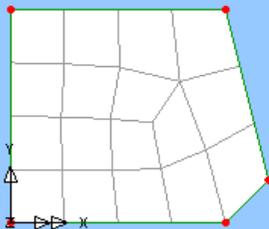
Case Study. Using Combined Lines

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.
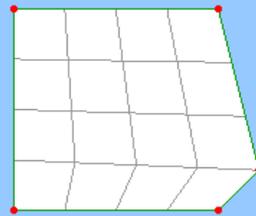
Irregular Mesh

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.

Defining Surfaces

Surfaces can be defined by the following methods:

❑ **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 Grid** Allows surfaces to be defined singly with respect to a geometry definition grid (which displays a regularly spaced set of points) or a **layout grid**, or by creating multiple surfaces in the horizontal or vertical directions, again with respect to a **layout grid**. See **Defining lines and surfaces with respect to a layout grid** for more details.

❑ **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**.
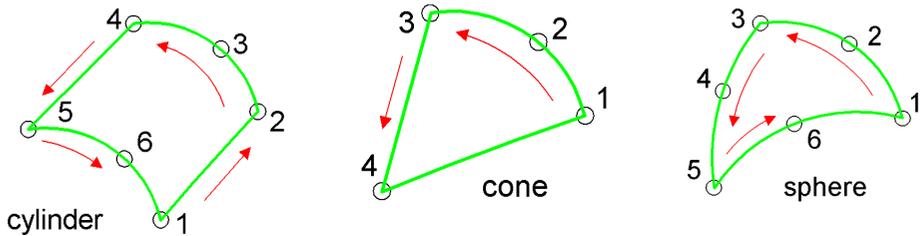
Surfaces generated from selected Lines
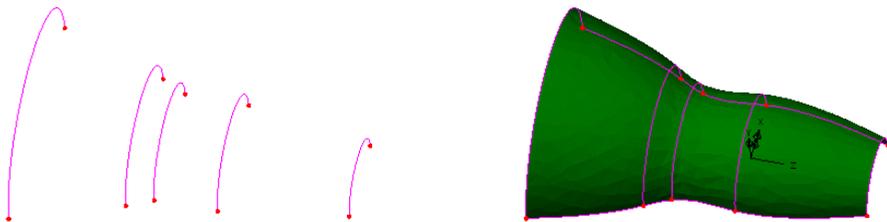
Lofted Surface generated from selected Lines

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

These are defined by using the **Geometry> Surface> Create Holes** menu item.

- ❑ **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 that 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** Deletes all selected 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.
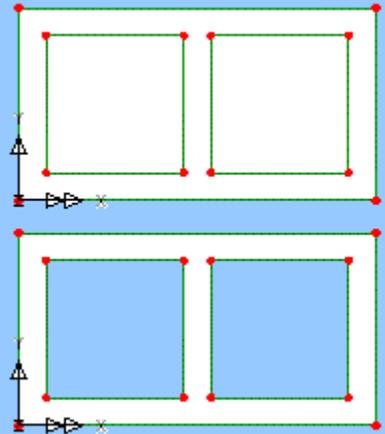
Redefine Perimeter

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.
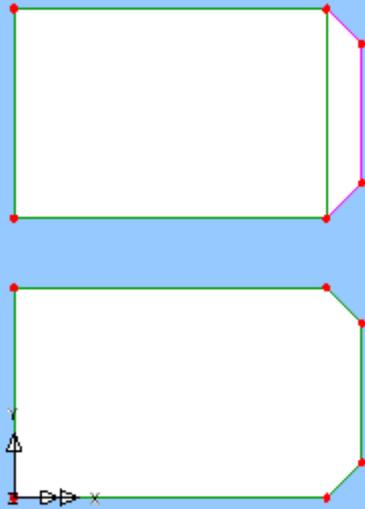
Sweeping Surfaces

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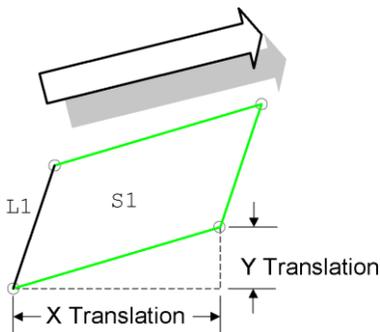
A cylindrical Surface is defined by sweeping Line 1 through a rotation:
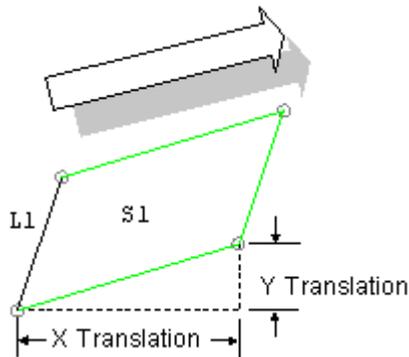
Offsetting

❑ **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.
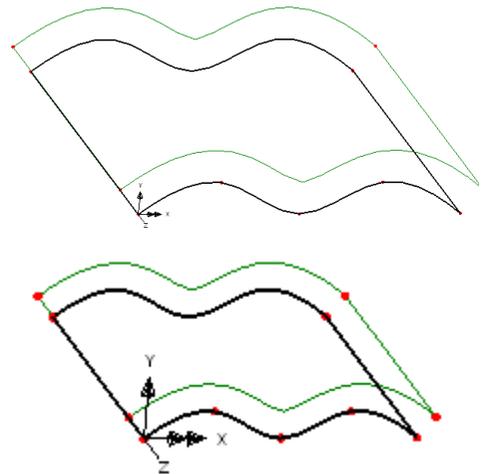
Splitting Surfaces

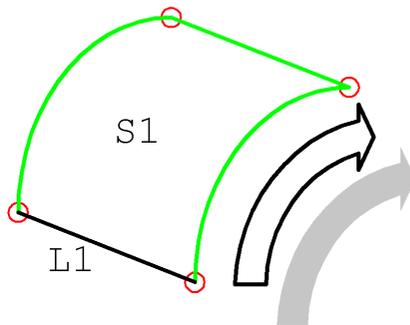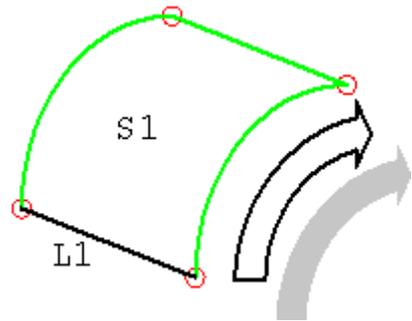Surfaces may be defined by splitting an existing surface in a number of ways using the menu items under **Geometry> Surface> Splitting**. Surface splitting commands contain an option to automatically delete the old Surfaces and Lines which have been split. Attributes assigned to the split feature will 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
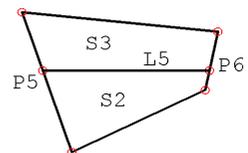
example shown here, the original Surface is split at Line 5. Points 5 and 6, defining Line 5, must lie on the original Surface boundary Lines.

❑ **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.
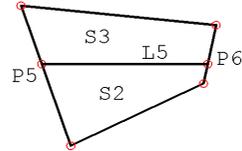
❑ **At a Point** Splits a selected Surface at a single Point defined on the surface inside the boundary. Four new Surfaces are created by manifolding straight Lines or arcs onto the existing Surface using the relative position of the splitting Point. In this example, the original Surface is split at Point 5. The resulting Surfaces (2-5), use parametric space to calculate boundary Point positions.
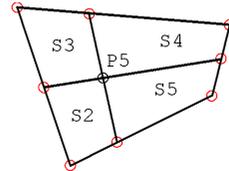
❑ **In Equal Divisions** Splits a selected Surface into separate Surfaces at a specified number of equal divisions. The direction of the split is expressed using local Surface axes. In the example shown here, the original Surface is split in its local x direction into equal divisions forming 3 new Surfaces.
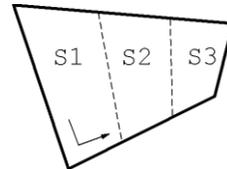
❑ **At Parametric Distances** Splits a selected Surface into separate Surfaces at specified parametric divisions. For example, specifying the parametric distances as 0.25 and 0.75 will split a Surface at the 1/4 and 3/4 position in parametric space in the specified local direction. In this example, the original Surface is split in its local y direction at parametric divisions 0.25;0.75.
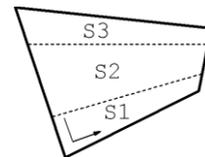
Joining

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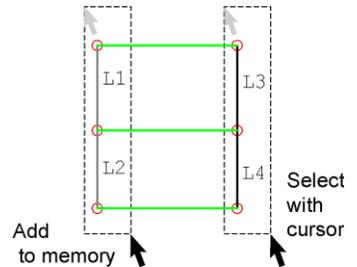
In the example shown Surfaces 1 and 2 are defined by first selecting Lines 1 and 2, then adding them to selection memory, then selecting Lines 3 and 4 and using the Surface by joining command.

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 create 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.
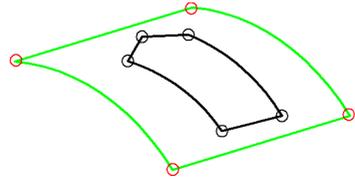
Extrusion

❑ **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.
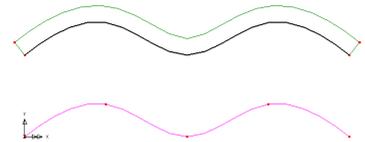
Volumes

About Volume Features

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

❑ **Solid Volumes** are defined by a number of connect 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** (a 4 sided volume with all
faces defined by triangular surfaces), **Pentahedral** (a 5 sided prism with top
and bottom faces defined by triangular surfaces and side faces defined by
quadrilateral surfaces), **Hexahedral** (a 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

Volumes can be defined by the following methods:

❑ 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).
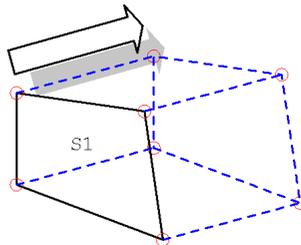
A surface is swept through a *translation* to create a Hexadedral.

A surface is swept though 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.
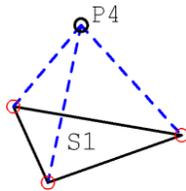
Surface 1 is joined to Point 4 to form a pentahedral.
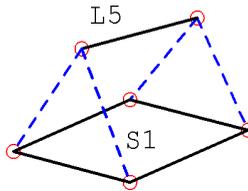
Surface 1 is joined to Line 5.
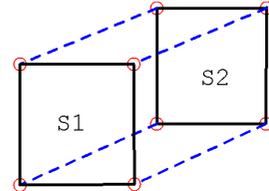
Surface 1 is joined to Surface 2 to form a hexahedral. Two triangular Surfaces joined this way would form a pentahedral.

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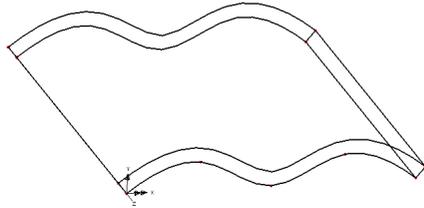
Coalescing volumes

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 a .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 Volumes 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.

Types of hollow volumes

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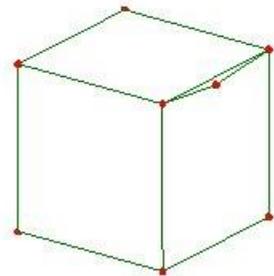
For the example shown, most of the lines form matching pairs. However, one of the defining surfaces has an edge that is defined by two lines, with a point in the middle. This pair of lines cannot be matched to the corresponding single line on the adjacent surface, and so this volume is classed as open.

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.

Defining hollow volumes

The user interface to enable the creation of hollow volumes may be invoked from the Advanced Dialog of the Geometry Properties dialog (accessed by **File> Model Properties> Geometry tab> Advanced**). 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.

Layout Grids

Layout grids are typically used to aid with modelling a 2D layout of a structure, but they can be copied to create multiple grid levels for a 3D model, as for example to provide a setting-out grid for each storey of a building frame. When layout grids are used points, lines and surfaces can be defined by cursor with reference to grid intersections by a variety of methods, doing away with the need to otherwise enter coordinate input to define these features.

Layout grids can be defined when creating a model from the New Model dialog, or at any time from the **Utilities> Layout Grid** menu item. Any number of layout grids can be created within a model. Once created, the data that defines each grid can be viewed, edited or copied, and set to be visible or invisible by accessing its context menu in the Utilities 🔧 Treeview. Grids are drawn by the Utilities 🔲 Layer.

Layout grid settings

Style

- ❑ **Rectangular** defines a grid of lines spaced at regular or varying distances in the X and Y directions.

  - **Skewed** (when checked) permits specifying a **Skew angle** for either set of lines in the X or Y direction.

- ❑ **Circular** defines a grid of lines spaced at regular or varying distances in the radial and circumferential directions.

  - **Straight circumferential lines** (when checked) draws the circumferential grid lines as straight lines as opposed to drawing arcs.

Origin and placement

The origin and the placement of the grid, the plane in which the grid will lie, and the rotation of the grid in that plane) can be stated.

Display Settings

❑ **Pen** settings permit the colour of the line and grid labels to be changed.

❑ **Drawing style** defines whether lines are drawn as Lines, Dots, or Lines and dots

❑ **Label settings** Grid labels can be specified for the grid lines in the X, or radial direction, and in the Y, or circumferential direction. Labels can be alphabetic or numeric, and be positioned at the top, bottom, top and bottom (both) or neither end of each grid line. Alphabetic labels will be auto generated as 'A', 'B', C' etc and increment to 'AA', AB' etc . An option to reverse the direction of the labelling is provided. By default grid labels are drawn flat to screen, with flat to grid (in the plane of the grid lines) an alternative option.

Grid data

For both styles of layout grid the **number of grid lines** in the X, or radial spacing, and in the Y, or circumferential spacing can be stated with respect to **fixed**, **absolute** or **incremental** spacings.

❑ **Fixed** means that a single value is used as the spacing between each pair of grid lines.

❑ **Absolute** means that each spacing is a distance from the origin.

❑ **Incremental** means that the first spacing is a distance from the origin, and all subsequent spacings are a distance from the previous grid line.

Examples of use
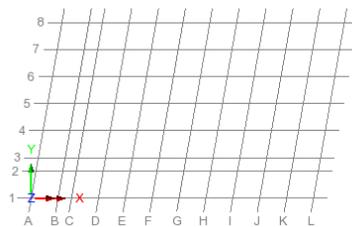
Rectangular skewed grid
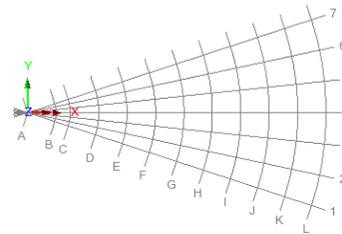
Circular grid (with an in-plane rotation angle)
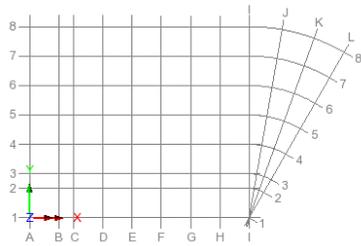
Rectangular and circular grids (in-plane)   Multiple rectangular grids (out-of-plane)
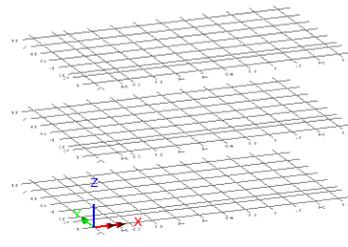
See also **Defining Lines and Surfaces by Grid**.

Defining lines and surfaces with respect to a layout grid

Defining single lines or surfaces

The **Geometry> Line> By Grid> Single** and **Geometry> Surface> By Grid> Single** menu items permit the creation of a single line or surface with respect to grid intersection points on a layout grid.

Defining multiple lines or surfaces

The **Geometry> Line> By Grid> Settings** and **Geometry> Surface> By Grid> Settings** menu items provide access to options to control how multiple lines or surfaces are created when defining those features with respect to a **layout grid** using the **Geometry> Line> By Grid> Multiple Horizontal** or the **Geometry> Surface > By Grid> Multiple Vertical** menu items.

By clicking and dragging the cursor to box-around a set of layout grid lines and points, lines or surfaces can be drawn either horizontally <u>in the plane of the grid</u>, or vertically <u>between grid layouts at selected grid intersections</u>.

For options relating to vertical features at least two vertically offset grids must be present with grid point intersections in alignment. Vertical features are always generated downwards from a selection made on a chosen layout grid to the lowest visible layout grid, so the selection must be made on the highest relevant layout grid to achieve a desired result.

Horizontal lines and surfaces

- ❑ **Break feature at each grid vertex** Creates a separate line or surface for each complete enclosed set of grid lines, and if the Alt key is pressed down at the same time, for each grid line intersected by the boxed-selection.

- ❑ **Create only one feature per mouse drag per grid** Creates just a single line or surface feature for each complete enclosed set of grid lines, and if the Alt key is

pressed down at the same time, for each set of grid lines intersected by the boxed-selection. When creating lines, if more than one row and more than one column of grid lines is selected, then a rectangle of four lines will be created between the outermost enclosed corner of grid points within the boxed selection.

Lines and Surfaces

❑ **Create feature(s) for nearest grid only** If checked, creates features only on the nearest layout grid. Otherwise, if this option is not checked and other grids exist with the same origin but in different levels, lines or surfaces will be created in all grid levels, providing the grid intersection points coincide.

❑ **Break vertical feature(s) at each grid** Creates a separate line or surface for each enclosed set of grid intersection points, and if the Alt key is pressed down at the same time, for each grid line intersected by the boxed-selection.

❑ **Create vertical feature(s) unbroken downwards to bottom visible grid** Creates just a single line or surface feature for each complete enclosed set of grid points/lines, and if the Alt key is pressed down at the same time, for each set of grid lines intersected by the boxed-selection.

Examples of use
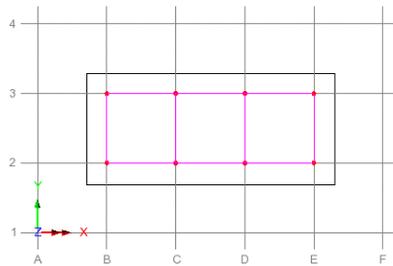
Define multiple horizontal lines by grid, with "Break feature at each grid vertex" selected (without the ALT key pressed)
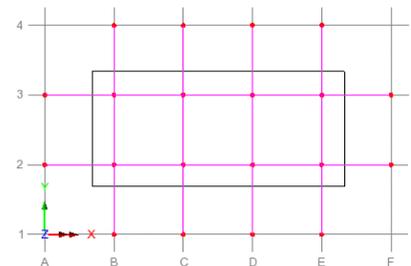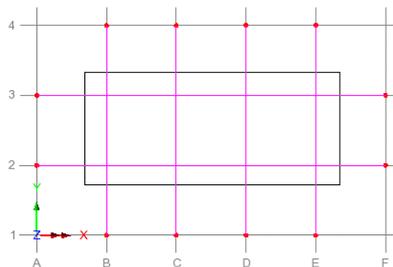
Define multiple horizontal lines by grid, with "Break feature at each grid vertex" selected (with the ALT key pressed)
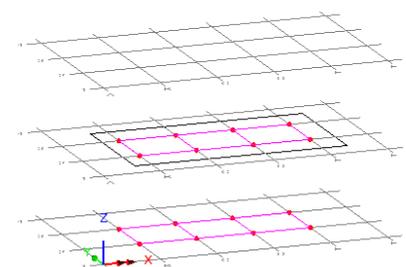
Define multiple horizontal lines by grid, with "Create only one feature per mouse drag per grid" (with the Alt key pressed)
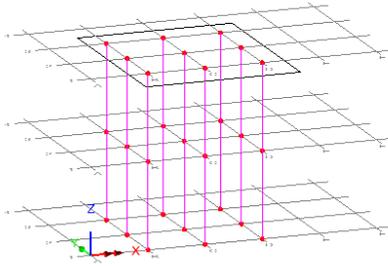
Define multiple horizontal lines by grid, by selecting a mid-level grid, with "Break feature at each grid vertex" selected (without the ALT key pressed)
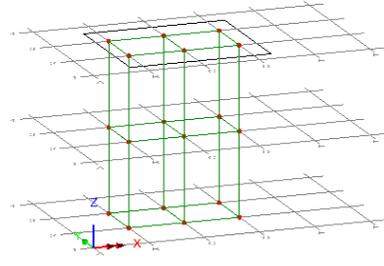
Define multiple vertical lines by grid, with "Break feature at each grid vertex" selected (without the ALT key pressed)
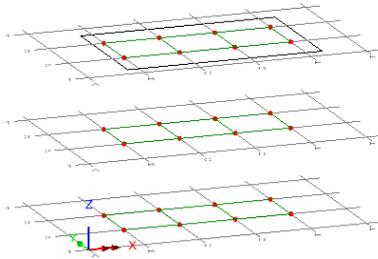
Define multiple vertical surfaces by grid, with "Break feature at each grid vertex" selected (without the ALT key pressed)
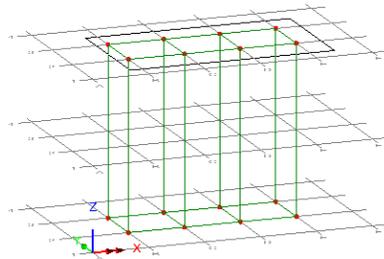
Define multiple horizontal surfaces by grid, with "Break feature at each grid vertex" selected (without the ALT key pressed)

Define multiple vertical surfaces by grid, with "Break feature at each grid vertex" selected, and "Create feature unbroken downwards to bottom visible grid" (without the ALT key pressed)

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.
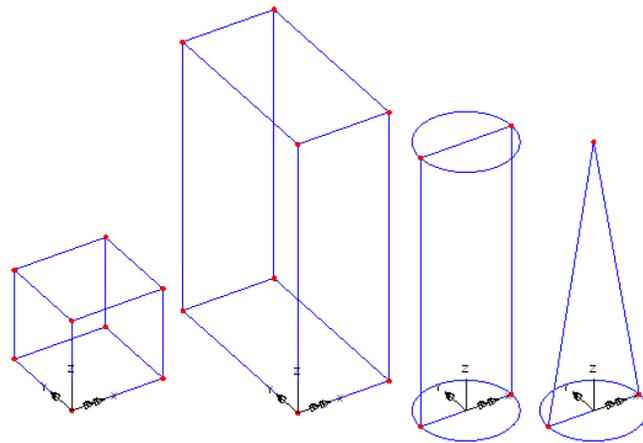
Notes.

- 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
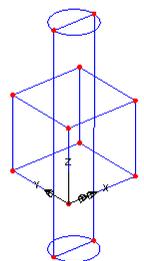
Separate (unconnected) cube and cylinder = two volumes

required. The slice plane may be defined in any global or local plane which may be visualised prior to the slice operation. A Surface of any shape may be used to slice a Volume by Subtracting the Surface from the Volume.
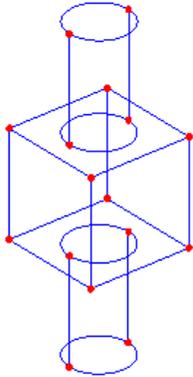
| Union of cube and cylinder (simplify internal geometry) = 1 volume | Union of cube and cylinder (no simplify internal geometry) = 4 volumes | Subtraction of cylinder from cube = 1 volume | Intersection of cube and cylinder = 1 volume |
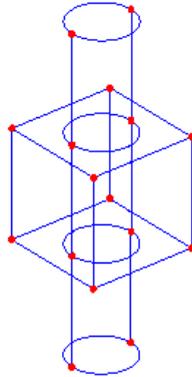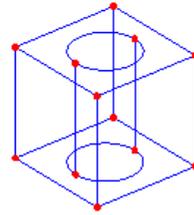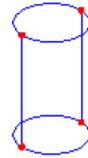
Geometry from Mesh

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.

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.
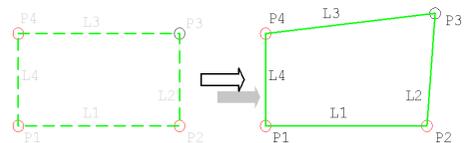
Moving can be used to separate features on a temporary basis to assist in the manipulation of features, for example when defining slidelines or joints.

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.
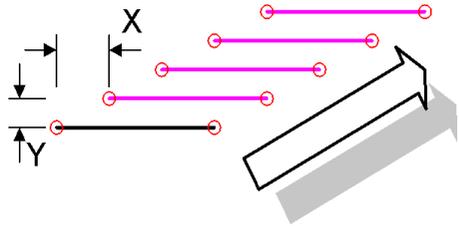
❑ **Sweeping Geometry** New geometry can be created by sweeping existing lower order geometry into higher order geometry using a transformation, as for example, by sweeping a Point to create a Line, or a Line to create a Surface, or a Surface to create a Volume. To do so a transformation or sweep type needs to be specified.
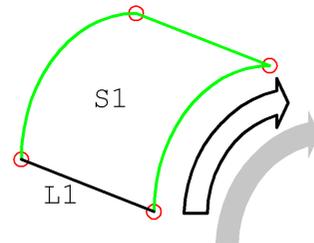
Note that when sweeping a line to create a surface, or a surface to create a volume, the surface axes or volumes axes of the newly generated surface of volume (and hence the axes of the elements within those features) may not align with the feature axes of the adjacent surface or volume, and will require re-orienting. See **Changing Geometry / Element Orientation** for more information.

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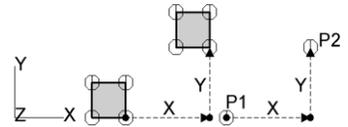
❑ **Rotation** (in a global plane) A specified angular rotation in either the XY, YZ, or ZX global plane at a specified origin.

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.
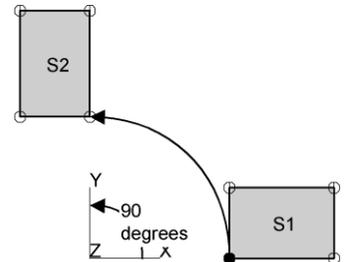
❑ **Mirror** A mirror plane may be defined by specifying three points in space to define an arbitrary plane, or two points to define a plane parallel to either the Z, Y, or X axis. Three Points added to selection memory can be used to define an arbitrary mirror plane, and two Points added to selection memory can be used to define a mirror line in the XY plane (not the YZ or ZX planes)
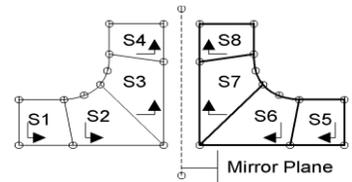
Note:.Care should be taken when mirroring Lines and Surfaces as their orientations may be reversed so some Surfaces may effectively be turned upside down. Lines may also point in the opposite direction. See **Changing Geometry Orientation** to resolve any problems.

❑ **Scale** A scale factor and the origin point of the scale is specified .
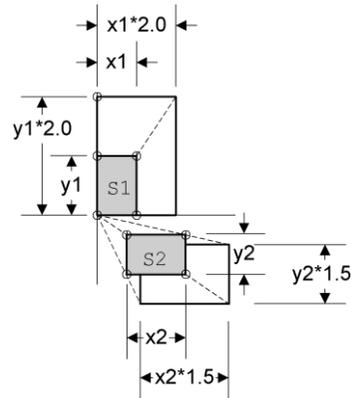
❑ **Matrix rotation** A direction cosine either specified directly, or using two or three Points added to selection memory. Points in selection memory define a plane, the rotation of the global XY plane to this new plane defines the transformation. Note. If the determinant of the matrix is not unity then the effects may not be as desired.
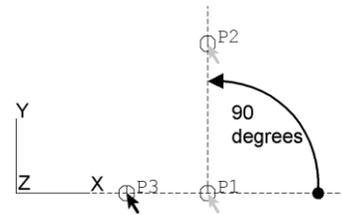
This example defines a rotation by defining a plane relative to the global Cartesian axis set by indicating three Points: at the origin (P1) along the local x axis (P2) and defining the local xy plane (P3). The resulting transformation is a rotation of 90 degrees about the global axes.

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 Tolerance

P1

P2

P3

P4
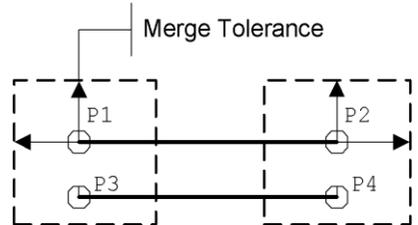
```
P1: LDG1, SUP1, EQV1 ...
P3: LDG1, SUP1, EQV1 ...
```

❑ **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.

```
L1: MSH1, MAT1, GEOM1 ...
L2: MSH1, MAT1, GEOM1 ...
```

P1   L1,L2   P2

❑ **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 ...
```

P4   L3   P3

L4   L2

S1,S2

P1   L1   P2

Merge Case Study 1

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

```
Line 1              P2
                 L1  ○
              MAT1
              MSH1
P1
○
```

Merge

```
Line 2              P2
                 L2  ○
              GMP1
              LDG1
P1
○
```

Merge Characteristics   **After Merging**

Off → No merging takes place

Exact → No merging takes place

Wild →

```
Merged and              P2
attributes           L1  ○
transferred
              MAT1
              MSH1
P1            GMP1
○             LDG1
```

Ignore Assignments →

```
Merged and              P2
attributes           L1  ○
transferred
              MAT1
              MSH1
P1            GMP1
○             LDG1
```
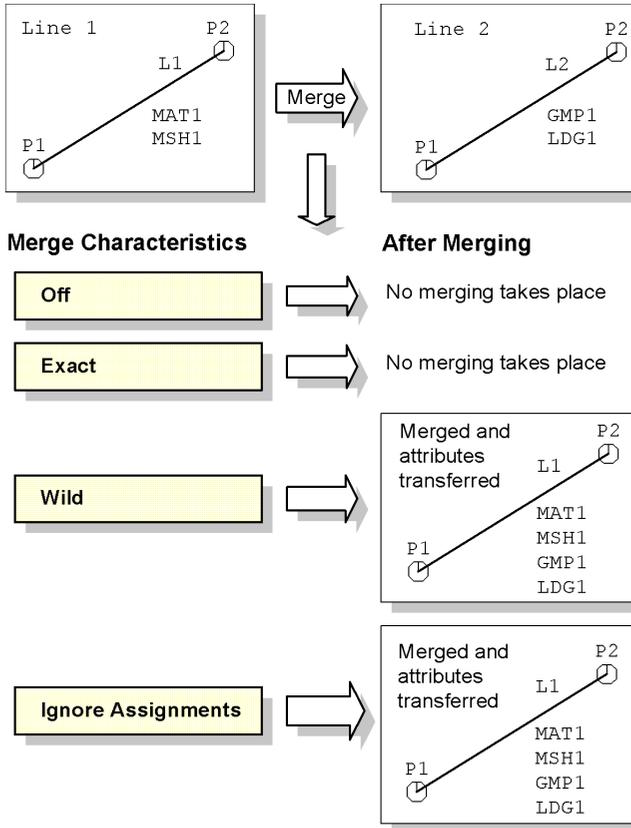
Merge Case Study 2

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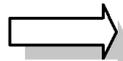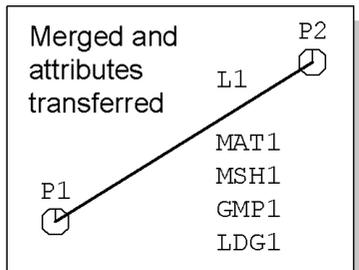
Using the Unmerge Commands

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.
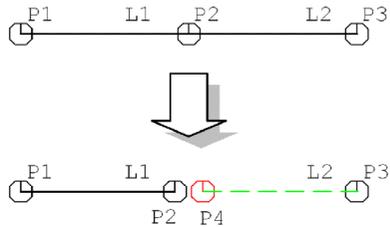
❑ **Unmerge Point in Surface**Unmerge Point 5 in Surface 2. A new Point 8 is copied from Point 5, a new Line 9 is defined and Line 6 is redefined using Point 8. New features are set to be unmergable. Point 5 and Point 8 are coincident.
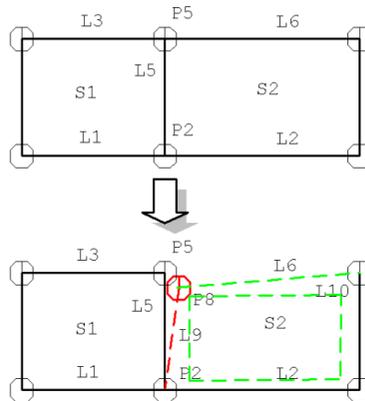
❑ **Unmerge Point in Volume**Unmerge Point 1 in Volume 1. A new Point, 2 new Lines and a new Surface are defined and affected Surfaces and the Volume are redefined. The new Point is coincident with Point 1.
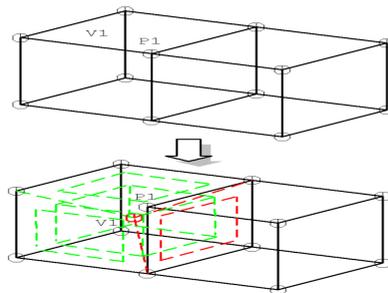
❑ **Unmerge Line in Surface**Unmerge Line 5 in Surface 2. A copy of Line 5 is defined which replaces Line 5 in the definition of Surface 2. Points are unaffected.
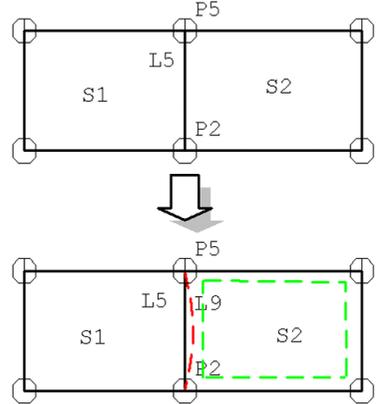
❑ **Unmerge Line in Volume**Unmerge Line 1 in Volume 2. A copy of Line 1 is defined joining the same end Points The new Line replaces Line 1 in the definition of the affected Surfaces in Volume 2
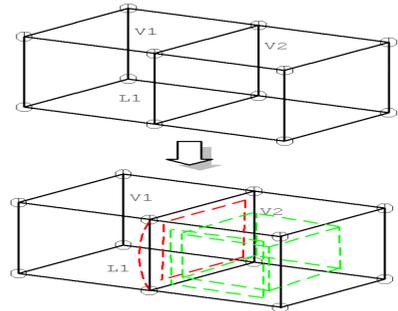
❑ **Unmerge Surface in Volume**Unmerge Surface 1 in Volume 2. A copy of Surface 1 is defined which replaces Surface 1 in the definition of Volume 2.

161
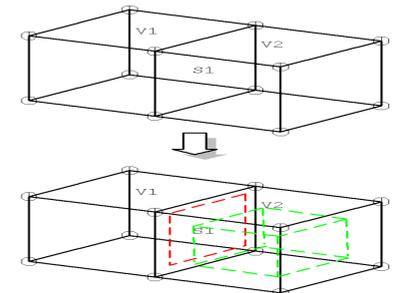
Case Study. Forcing Features to Merge

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.
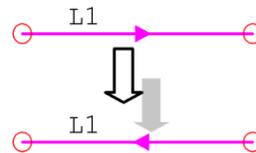
In the Surface example the local x axis remains along the first Line in the Surface definition, but the Surface normal is reversed. Elements meshed on this Surface will be inverted.
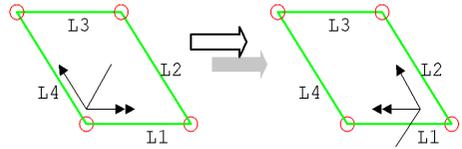
❑ **Axes to Surface** This command reorders the Surfaces that define a Volume such that the selected Surface becomes the first Surface in the volume definition. The z axis of the Volume is orientated normal to the first Surface, and pointing from that Surface towards the middle of the Volume. When the 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.

Possible Volume axis before alignment with Surface axis

Surface Axis

Volume axis already aligned with Surface axis

Volume axis after alignment aligned with Surface axis
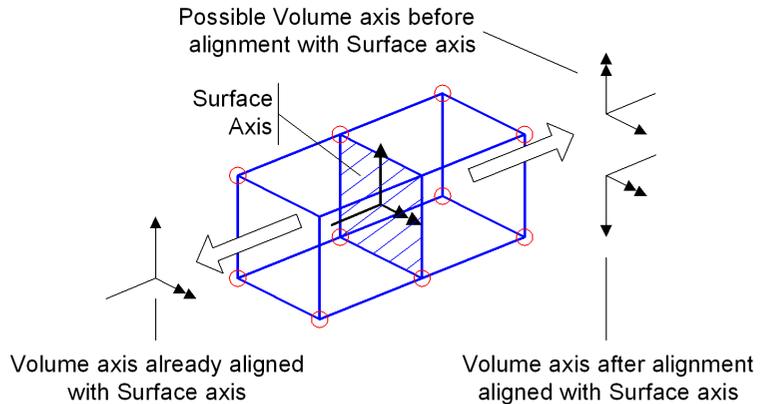
❑ **Cycle** Surfaces (or surface/solid elements in a mesh-only model) may have their definition order cycled, and Volumes may have the definition of the first Surface in the Volume definition cycled (and for mesh-only models, Elements may have the definition of the first Face in the Element definition 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.
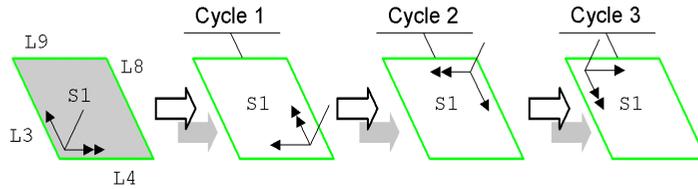
- ❑ **Cycle Relative** Cycles the definition order relative to the first feature of the same type in the current selection. Lines may have their local x-axes aligned to a chosen Line, and Surfaces and Volumes (or surface/solid elements in a mesh-only model) may be reoriented in this way. In this example Surfaces 2, 3 and 4 are cycled using Surface 1 (shown greyed) as the reference orientation.
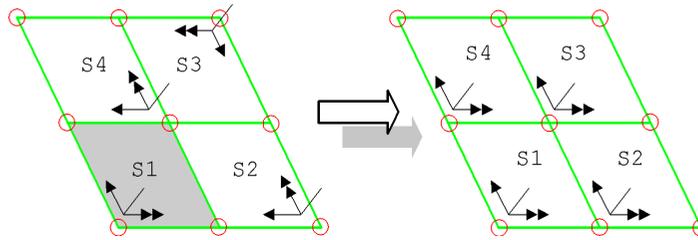
- ❑ **Cycle to Neighbours** (Mesh-only models) Cycles the element axes of all neighbouring and selected elements relative to the first element in the current selection. Whilst similar in effect to the Cycle Relative option for the example shown above (which would obtain the same result for that particular 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.
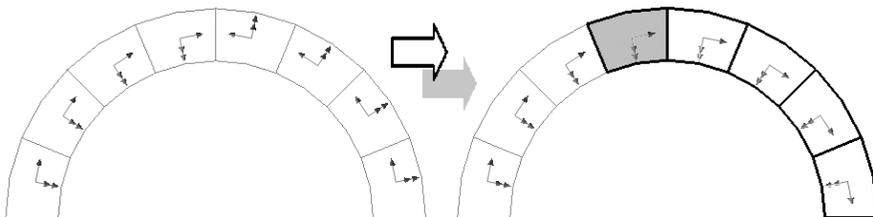
Cycle to Neighbours (correctly aligns element axes of the selected elements with initially selected neighbouring element)
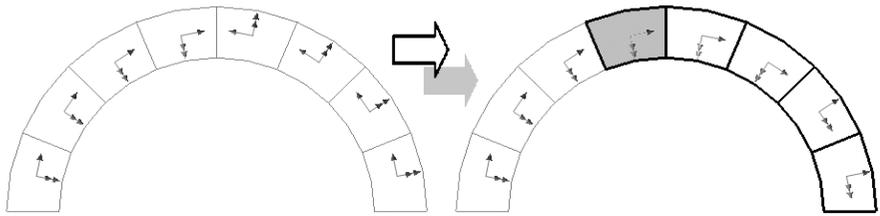
Cycle Relative (only aligns element axes of the selected elements to best angle with respect to initially selected neighbouring element)

❑ **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.

Measuring Line Geometry

Three measurement facilities are available to report dimensional data about lines. These are accessed from the **Geometry> Line** menu. They are:

- ❑ **Distance between lines**
- ❑ **Angle between lines**
- ❑ **Length of line**

In each case the lines to be measured should be selected prior to the selection of the corresponding menu command.

Note. **Angle between lines** will always report the smallest angle between the two lines selected, regardless of their relative position, direction or selection order.

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 shown in the Attributes 🍄 Treeview. They can be assigned to selected 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** attributes describe the element type and discretisation on the geometry.

❑ **Geometric** attributes specify any relevant geometrical information that is not inherent in the feature geometry, for example section properties or thickness.

❑ **Material** attributes define the behaviour of the element material, including linear, plasticity, creep and damage effects.

❑ **Support** attributes specify how the structure is restrained. Applicable to structural, pore water and thermal analyses.

❑ **Loading** attributes specify 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** attributes 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.

❑ **Activate and Deactivate** 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.

❑ **Search Area** restricts discrete (point and patch) loads to only apply over certain areas of the model.

❑ **Age** defines the time between creation and activation of features in 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.

Results Processing Attributes

These attributes can be assigned to the model before or after an analysis in order to modify or obtain results of the chosen type.

❑ **Results transformation** attributes allow results to be transformed (orientated) to a consistent direction.

❑ **Inspection locations** are defined points of interest on the model for results extraction / viewing purposes.

❑ **Wood-Armer** attributes contains details of slab reinforcement arrangements.

❑ **Steel Frame Design** attributes are used to specify additional information about model features in order to enable design checks to supported design codes.

❑ **RC Slab Design** attributes are used to specify the reinforcement arrangement to be used throughout, or in a specified region of a model.

Design Factor Plot Attributes

❑ **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.

⚬ An attribute has been assigned to the model (coloured image)

⚬ An attribute has yet to be assigned to the model or used in the definition of another attribute (grey image)

⚬ A black dot on a coloured attribute icon shows that an attribute, which has been assigned to the model, has been set active (for example setting active a local coordinate assignment).

⚬ A black dot on a grey attribute icon shows that an attribute, which has yet to be assigned to the model, has been set active.

━ Fibre in a structural analysis, or active lamina in a composite analysis.

━ Active fibre in a structural analysis, or active lamina in a composite analysis.

⊞ An attribute icon (coloured or grey) 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.

⊘ A 'Not valid' symbol shows that an attribute assignment is pending or invalid in some way, requiring investigation and correction.

⚠ A warning symbol will appear in the following situations:

- If any attributes that were previously created have been superceded by a change to a model. In this case a model must be re-solved to re-create those attributes..

- If attributes defined are not compatible with a stated **analysis category**.

⚙ Direction definition options

Attribute manipulation

Attributes are manipulated using the context menu in the Attributes ⚬ Treeview (accessed by clicking the right mouse button on the attribute). The following attribute-specific menu options are available:

General editing

- ❑ **Edit Definition** Allows the properties that were originally defined to be modified on the dialog that was used to create them.
- ❑ **Edit** / **Edit Attribute** Allows the properties of the attribute to be modified. Note that for particular structural loading attributes, editing the properties of multiple load attributes of the same loading type is possible. See **Editing of Multiple Loading Attributes** for details.
- ❑ **Copy** Enables the attribute to be copied and assigned to a selected group, or view window in the current model using the **Paste** option.
- ❑ **Rename** Attributes can be given meaningful names, for example 'Steel' to describe a material, or 'Beam - 4 Divisions' to describe a Line mesh.
- ❑ **Delete** Deletes the selected attribute. See **Deleting attributes from the Treeview** below.

Assignment editing / visualisation

- ❑ **Edit Assignments** Simultaneously view, edit and manage any number of attribute assignments. Editing of data can be by individual selection, or by selection of the parent folder containing a set of attributes. For loads, editing of multiple assignments is only possible for selected loads of a common type.
- ❑ **Select Assignments** Selects in the view window the features that have the selected attribute assignment.
- ❑ **Deselect Assignments** Deselects from the view window the features that have the selected attribute assignment.
- ❑ **Visualise Assignments** Switches on and off in the view window 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.

Visibility

- ❑ **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 plotting

- ❑ **Results Plots** permits results for selected attributes to be selectively plotted.

Temperature

❑ **Temperature dependence** Allows editing of temperature dependent properties of material attributes.

Section property updating

❑ **Update from library** Updates all current section properties for the selected geometric attribute with those held in a library for a section of the same name

Prestress only

❑ **Create discrete loads** Creates a duplicate analysis with loadcases of the same names as those in a stated analysis, and as many discrete load attributes, search area attributes, and assignments as necessary to copy and represent the prestress data for the tendon loading attribute(s) selected.

❑ **View forces and moments** Shows the discrete loading values (forces and moments) calculated by the prestress loading facility for the attribute tendon load assignment selected in the Analyses treeview.

Making assignments

❑ **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.

Attribute names are added to the Attributes treeview in creation order, but can be re-ordered by using the **Sort by name** context menu item accessed from the Attributes tab.

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.

Editing of multiple loading attributes

For particular structural loading attributes, editing of the properties of multiple load attributes of the same loading type is possible. See **About Loading**.

Currently only editing of multiple load attributes of the same type is supported. Note that all other non-load attributes require editing individually.

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 operation 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 selected 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.

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.

❑ **Hidden parts** Displays all elements and not just the visible element faces. **Dotted** displays the hidden edges in a dotted line.

❑ **Show nodes** Draws the mesh nodes. Nodes define the vertices of elements.

❑ **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. The **Threshold (deg)** value controls when edges of elements are drawn. Edges of adjacent elements are drawn when the angle between the normals of adjacent elements exceeds the threshold value.

❑ **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**.

Mesh Visualisation settings

- ❑ **Joint elements** Marks any joint elements in the model with a symbol (user-definable).
- ❑ **Active mesh** Marks the active elements with a symbol (user-definable).
- ❑ **Point mass elements** Marks the point mass location with a filled circle of a user-definable size.
- ❑ **Beam end releases** Draws the beam end release for elements that use end releases.
- ❑ **Rigid zones** Draws the extent of any **rigid zones** in a thicker and darker mesh colour.

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.

Note that the inclusion of rigid end zones in a line mesh attribute does not change the number of line mesh divisions defined or reported for that element in the Line Mesh Spacing dialog. The defined number of line mesh divisions is fitted to the length of line that remains if any rigid end zones have been defined.

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.

Internal points

For thick beam elements, and for steel frame or RC frame design checking use only, each element is discretised into ten divisions for calculation and results viewing

purposes. This produces nine equally-spaced internal points along each element with two end points coinciding with the nodal locations of the element.

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.

Beam End Conditions

Included Joints and End Releases for Beam Elements

Joints can be included at the ends of each Line for supported element types (grillage element and all 3D beam elements) by choosing to include a joint as part of specifying an end condition for a line mesh attribute. The joints are automatically selected to best suit the non-rigid element type. When a joint is inserted an additional node is added at a zero distance from an end node. If joints are included but no joint material properties are assigned to them, default properties are used instead using the material and geometric properties that are assigned to the Line. If required, different joint material properties can be defined and assigned to relevant ends of lines to overwrite the default joint values. So, in any given analysis, it is possible for any line to have three material assignments – one for itself, one for the joint at its start, and one for the joint at its end. The latter two must be joint materials, and must be assigned using the 'Assigned to line end(s)' option when the assignment is made.

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 Conditions button and set appropriate end releases. 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.
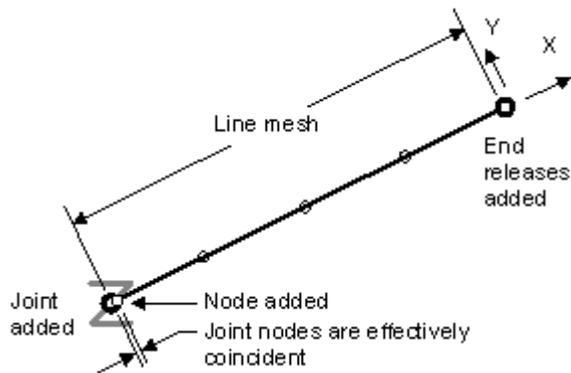
Included joints and end releases for beam elements

Rigid Zones for Beam Elements

When modelling the connections between beams and columns in a building frame, a short rigid length of beam element may be required to model the length of the beam that extends from the nodal intersection with one beam or column, to a position on the beam that defines the extent of a rigid zone for modelling and analysis purposes.

When rigid zones are defined additional rigid elements (of the same type as the line element itself) are inserted at the start and / or end of each line mesh 'set' of spacings, but the inclusion of rigid end zones in a line mesh attribute does not change the number of line mesh divisions that were defined for that element. Released freedoms are not tabulated at the end of the line to which they apply, but at the start of the non-rigid elements, for which the line mesh spacing that is defined is applied to. For more details see **Rigid Zones.**

Note that a line meshed with rigid end zones may not be used to define a meshed surface.
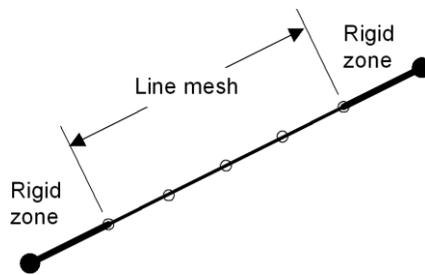
Rigid zones for beam elements

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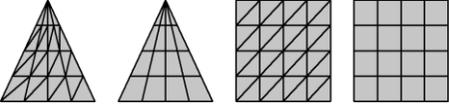
❑ The Surface mesh may be graded using mesh spacing parameters in 'None' element Line meshes assigned to the boundary Lines. In the examples shown here mesh spacing has been used to bias the elements into the apex of the triangle or one corner of the rectangle.
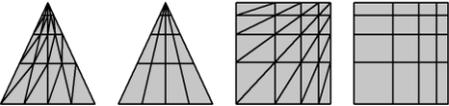
Irregular Surface Meshing

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
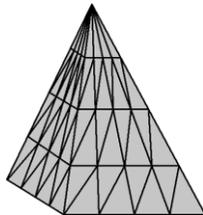
Tetrahedral Elements | **Pentahedral/Tetrahedral Elements**
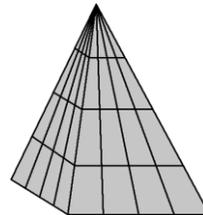
Pentahedral Volumes
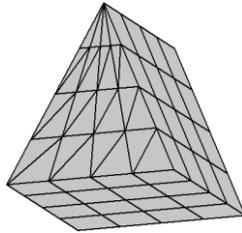
Pentahedral Elements   **Hexahedral/Pentahedral Elements**
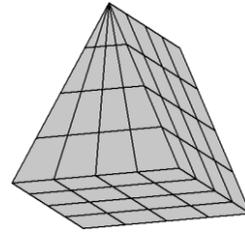
Hexahedral Volumes
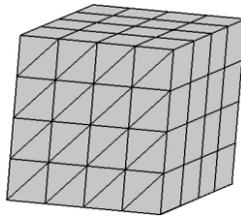
Pentahedral Elements   **Hexahedral Elements**   **Hexahedral/Pentahedral Elements**
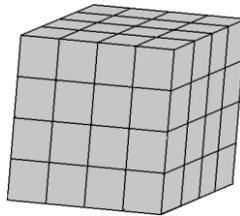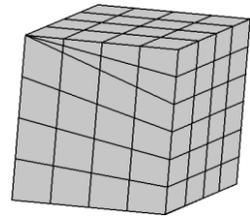
The mesh density for volumes is taken from the boundary surface mesh density.

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.
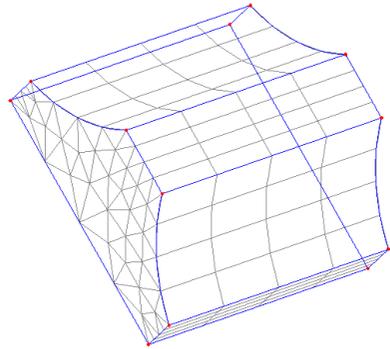
Case Study: Meshing Volumes by Extruding Irregular Surfaces

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.
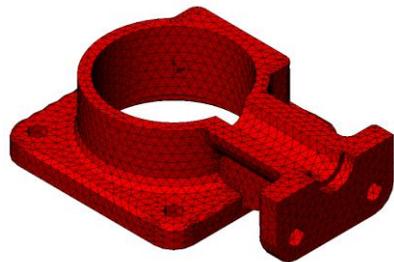
Notes

- 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.
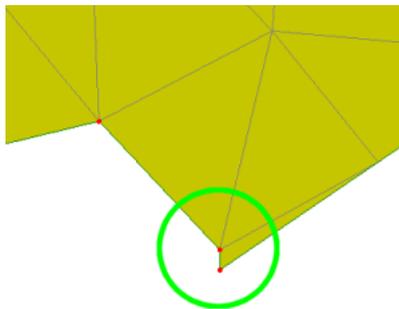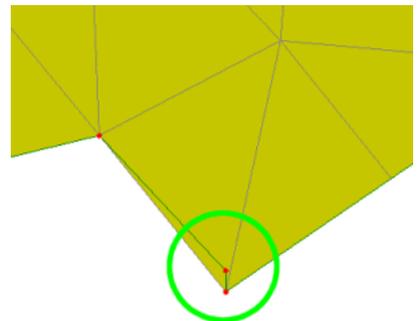
Mesh before edge collapsing   **Mesh after edge collapsing (short edge removed)**
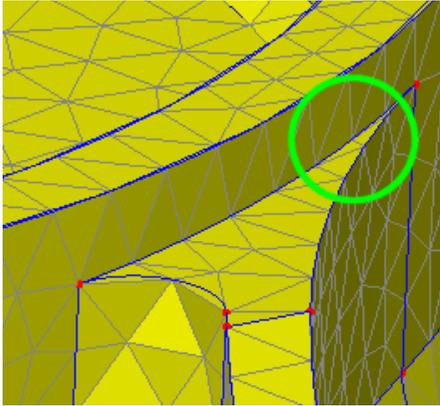
Mesh before edge collapsing
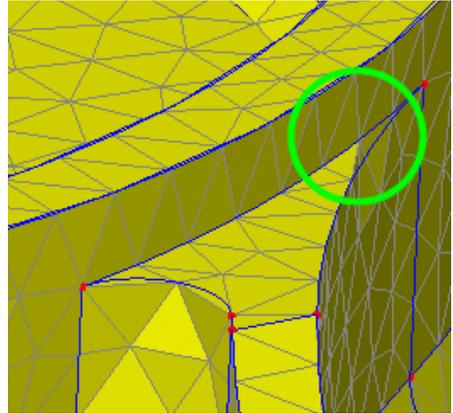
Mesh after edge collapsing
(Elements with small subtended angles removed)

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).
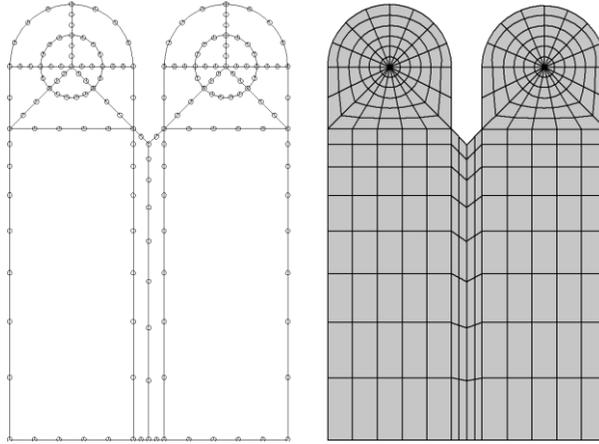
- **Irregular Surface Meshing The applied boundary discretisation (top) produces the irregular mesh pattern on the Surface (bottom).**
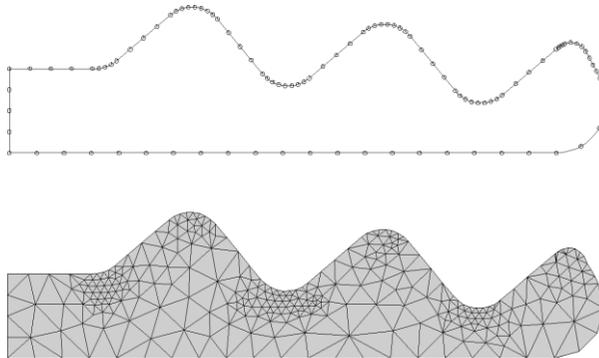
Default Number of Mesh Divisions

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.

With a Surface or Volume already drawn and selected, a background grid can be defined by selecting the **Utilities> Background Grids** menu item.
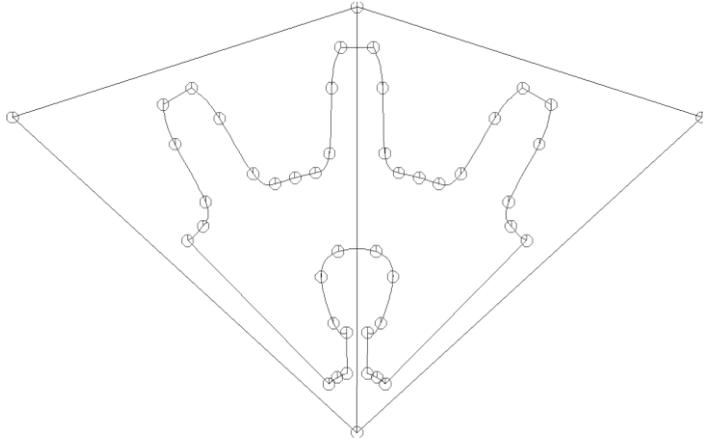
Constant Mesh Spacing	**Varied Mesh Spacing**
Same spacing parameters (Point meshes) are assigned to all Points in background grid.	Different spacing parameters (Point meshes) are assigned to the top Points (spacing=7) and the bottom Points (spacing=1) in the background grid.
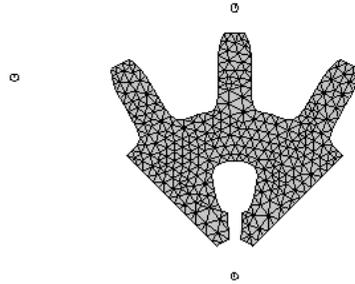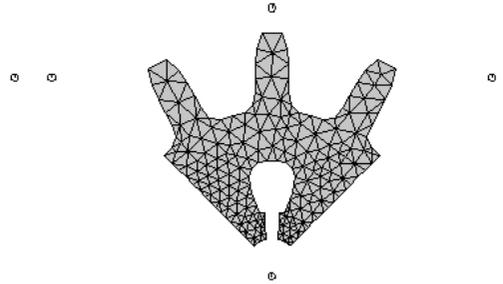
Notes

- 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.
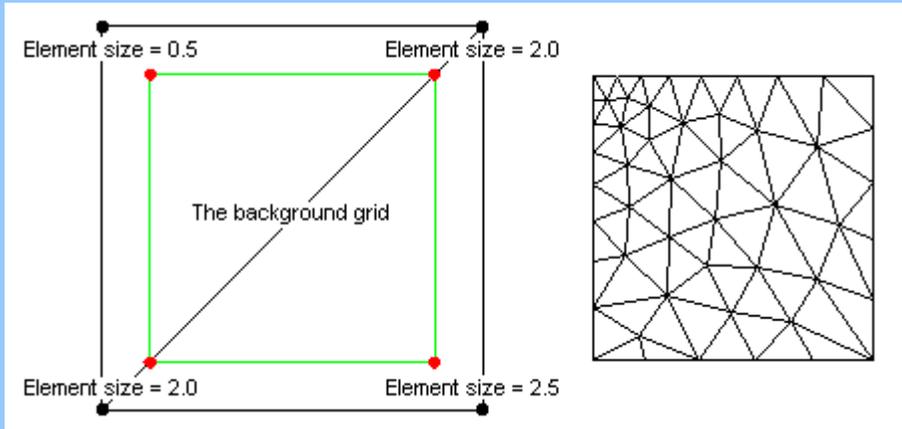
Composite Material Assignment

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.
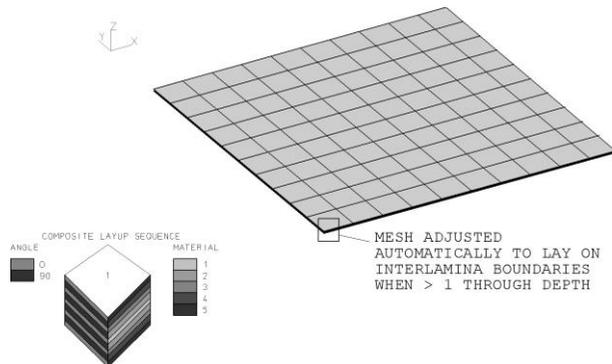
Connecting Beam and Shells

Beam Shell Connectivity

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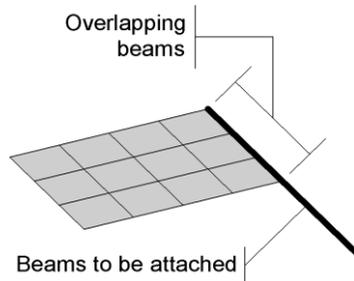
Beam Solid Connectivity

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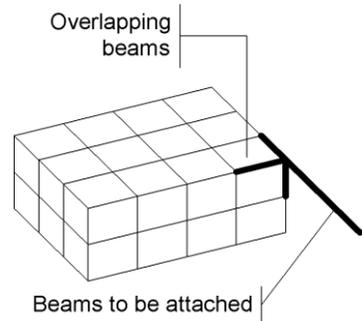
Shell Solid Connectivity

Extend the shells over a portion of the solids indicated by the dark shaded area.
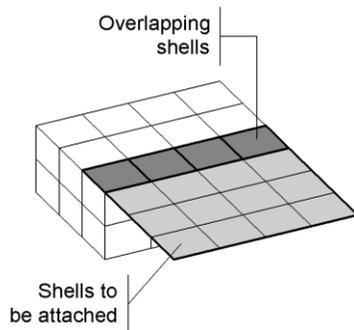
Case Study: Connecting Shells and Solids

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.
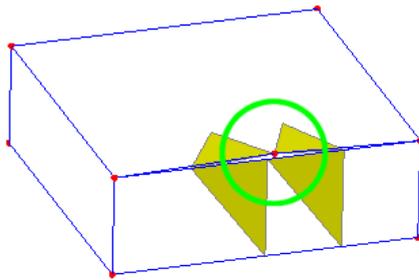
Unmerged nodes on volumes which failed to mesh.
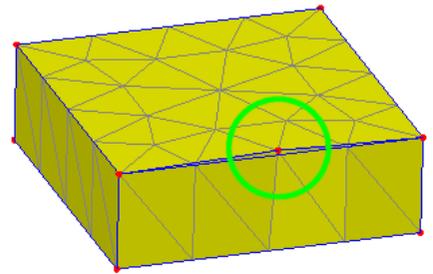
Mesh with nodes merged by adjusting node merge tolerance

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.

Modelling cables

Modelling of cables generally requires modelling of tension-stiffening behaviour where the axial stiffness of the cable is dependent upon the sag of the cable, which in turn is dependent upon the tension in the cable. As a result modelling of cables generally requires a geometrically nonlinear analysis that may involve a number of nonlinear behaviours such as:

❑ **Large displacements** Cable-supported structures are often highly stressed and relatively light. Consequently large displacements can occur that alter the global geometry of the structure. These must be taken into account if the results from the model are to be meaningful.

❑ **Stress stiffening in each cable** Long inclined cables "sag" under dead load, with the sag decreasing when cable tension increases. The stiffness of the cable becomes greater as it is straightened.

❑ **Tension-only behaviour** Where short cables can be idealised as either having full stiffness in tension and zero stiffness in compression, this behaviour can be included in the model in preference to the more complex stress-stiffening approach.

However, in some cases, cables may be adequately represented with linear assumptions.

A number of different approaches may be used to model cables either considering or ignoring any initial sag in cables. An overview of the elements available and modelling and analysis methods follows, along with compatibility considerations for use with cable tuning facilities.

Modelling cables with sag

Modelling of cables when cable sag is present requires a nonlinear analysis and can be done using the following elements and methods:

❑ **Beam elements with GNL** For this, lines representing cables are modelled using beam elements in conjunction with a geometrically nonlinear analysis. In order to take both large displacements and stress-stiffening into account, a full nonlinear approach should be used. The lines representing the cables are assigned a thick nonlinear beam (BMI21) mesh attribute, with sufficient divisions in order to model the sag of the cable under transverse loads (such as self weight). Initial stresses in the cables can be added, using a structural initial stress/strain load (or using a structural temperature load if preferred) - see "Structural Loading" on page 360 for details. It would be necessary to model the cable with a line representing the initial sag if the tensions are relatively low. Nonlinear controls must be set, including geometric nonlinearity. It is suggested to use BMI21 or BMI31 elements with the co-rotational option set (by visiting the Nonlinear and Transient control options). Better results may be obtained if the Total Lagrangian option is also switched on. Note that Total Lagrangian option should not be used isolation.

Modelling cables without sag

Modelling of cables without sag using any of the approaches below is an approximation, which must be selected on a case-by-case basis. Linear assumptions are only applicable under conditions where the cables are in tension for all design conditions; where tension remains always high, typically of the order of 30% of the Ultimate Tensile Stress; or when the horizontal offset of the cable end is small in relation to the overall length of the cable.

- ❑ **Fully linear** For this, lines representing cables should be assigned a linear material, and should be meshed with only one 2D or 3D bar or beam element per line. The mesh attribute should have one mesh division, avoiding unrestrained nodes along the length of the cable, which would lead to unrestrained lateral deformations in teh case of bar elements. Initial stresses/strains in the cable may be optionally determined by running a linear cable tuning analysis that calculates load factors for selected lines in a model that represent cables in order to achieve defined target values set for various feature types and results components. For a linear cable tuning analysis no nonlinear controls or assignments must be present. The approximation used for the cable stiffness will be unchanged by the tensioning of the cable.

- ❑ **Tension-only but otherwise linear** In some cases, short cables can be idealised as having tension-only behaviour. Lines in a model representing cables should be assigned a 2D or 3D bar element mesh attribute as appropriate. The bar mesh attribute should have one mesh division, avoiding unrestrained nodes along the length of the cable, which would lead to unrestrained lateral deformations, and the behaviour is modelled using **tension / compression only bar material**. Nonlinear controls must be set, but geometric nonlinear formulations do not need to be invoked because the behaviour of the bars will be linear when in tension. This approach does not take into account either large displacements or stress stiffening in each cable but may be acceptable in certain applications.

- ❑ **Linear with large end displacements** The large displacement behaviour of cables in which there is no sag (such as in vertical stays (hangers) or short, highly stressed inclined cable stays) can be taken into account with a simplified nonlinear approach. The lines representing cables may be assigned a 2D or 3D bar element mesh attribute as appropriate. Generally the bar mesh attribute should have one mesh division, avoiding unrestrained nodes along the length of the cable, which could lead to unrestrained lateral deformations. Initial stresses in the cables/stays/hangers can be added, using a structural initial stress/strain load (or by using a structural temperature load if preferred) - see "Structural Loading" on page 360 for details. A nonlinear cable tuning loadcase can also be used to apply initial stresses. Nonlinear controls must be set including geometric nonlinearity (the Total Lagrangian option is appropriate for bar and beam elements). This approach will not reflect stress stiffening in each cable if any cable sag is present.

Compatibility of cable modelling approaches with cable tuning facilities

Cable modelling approach	Use with linear cable tuning	Use with nonlinear cable tuning loadcase
Beam elements with GNL	Not valid	Valid
Fully linear	Valid	Valid
Linear other than tension-only behaviour	Not valid	Valid
Linear with large end displacements	Not valid	Valid

See **Linear cable tuning** and **Nonlinear cable tuning loadcase** for more details.

Worked examples

See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Linear cable tuning analysis of a pedestrian bridge', which shows how linear cable tuning analysis using bar elements is carried out for a simple structure.

See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > '3D Nonlinear Static Analysis of a Cable Stayed Mast' for an example in using beam elements with geometric nonlinearity (without sag).

Rigid Zones

When modelling the connection between beams and columns in a steel or concrete building frame, a short rigid length of beam element may be required to model the length of the beam that extends from the nodal intersection with one beam or column - the 'intersection of interest', to a position on the beam that defines the extent of a rigid zone for modelling and analysis purposes.

Rigid zones can be specified when defining a Line mesh attribute using the **Attributes > Mesh > Line** menu item. Different properties can be defined for the start and/or end of a line mesh attribute, and rigid zone lengths can be user-defined, or calculated according to a formula based upon the size and eccentricity of intersecting beam and column members.

Additional rigid elements (of the same type as the line element itself) are inserted at the start and / or end of each line mesh 'set' of spacings, but the inclusion of rigid end zones in a line mesh attribute does not change the number of line mesh divisions that were defined for that element. The defined number of line mesh divisions are fitted to the length of line that remains when any rigid end zones have been defined. Rigid zones receive the same loading as the non-rigid elements assigned to a line. A rigid zone factor specifies the degree of rigidity of the rigid end zone element that is

inserted at the end of a beam element. The value is constrained between 0 (fully flexible) and 1 (fully rigid).

When rigid zones are created, the released freedoms are not tabulated at the end of the line to which they apply, but at the start of the non-rigid elements, for which the line mesh spacing has been defined. The following image shows line meshing options related to rigid zones.
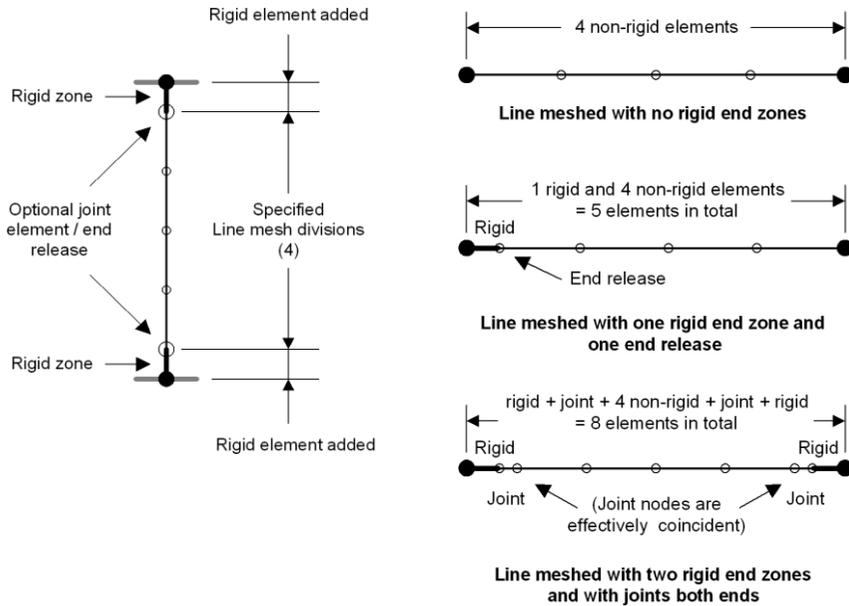
Line meshing options showing introduction of rigid zones for columns and beams

Examples of line meshing with rigid zones

Joints (defined at rigid end zones)

Joints can be optionally included at the transition between the rigid zone element and the non-rigid elements as part of defining the end conditions for a beam. These joints, which are automatically inserted by Modeller, are chosen according to the non-rigid elements to which they connect. The inserted joints are given default linear joint material properties (also based upon the material and geometric properties of the line on which they occur) sufficient for a suitable rigid connection to be achieved. Note that the attributes for joints created in this way do not appear in the Attributes treeview. If required, different joint material properties can be defined and assigned to relevant ends of lines to overwrite the default joint values. Point-type joint material properties can be assigned to the ends of lines with rigid zones in addition to the material properties assigned to the line for the beam elements. Different joint properties may be assigned to each end of the line or the same properties may be assigned to both ends according to the selection made on the assignment dialog. So, in

any given analysis, it is possible for any line to have three material assignments – one for itself, one for the joint at its start, and one for the joint at its end. The latter two must be joint materials, and must be assigned using the 'Assigned to line end(s) option when the assignment is made.

Rigid end zone properties

The material properties of the elements automatically added to represent the rigid zones are set to be the same as those of the adjoining non-rigid elements unless the density is to be excluded as specified by the 'Include mass' check box on the dialog shown on the Rigid zone page of the Line Mesh End Conditions dialog.

Definition or calculation of rigid zone length

Rigid zone length calculation is only for use with vertical lines representing columns that intersect with horizontal beams at 'intersections of interest'.

- Columns, for the purposes of rigid zone calculation, are lines that lie along the **vertical axis** defined for the model, that lie at intersections of interest. When considering the rigid zone of the column itself, the column may be above or below the intersection.

- Beams, for the purposes of rigid zone calculation, are straight lines that meet a vertical line representing a column at an intersection of interest. Non-horizontal lines may meet at the intersection, but these will be ignored, and if they are marked as needing their rigid zone length to be automatically calculated, an error will be produced. When considering the rigid zone length of a beam, the column considered is always below the intersection, unless there is no column below but there is a column above. in which case the column above is used in any rigid zone length calculation made for the beam.
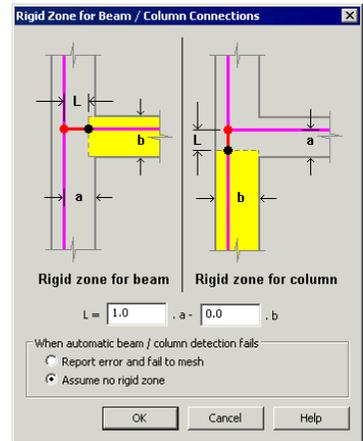
The following options are available to define a rigid zone:

- ❑ **None** No rigid zone is to be defined.
- ❑ **Specified Length** A user-specified rigid zone length
- ❑ **Beam / column connection** A formula based calculation based on the width and depth of connecting column and beam members can be defined using the **Specify...** button. For example, with reference to the dialog image right, a rigid length (L) of a beam could be defined as being 100% of the distance from the

nodal line to the face of a column (a) minus 25% the beam's own depth (b), as in L=(1 x a) - (0.25 x b).

One of two options inform Modeller what to do when automatic beam /column detection for the above option fails:

- **Report error and fail to mesh** Any error will be written to the text output window, describing the condition(s) that prevent(s) rigid zones being calculated for the features selected.

- **Assume no rigid mesh** For ends of columns, such as those at the foot of the structure, no rigid zone will be assumed since no beams frame-in to the column at these locations.

❑ **Rigid zone factor** specifies the degree of rigidity of the rigid end zone element that is inserted at the end of a beam element. The value is constrained between 0 (fully flexible) and 1 (fully rigid).

❑ **Include mass** allows the inclusion or exclusion of mass from the rigid end zone elements. If mass is excluded then the material density is tabulated as zero for the rigid end zone elements.

Assigning rigid zones to lines in a model

Rigid zones are assigned to lines in a model as part of assigning the line mesh attribute that contains the rigid zone definition. Single, multiple, or all lines in a model (if appropriate) can be selected prior to assigning an attribute. Lines having ends that do not intersect with other lines such as lines representing columns that 'sit' directly on supports, or ends of lines representing cantilevered beams, are not assigned rigid zone lengths. Because the calculation of rigid zone lengths is dependent upon both mesh and geometric section properties being assigned to each line feature there are potential issues in relation to the order in which they are assigned to a model.

- If geometric section properties are assigned to a line before any line mesh assignment is made for that line a standard "Unable to visualise without a mesh" message will appear and no fleshing will be seen until the mesh is assigned.

- If a line mesh is assigned to a line before any geometric section property assignment is made for that line an initial attempt at meshing a line will not calculate an automatic rigid zone length because the geometric section properties for that line (and from which derived offset distances and eccentricities are extracted) cannot be immediately found. In this situation no rigid elements will be created, and the line will be initially meshed with only non-rigid elements. A subsequent assignment of a geometric section will invalidate the mesh on all the assigned lines and connected lines, and cause Modeller to recalculate rigid zone lengths for all those assignments.

- In a multiple analysis model, whilst it is possible to assign different geometric properties to each line in each analysis, note that all analyses must share the same mesh since only the geometric section assignments made for the base analysis are used.

Visualisation of rigid zones

Rigid zone lengths are visualised on the Mesh layer as a thicker line and in a darker colour of the specified mesh colour. Defining rigid zones changes the fleshed length of beam sections. The following image shows what is displayed in each case.
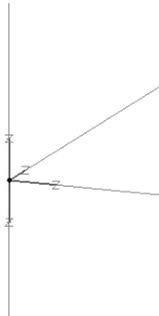
Mesh layer visualisation of connection with rigid zones
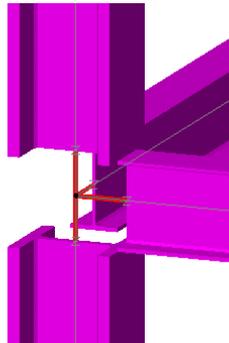
Fleshed geometry visualisation of connection with rigid zones (Rigid zone lengths are drawn in same colour as the point colour)
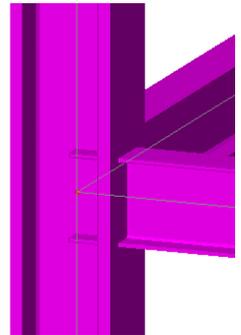
Fleshed geometry visualisation of connection with no rigid zones. (Default fleshing is shown)

Results processing

By default, the elements representing rigid ends of beams do not present element-based results when results are viewed. For instance, a bending moment diagram would only be plotted for the non-rigid elements and not for the regions where rigid zones were defined. If it is required to view results in rigid zones this can be achieved by accessing the Model properties diialog, selecting the Solution tab, pressing the Element options button and checking (selecting) the Show results for rigid zone elements option.

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.

Automatic insertion of joints to ends of line beams

Joint elements can be automatically introduced at the end of lines by choosing to include a joint as part of specifying an end condition for a line mesh attribute. See **Meshing a Model - Beam End Conditions** for more details.

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
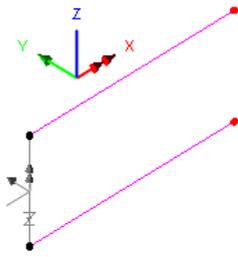
Joint between two points

(Lower point is the master)
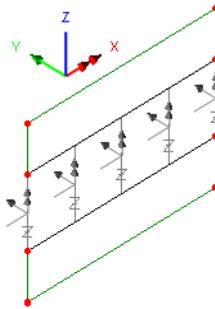
Joint / Interface mesh
between two lines

(Lower line is the master,
four line mesh divisions are
assigned to line)
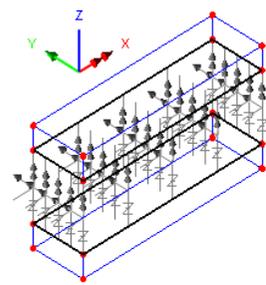
Joint / Interface mesh between
two surfaces

(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.
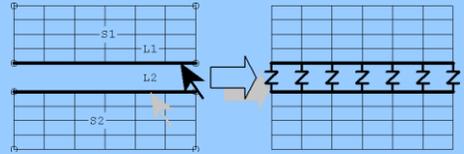
Note. The unmerge facility allows coincident features to be created from a single feature and also allows a feature to be set as Unmergable, so it will not be accidentally merged back with another coincident feature. See **Merging and Unmerging** for more details.

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.
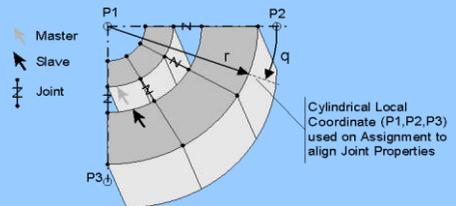
Joint Material and Geometric Properties

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
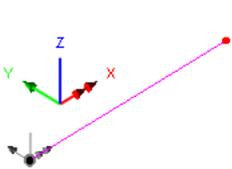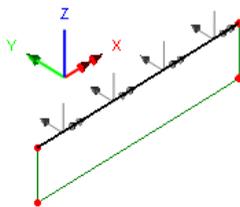
Lumped mass at a point   Distributed mass along a line   Distributed mass over a surface
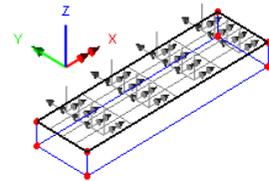
Non-Structural Mass Local Axis Direction

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.

Mode 1 - Open   **Mode 2 - Shear**   **Mode 3 - Tear**
(orthogonal shear to mode 2)
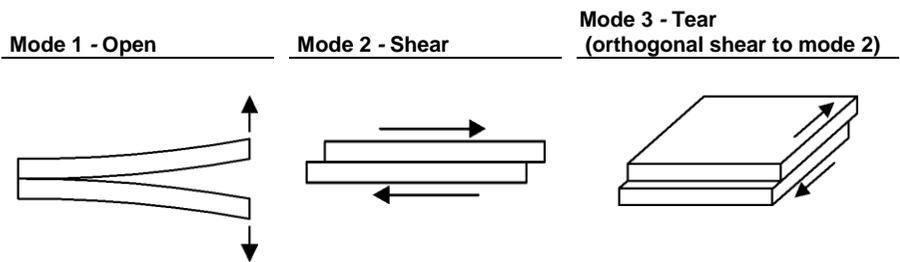
The interface elements are used to model delamination in an incremental nonlinear analysis. These elements have no geometric properties and are assumed to have no thickness.

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.
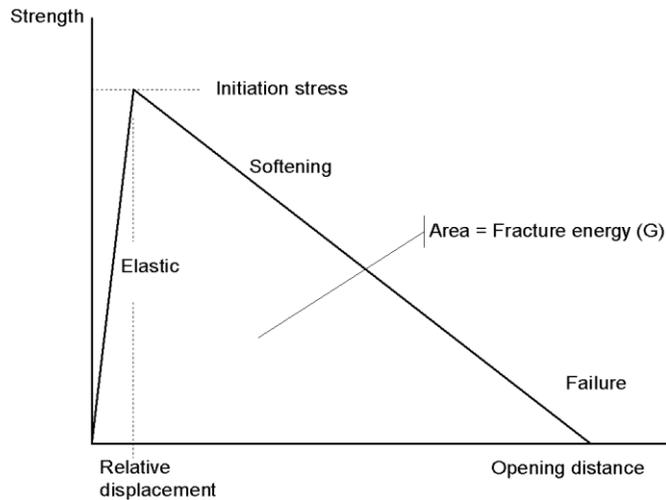
Material Parameters

❑ **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

LUSAS 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)**.

Selecting elements

Selection of a point, line, surface, or volume mesh element is made on the appropriate Mesh dialog accessed from the **Attributes> Mesh** menu item.

The setting of an **analysis category** (2D Grillage/Plate, 2D Inplane, 2D Axisymmetric, 3D) for a model filters all available elements and lists only those elements on each Mesh dialog that will be suitable for use for that type of modelling.

Selection of an element type and an interpolation order (linear or quadratic), will determine the actual element name chosen for use. For advanced users, element names may be manually entered, but this should be used with caution since elements chosen may not be supported in certain levels of the software, or may be inappropriate for a particular type of analysis.

See *Element Types and Availability* in the *Element Reference Manual* for full details of all elements.

See the *LUSAS Theory Manual* for full details of the element formulations.

Point Elements

Non structural mass and Joint elements are defined at or between points. Elements in plain text are available in Standard product versions of LUSAS software. Elements in **bold** text are in Plus versions.

See *Element Types and Availability* in the ***Element Reference Manual*** for full details of all elements.

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 Elements

The following table lists selected 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.

See *Element Types and Availability* in the ***Element Reference Manual*** for full details of all elements.

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, BMX3	-	**BS3, BS4**
Thick beam	BMI2 (LT)	**BMI3**	BMI21 (LT)	BMI21 (LT), **BMI31**
Thick cross-section beam	-	-	-	**BMX31**
Thick warping beam	-	-	-	BMI21W, BMI22W, BMI31W, BMI33W, BMX21W, BMX22W, BMX31W, BMX33W
Plain strain beam	-	BMI2N, BMI3N	-	-
Engineering grillage	GRIL (LT)	-	-	-
Cross-section beam	-	BMX3	-	**BSX4**
Semiloof beam	-	-	-	**BSL3, BSL4, BXL4**
Axisymmetric membrane	BXM2	BXM3	JNT4	-
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 a 2D frame, if there is any possibility of wanting to develop that frame into a fully 3D model it is recommended to use a 3D **analysis category** and use 3D elements.

- Setting an **analysis category** will filter the available elements such that only those applicable are available. If other elements are used note that no check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, but LUSAS Solver will stop the analysis if the element is unsuitable.

Surface Elements

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.

See *Element Types and Availability* in the *Element Reference Manual* for full details of all elements.

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

- Setting an **analysis category** will filter the available elements such that only those applicable are available. If other elements are used note that no check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, but LUSAS Solver will stop the analysis if the element is unsuitable.

Volume Elements

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.

See *Element Types and Availability* in the *Element Reference Manual* for full details of all elements.

Generic Element Types	Tetrahedral		Pentahedral		
	4 noded	10 noded	6 noded	12 noded	15 noded
Stress	-	-	PN6	-	**PN15**
Thermal	TH4	**TH10**	PF6	-	**PF15**
Explicit dynamic	TF4	**TF10**	**PN6E**	-	-
Stress composite	**TH4E**	-	**PN6L**	**PN12L**	**PN15L**
Thermal composite	-	-	**PF6C**	**PF12C**	**PN15C**

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

- Setting an **analysis category** will filter the available elements such that only those applicable are available. If other elements are used note that no check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, but LUSAS Solver will stop the analysis if the element is unsuitable.

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 for single constant cross-sections by:

- Using the **Attributes > Geometric > Section Library** menu item to access supplied and user-created sections in the **section library.** These stored sections can be rotated about the centroid of the section or mirrored to achieve a desired orientation when assigned to a feature in a model. For **design code checking of steel beams** a rotation about centroid value of 0 degrees must be used, and if, for this use, sections need to be rotated 90 degrees from their otherwise orientated position, the setting of a beta angle of 90 degrees either during the line mesh assignment, or after line mesh assignment by changing the value on the line mesh assignment dialog. After assignment this can be accessed from the **properties panel**. Applying a local coordinate system is an alternative method to achieve the same result.

- Using the **Attributes > Geometric > Line** menu item to enter section properties directly on the Geometric Line dialog for a particular chosen **Usage** with an element type, or **ASPC Thickness** (used to define a thickness for **arbitrary section property calculation** purposes).If the Enter Properties option is chosen and you wish to see your section fleshed you will have to define your own beam cross-section information if the section was not selected from a standard section library or created by using the arbitrary section property calculator. Note that the arbitrary section property calculator should generally be used to define sections in preference to manually defining geometric line property data since cross-sectional values (that are used by the fleshing facility) are stored as part of the section property calculation, meaning that fleshing of these sections will take place automatically.

Reinforcement details (for RC frame design checks)

- Provided an **analysis category** of 3D is in use, reinforcement details can be defined for line beams using the **New** option from this droplist. See **line reinforcement** for details. Any previously defined line reinforcement utilities are also listed for selection.

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. The See **Tapering Sections** below 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.

Compound sections can be defined by:

- Using the **Attributes > Geometric > Compound Section** menu item to create a compound section that is made up from supplied and user-defined library sections that are positioned relative to each other, and have differing material properties. See **Compound Sections** for details.

Infilled or Encased sections can be defined by:

- Using the **Tools > Geometric > Infill/Encased Section** menu item to define the geometry for a range of infilled or encased section types either with reference to sections held in the main section library, or from user-defined values. See **Infill/Encased Section Property Calculator** 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.

Section Property Modification

Properties affecting the analysis stiffness of geometric line section properties can be reduced or otherwise factored, whilst leaving their defined dimensions and section properties intact for design calculations, by creating and assigning a section property modifier to selected features and for a particular analysis. Section property modifier attributes are defined for each feature type using the **Attributes> Geometric> Section Property Modifier** menu item and then **assigned** to the required geometric feature.

Notes

- Assignment of a section property modifier attribute to a beam only affects the values tabulated to the Solver data file. The properties shown in the geometric attribute dialog, and accessed through the LPI and User Defined Results facilities remain as initially defined.

- Where a section property modifier attribute is assigned to a compound or infill section the effects of the two factors (one user, one automatic) will be multiplied into a single set of factors.

Options

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.

Standard properties	Additional properties for design checking	Plastic properties
A - Cross-sectional area	kx - Radius of gyration about x axis	Ap - Plastic cross-sectional area (= elastic area)
Ixx - Second moment of area about x axis	ky - Radius of gyration about x axis	
	kz - Radius of gyration about x axis	Zpy - Plastic section modulus about y axis
Iyy- Second moment of area about y axis	Cw - Warping torsional constant	
Izz - Second moment of area about z axis	xo - Shear centre, distance from centroid along x axis	Zpz - Plastic section modulus about z axis
Ixy, Iyz, Izx - Product moments of area	yo - Shear centre, distance from centroid along y axis	Zpt - Plastic torsional section modulus
J - Torsional constant	zo - Shear centre, distance from centroid along z axis	Yp - Plastic neutral axis, distance from centroid along y/z axis
Asx - Effective shear in x direction		
Asy - Effective shear in y direction		Zp - Plastic neutral axis, distance from centroid along y/z axis
Asz - Effective shear in z direction		

Plastic geometric properties are required for beams when using the stress resultant material model (model 29).

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 by pressing the Tab key in last row and cell of the grid.. 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.
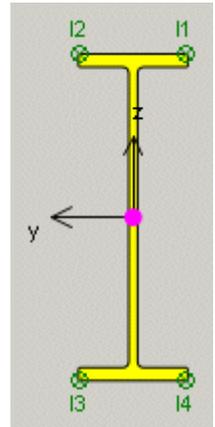
- **Cross-section** information is used to define a series of quadrilateral shapes that represent the cross-sectional shape of the section. These only need to be defined if a beam's properties have been defined manually and it is required that the beam's shape is visualised when using the fleshing option, 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 internal points) can also be set.
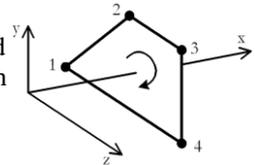
Once defined, the geometric section properties are added to the Treeview using the **OK** or **Apply** button. The section is then available for assigning to the appropriate Lines on the model. Assigned beam section properties may be fleshed using the fleshing button or from the Attributes Treeview.

Eccentricities

- Beam elements accommodate **Eccentricities** which are measured **from** the bending plane **to** the nodal line in the local element direction. Beam section properties are always input relative to the beam axis.
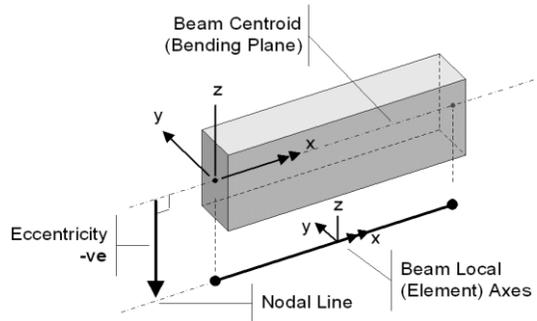
- When sections are defined at either ends of a tapering beam the eccentricity of one section to the other to achieve the vertical and horizontal alignment setting specified is automatically calculated. Subsequent entering of an eccentricity value for the 'master' end in the Value field will automatically 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.

Thicknesses

- ❑ Structures modelled using plate, membrane or shell elements require a **Thickness** to be defined for each surface.
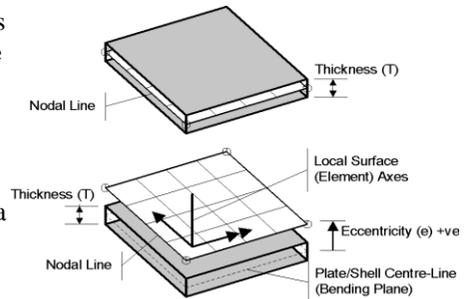
Eccentricity

- ❑ 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 the local z direction) can be defined and the internal position(s) of the transverse shear spring(s) for joint elements

with non-coincident nodes can be specified using parametric distance factors. See the *Element Reference Manual* for details

Geometric Properties of Thermal Link

Geometric properties for a thermal link are defined using the **Attributes > Geometric > Thermal Link** menu item. The following are supported:

- ❑ **Points** - for which an **Area** must be specified in model units.
- ❑ **Lines** - for which a **Thickness** must be specified in model units. When solved, the thickness entered will be multiplied by the length of each portion of the line represented by each link, and tabulated as the area of each link. An average value is taken if the lengths / thicknesses vary between master and slave lines.
- ❑ **Surface** - for which nothing is specified. When solved, the area of each surface is calculated and tabulated. An average value is taken if the areas vary between master and slave surfaces.

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 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 visualisation of beams of standard or arbitrary cross-section (that are held in section libraries), and surface thicknesses, that are assigned to features of a model can be toggled 'on' and 'off' using the fleshing 🔧 button.

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** (a distance equal to half of the height of width of the section, whichever is the larger) or **Specified distance** (in model units) 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. Note that only elements that are activated in a particular loadcase are fleshed.

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.
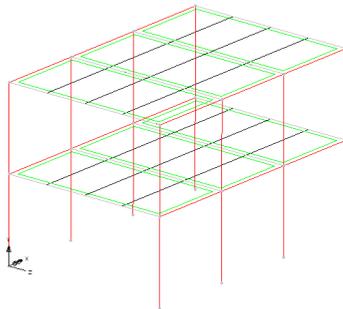
Visualisation of Attributes Without Fleshing
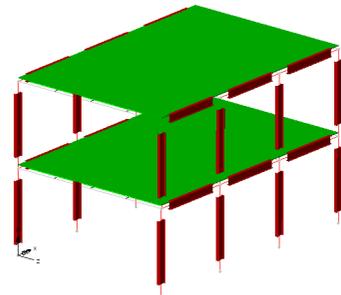
Visualisation of Attributes With Fleshing (and end shrinkage applied)

Reinforcement Attributes

Reinforcement bar attributes are used to model the steel reinforcement in reinforced concrete. They can be defined by using the by **Attributes> Geometric> Bar reinforcement** menu item. When used with the Smoothed Multi Crack Concrete Model 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 (the 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 (the Equivalent bar method) as would be used typically in a 2D plane strain analysis.

The input fields on the dialog have different meaning according to the model type and reinforcement type specified.

❑ **Bar area** The cross sectional area of an individual reinforcing bar if discrete bars are being defined in a 3D model, or the equivalent area of a number of reinforcing bars if a bundle or simplified arrangement of bars is being considered in a 2D model.

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** - For both 'Discrete bars' and 'Equivalent bars' reinforcement types (see below) this relates to the diameter of a single bar if individual reinforcing bars are being considered (as in the examples shown below), or the equivalent diameter of a bundle of bars if bundled bars, or a simplified arrangement of bars is being considered. The bar diameter is described in clause 7.3.4 expression 7.11 of Eurocode EN1992-1-1 and has the same meaning in the **crack width calculation utility**.

❑ **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.
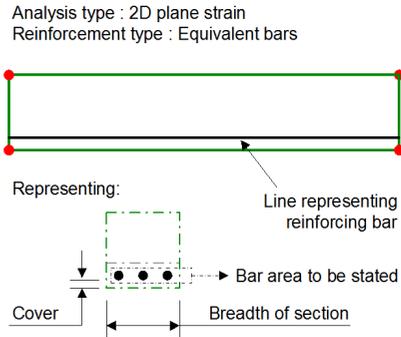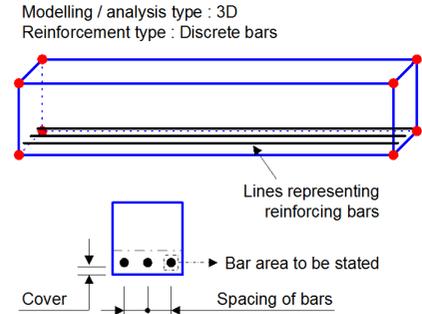
Bar area to be stated for equivalent bars   **Bar area to be stated for equivalent bars**

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.

- Reinforcement bar attributes are not used by the RC Slab Designer.

Wood-Armer / Clark-Nielsen Attributes

Wood-Armer and Clark-Nielsen attributes can be defined using the **Attributes> Wood-Armer...** menu item. These attributes contain details and properties of slab reinforcement arrangements. When assigned to relevant features (see below), they permit calculation of top and bottom reinforcement moments for plates and grillage elements and top and bottom reinforcement forces for shell elements according to Wood-Armer / Clark-Nielsen theory. Multiple slab design attributes of these types may be defined and assigned to relevant features on a model.

❑ **Analysis category** Shows the **analysis category** in use. Wood-Armer / Clark-Nielsen attributes can only be defined for 2D Grillage / Plate and 3D analysis categories.

❑ **Reinforcement angle** sets the angle in degrees between the x direction reinforcement and the y' angle reinforcement. Reinforcement angles are measured anticlockwise from the x-axis towards the y-axis. To carry out Wood-Armer calculations on grillage elements an additional equivalent plate width parameter must be specified in the geometric properties to convert bending and twisting moments into equivalent plate moments. Wood-Armer moments for shell elements include both membrane and flexural effects using Clark's method.

❑ **Reinforcement Distances** from the centre of the reinforcement bars to the **top** and **bottom** faces of the concrete section. These values are only required for slabs modelled with shell elements.

Design components

When carrying out a design based upon Wood-Armer (or Clark-Nielsen) calculations, the angle θ of the failure cracks can be entered using the k-factor parameter (k=tan θ. In principle, any value for k can be chosen and an infinite number of possible combinations capable of resisting a particular set of applied forces exists. For design purposes, a value of θ (or k) may be established to minimise the resistive forces applied, hence minimising the steel area requirement. The minimum k factor for orthogonal reinforcement is +/- 1, or for θ is +/- 45 degrees. For skew reinforcement, the minimum k factor depends upon the direction of the reinforcement.

❑ **Minimised total weight / area of reinforcement** If the k factor is not specified,a default of 1.0 is used and the area of reinforcement is minimised.

❑ **k factor for non-minimised area of reinforcement** allows a k factor (tan θ) to be specified so that Wood-Armer / Clark-Nielsen calculations can be carried out without minimisation of the reinforcement areas in order that the load capacity of an existing reinforced concrete slab can be assessed.

Assessment utilisations

These are based upon known moment resistances.

❑ **Display assessment utilisations** allows input of resistive moments or forces (instead of the k factor) for top and bottom surface reinforcements for assessment purposes.

- For Wood-Armer all values of **top and bottom rebar moment resistance** must be greater than zero.

- For Clark-Nielsen all values of **top and bottom rebar force resistance** must be greater than zero.

- If assessment values are specified, utilisation factors are calculated for each position on the model. Factors above 1 indicate failure.

On completion a Wood-Armer attribute with additional slab design data is created in the Attributes 🔧 Treeview.

Assignment

- For models created with plate and shell elements the Wood-Armer / Clark-Nielsen attribute can be assigned to surfaces.

- For models created using grillage elements the Wood-Armer / Clark-Nielsen attribute must be assigned to the points in the grillage model where grillage moments will be determined (that is, at the intersection of longitudinal and transverse members).

Notes

- See **Wood-Armer Reinforcement Calculations** for more information.

- Top and bottom reinforcement moments are calculated for plate and grillage elements, and top and bottom reinforcement forces are calculated for shell elements. Consult the *Element Reference Manual* for Wood-Armer results column details.

- Contours of Wood-Armer and Clark-Nielsen components, and equivalent plate moments can be plotted. See **Visualising the Results** for details.

- Plate equivalent moments are always available for a grillage model whenever the equivalent plate width is nonzero, as they do not require any inputs from the Wood-Armer attribute. They are listed as a Force/Moment entity, and are named "MX(Epw)", "MY(Epw)" and "MXY(Epw)".

- See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Wood-Armer bridge slab assessment' for a worked example covering Wood-Armer calculation.

- For details of Wood-Armer reinforcement calculations refer to the *Theory Manual.*

Steel Frame Design Attributes

Steel Frame Design Attributes are used to specify member design values and in particular member lengths and end conditions used in determination of buckling capacities. They are assigned to line features in a model to specify additional information about those features (information that is not required for a structural analysis to be carried out) in order to enable **steel frame design** checks to be carried out to a supported design code.

Steel frame design attributes can be defined using the **Attributes> Design> Steel Frame Design...** menu item that is available for use with selected Bridge and Civil software products for licences that support this facility, and only if a model **analysis category** of 3D is in use. The steel product standards that are available for selection on the resulting dialog depend upon the steel design code selected from the dialog displayed when using the **Design> Steel Frame Design** menu item.

The following design codes are currently supported:

- ❑ **AASHTO LRFD 7th Edition** AASHTO LRFD Bridge Design Specifications, 7th Edition, American Association of State Highway and Transportation Officials, 2014.
- ❑ **AASHTO LRFD 8th Edition** AASHTO LRFD Bridge Design Specifications, 8th Edition, American Association of State Highway and Transportation Officials, 2017.
- ❑ **CSA S6-14 Canadian Highway Bridge Design Code** Canadian Highway Bridge Design
- ❑ **EN1993-1-1: 2005+A1:2014** - EN 1993-1-1:2005 Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings.
- ❑ **EN1993-2: 2006** - EN1993-2: 2006 Eurocode 3: Design of steel structures - Part 2: Steel Bridges. The Eurocode National Annexes currently supported are:
- ❑ **AS 4100-1998 Steel Structures (Australia)** - Australian Standard: AS4100-1998, Steel structures, Standards Australia International Ltd, Sydney.

Defining member buckling lengths

It is possible for a member to have unsymmetric restraints leading to different buckling lengths about each axis. In LUSAS a member's unrestrained length may be taken from either a line or a **combined line**. This allows a member to be defined by more than one line and multiple buckling lengths. The choice of whether to use the line length or combined line length for each buckling phenomenon is specified in the Steel Frame Design Attribute dialog. Care is required in defining and assigning steel frame design attributes in order to ensure that the correct buckling lengths are assigned for all valid situations in the model. It is recommended that the concept is understood first in a test model as the unbraced length can easily be underestimated.

Examples of member buckling lengths

Example 1, below, shows the invalid use of a combined line to define a member. Restraint is provided in the major and minor directions at each end of the member and in the minor direction only at the point where member (Line 4) is connected. In the steel frame design attribute the buckling length is set in the attribute to use the combined line length about the major axis (between end restraints) and to use the individual line lengths about the minor axis. The unrestrained length in the minor direction will be taken as line length. This would be correct for the first line (Line 1) up to the restraint but the unrestrained length of the second and third lines (Line 2 and Line 3) will be taken erroneously as their individual lengths, as opposed to a length for the sum of these two lines. Members should therefore not be made up of a series of lines where there are no intermediate points of restraint at the end points of each line. Additionally a member should only ever be defined with a single combined line, because overlapping combined lines will cause errors in the determination of buckling lengths.

Examples 2, 3 and 4 show valid uses of combined lines and lines to calculate buckling lengths.
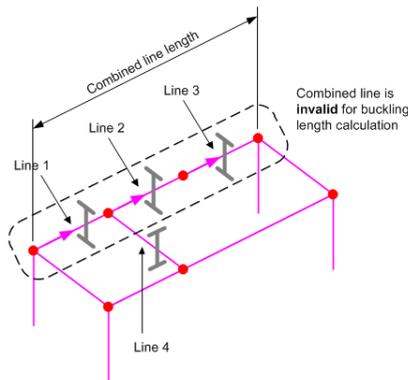
For the member highlighted	Buckling length is obtained from...
Major axis	Invalid calculation
Minor axis	Invalid calculation

For the member highlighted	Buckling length is obtained from...
Major axis	Combined line
Minor axis	Line

Example 1: Invalid use of a combined line to define a member (when the ends of some internal lines defining the member have no restraint)
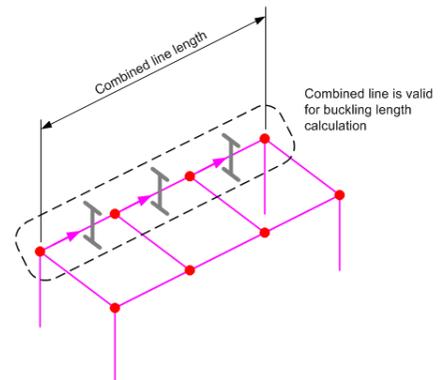
Example 2: Valid use of a combined line to define a member using multiple lines(when the end of each line defining the member is restrained)
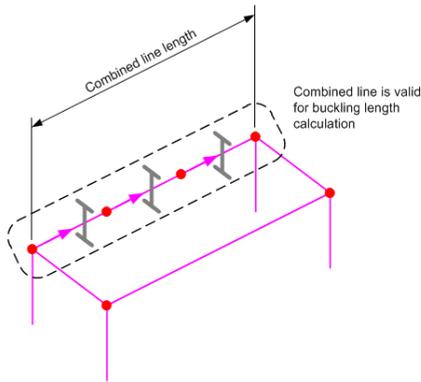
For the member highlighted	Buckling length is obtained from...
Major axis	Combined line
Minor axis	Combined line
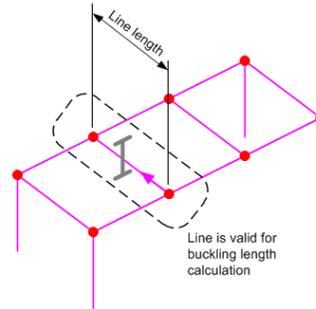
For the member highlighted	Buckling length is obtained from...
Major axis	Line
Minor axis	Line

Example 3: Valid use of a combined line to define a member when no internal restraints are provided for all lines defining that member. **Example 4: Valid use of a line for a single member**

Notes

- Member lengths for buckling are always taken from the actual line or combined line length and will ignore the presence of any **rigid end zones**. This is a conservative assumption.

- The steel frame design attribute requires the lines that are present in a combined line to all be in the same direction.

- When geometry data is imported from third-party systems, it may be the case that individual lines are created inside Modeller instead of a single line between two points. In such situations it may be possible to create combined lines to define members depending upon restraints used, otherwise, if not appropriate, the line geometry should be simplified with a single line representing a single member.

- See specific design codes for details of buckling lengths.

RC Slab Design Attribute

Reinforced concrete slab design attributes can be defined using the **Attributes> Design> RC Slab Design...** menu item that is available for use with selected Bridge and Civil software products only.

Slab design attributes specify the reinforcement arrangement that is to be used for RC Slab Design calculations. Top and bottom reinforcement layout, size, spacing, cover and skew angle of bars can be defined in accordance with particular country requirements. Multiple slab design attributes may be defined and assigned to relevant features on a model either before or after a model has been solved, but must be assigned before the RC Slab Designer can be used.

- ❑ **Reinforcement library** This contains selections for Metric bar sizes, Australia bar sizes, and Canada and American bar size notations.

- ❑ **Bar sizes** The sizes and designations that are shown are appropriate to the region for the selected Code of Practice. User-defined bar sizes can be added by typing over a value in the drop-down list. This option may be used to represent equivalent or bundled bars. A positive value greater than zero must be entered.

- ❑ **Bar spacing** is measured centre-to-centre, perpendicular to the bar direction.

- ❑ **Cover** is measured from outermost part of the bars laid in the outermost layer of reinforcement to the nearest concrete face. It is intended that the value entered here is the actual cover, which may include allowance for fixing tolerance.

- ❑ **Use nominal cover in crack width calculations** A nominal cover can be entered where different from the actual cover. The nominal cover will be used to determine crack widths in the RC slab. Further details are provided in the **RC Slab Design : Design Code Settings** pages for each design code.

- ❑ **Skew angle of bars in y' from x** Used to arrange bars in a non-orthogonal arrangement where the skew angle can lie between 0 and 180 degrees (90 degrees is perpendicular). The angle entered defines the orientation of the y' reinforcement with respect to the x-direction, measured in an anti-clockwise direction. The angle specified is used to determine Wood-Armer / Clark-Nielsen results in the x and y' (skew) directions (for comparison to flexural resistance in the two corresponding directions) and in the determination of crack widths when chosen.

On completion a Wood-Armer attribute with additional slab design data is created in the Attributes 🕸 Treeview.The attribute is distinguished from normal Wood-Armer attributes by being held in a RC Slab folder and with a name that contains all bar diameter, cover and skew angle information as defined.

Assignment

- For models created with plate and shell elements the resulting Wood-Armer attribute can be assigned to surfaces.

- Unlike the general Wood-Armer attribute, this attribute cannot be assigned to models created with grillage elements.

- Assignment of the RC slab attribute therefore has the same effect as assignment of a Wood-Armer attribute.

See **RC Slab Design: Overview** for a general overview of the slab design process.

RC Frame Design Attributes

Reinforced Concrete frame design attributes are used to specify additional Code of Practice specific information about model line features that represent concrete members in order to enable **RC frame design** checks to be carried out. Note that this additional information is not required for an initial structural analysis to be carried out.

RC frame design attributes can be defined using the **Attributes> Design> RC Frame Design...** menu item that is available for use with selected Bridge and Civil software products for licences that support this facility, and only if a model **analysis category** of 3D is in use.

The options that are available for selection on the resulting dialog depend upon the design code selected from the dialog displayed when using the **Design> RC Frame Design** menu item.

The following design codes are currently supported:

❑ **EN1992-1-1: 2004** - EN 1992-1-1:2004 Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings.

❑ **EN1992-2: 2005** - EN1992-2: 2005 Eurocode 2: Design of concrete structures Part 2: Concrete bridges - Design and detailing rules.

Typical information defined in a RC frame design attribute includes:

- Beam and column minimum reinforcement ratios

- Calculation methodology

- Parameters for stress limitation and crack control

- Creep parameters

Once defined, a Design attribute entry is created in the Attributes ❤ Treeview ready for assignment to selected lines in a model.

Note that the same RC frame design attribute is used for both EN 1992-1-1:2004 and EN1992-2:2005 design codes.

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:

❑ **Australian steel sections**

- ❑ **Canadian steel sections**
- ❑ **China steel sections**
- ❑ **EU steel sections**
- ❑ **Indian steel sections**
- ❑ **KS steel sections** including Korean rail sections
- ❑ **New Zealand steel 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' Girders', Northeast Bulb 'T'
- ❑ **Australia and New Zealand beams**: Super-T beams T1 to T5 (open and closed)
- ❑ **Canada 'I' beams**

In addition, user-created sections 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.

Note. LUSAS Modeller re-computes all section properties from the section dimensions that are published in supported codes of practice, rather than use the published section property values directly. This is done because published section properties do not contain all the section properties that are required for both analysis and design use, and additionally, for some codes, some published properties are clearly incorrect. Note that, in most cases, LUSAS calculated section properties are within 1% of the corresponding published values.

Usage

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 User-defined Sections to the Section Library

In addition to sections provided in the geometric beam section library, other user-defined sections can be added to the library by using Section Property Calculator facilities. These are accessed from the **Tools > 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, infilled and encased 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 **analysis category**, as shown on the multiple varying section 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 in the Distance columnare 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 **<u>from</u>** the bending plane of the section **<u>to</u>** 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.
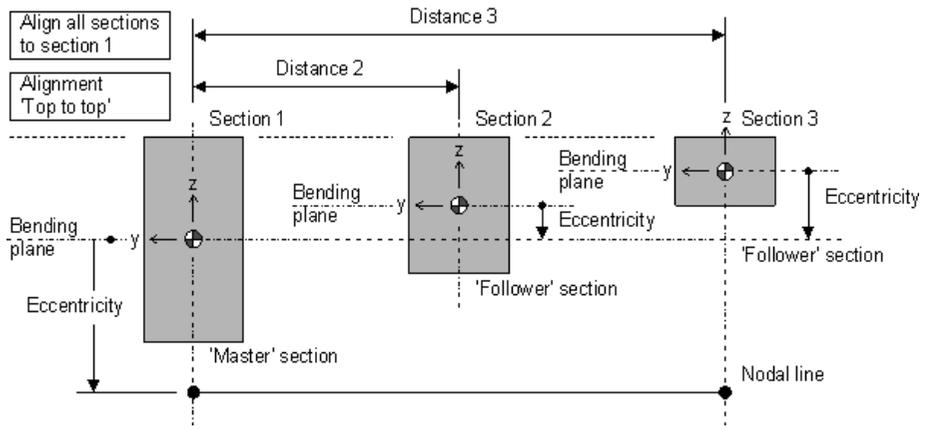
Whilst not commonly used, user-defined individual eccentricity can be entered for selected sections by selecting the **Individual eccentricity** menu item from either the Vertical or Horizontal Alignment drop-down menu, then accessing a section's properties by clicking the launch dialog button ⬇ in this 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
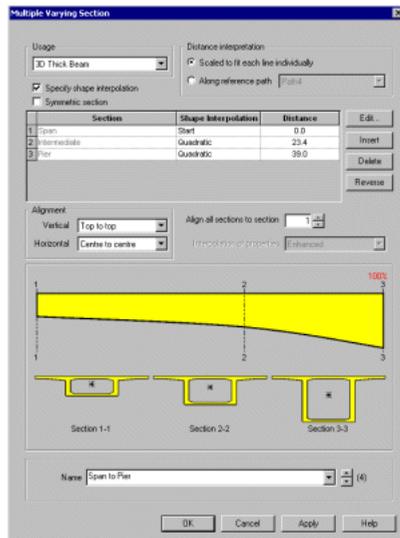
Varying section distances defined for assignment of the line attribute to a single line.
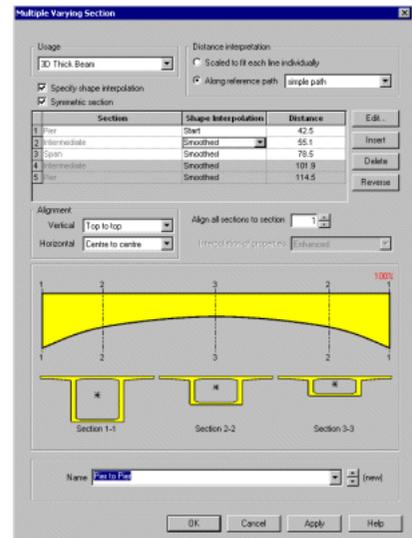
Varying section distances defined for assignment to multiple lines with reference to a path.

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.

When modelling varying sections with constant beam 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 only one element per line may be required.

Single geometric line attribute of 3 defined varying sections assigned to a single selected line
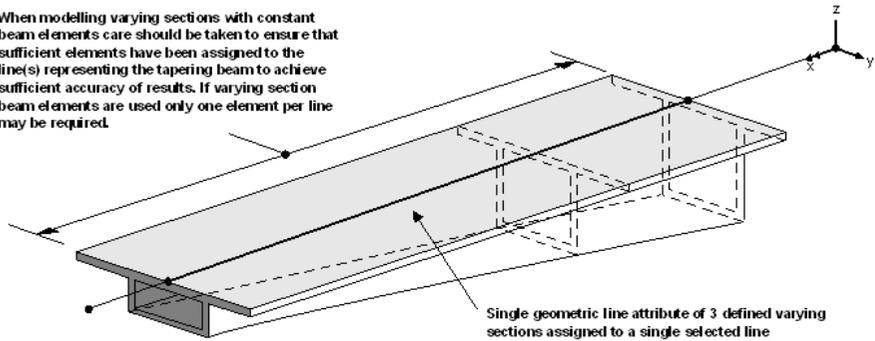
Single line beam assigned a single multiple varying section line attribute. (For clarity beam line has been visualised at top of section.)

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.
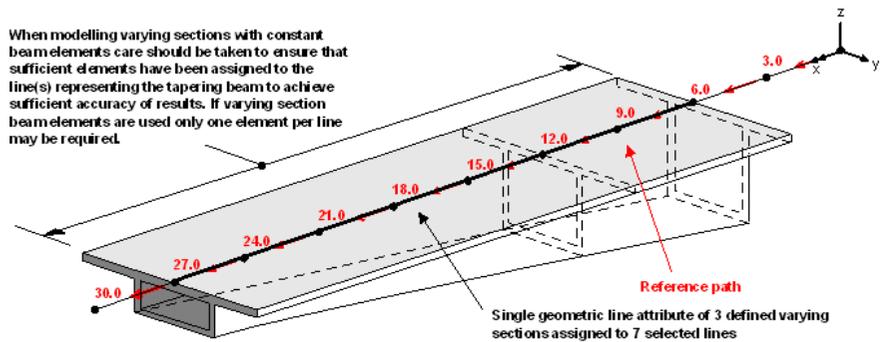
**Multiple line beams assigned a single multiple varying section line attribute with reference to a path.
(For clarity beam line has been visualised at top of section)**
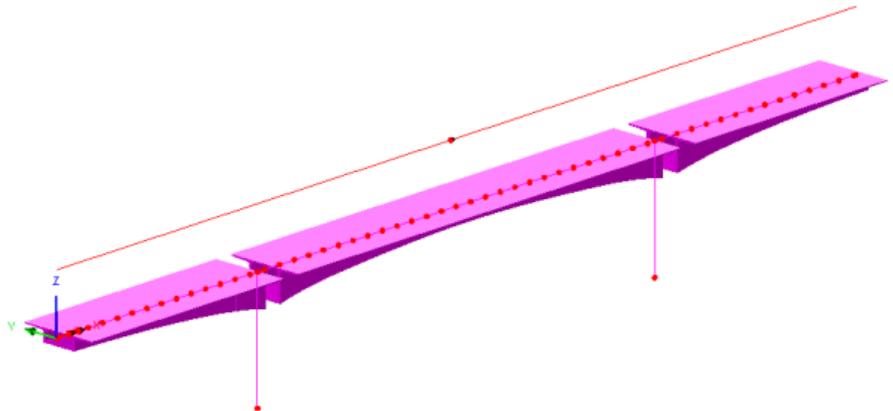
**Three varying section line attributes assigned to multiple sets of lines of a model with the use of a
reference path**

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.

Compound Sections (from library items)

Compound sections can be defined by using the **Attributes > Geometric > Compound Section** menu item. Compound sections are defined from existing library sections that are positioned relative to each other, and can have different material properties assigned to them. The compound section attribute created can be assigned to lines in the model in the same way as any other geometric attributes.

Note that the compound section attribute created will not provide a composite section. Instead, because the sections are expected to be disconnected, equivalent properties are computed on that basis. For composite sections where the sections are expected to be connected, the **arbitrary section property calculator** should be used.

Basic usage and section selection

A compound section is defined for use with a particular **analysis category.**

In the section selection table the sections that will be used to generate a compound section must each be added, and horizontal and vertical alignment settings for chosen points on each section must be specified. Note that user-defined sections need to be saved to the local or server libraries prior to using this facility. Sections can be assigned different material properties.

- Clicking on the launch dialog button ⬇ in the **Section** cell allows a pre-defined section to be chosen from a section library. Note that sections may be rotated or mirrored prior to having their entry being inserted into the compound section grid.

- Clicking on the drop-list button ▼ in the **This sect** or **Align to** or **Other sect** cells of the Horizontal or Vertical alignment columns governs how the left, right, bottom, top, centroid or fibre positions of sections that are added to the grid are set out relative to each other.

Alignment settings

❑ **This Section** This setting specifies a reference point for the section that is to be positioned with respect to another section according to the 'Align to' and 'Other section' settings made in that grid row. For Horizontal Alignment 'Left, 'Centroid' and 'Bottom', followed by a list of all fibres defined for the section are listed. For Vertical Alignment 'Top', 'Centroid', and 'Bottom', followed by a list of all fibres defined for the section are listed. See ' Examples of compound section definitions' below for representative uses of this setting.

❑ **Align to** This setting specifies how the reference point should be positioned, either with respect to another Section (by selecting the Section number from the drop-down list of those available), or be positioned by entering a coordinate by selecting the 'Coords' option.

❑ **Other section** When the 'Align to' field specifies a section, this field allows the selection of a position on the other section, in the same way as the 'This Section' field. If 'Align to' is set to 'Coords', the last field changes from a drop-down list to an edit field to allow a value to be entered that represents the distance that the reference point of the section should be positioned away from the other section.

Section visualisation

As the compound section is built-up, a visualisation of the longitudinal and vertical alignment and of the cross-section shapes used is displayed on the dialog. A magenta filled circle represents the location of the nodal line about which the section will be positioned when it is assigned to a line on the model. The selected section in the grid is shown highlighted in bright yellow, with its fibre locations marked.

To see the effect of a change of setting the cursor should be clicked again in another cell. Consistently with the main section library, the visualisation can be inspected by zooming and panning in the display panel. To start zooming and panning, click the section image. The cursor will change to indicate that zoom and pan mode is active. 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. Click any other control on the dialog to leave pan and zoom mode.

Examples of compound section definitions
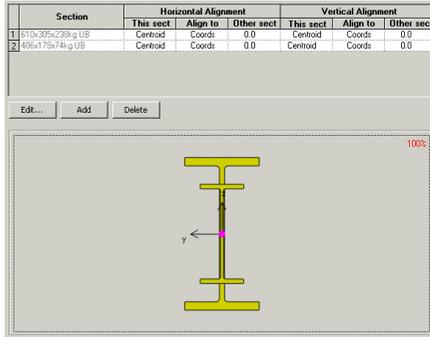
Two sections are initially added and sit at a common centroid awaiting alignment settings to be made.
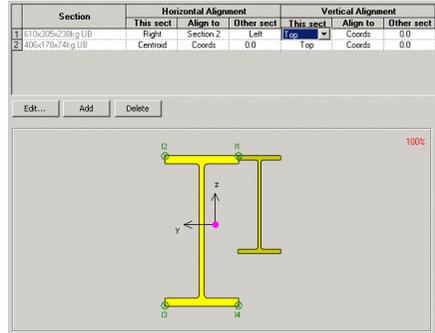
The right-hand side of Section 1 is set to be aligned with the left-hand side of Section 2. The top of both sections is aligned. The new centroid position for the combined section is automatically calculated.
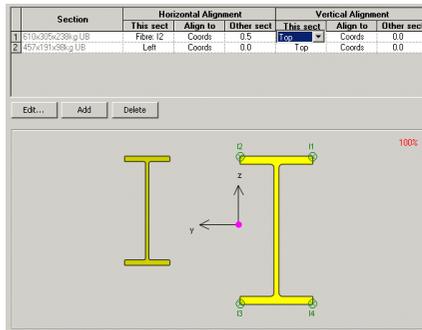
Fibre location 2 of Section 1 is positioned 0.5 model units horizontally from the centreline of Section 2. The top of both sections is aligned. The new centroid position for the combined section is automatically calculated.
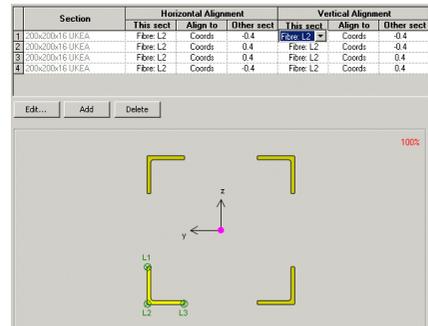
Four angle sections are defined (with three being rotated about their centroid during selection). The relative positions of fibre location 2 are specified for each section.

Section materials

- ❑ **Specify material for each row** When un-ticked, the Reference material is assumed to be the same material. This is the default option.

- ❑ **Reference material** When 'Specify material for each row' has been selected a reference material can be selected from a drop list of current material attributes in the model, or a new material defined. Each section in the grid can then have a material attribute assigned to it using the drop-down list in the final column. From the reference material and the material defined for a section in the grid, the modular ratio is derived and used to calculate the combined (transformed) section properties.

Table related buttons

❑ **Edit**, **Add** 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. Pressing the Add button will, by default, repeat the previous entry as the entry.

❑ **Section details** shows the calculated section properties for the compound section.

Attribute name

The full name of the geometric line attribute added to the ⚛ Treeview will include the Attribute name followed by an automatically created description based upon the number of section library items used. The description appended to the attribute name cannot be edited.

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.

Notes

- This facility is not appropriate for sections which are welded to form a closed section. Where sections are in contact no connection will be modelled between them in the calculation of the torsion constant and shear stress. The calculated torsion constant is the sum of the torsion constants for the component sections.

- For sections consisting of components that are to act compositely, the Arbitrary Section Property Calculator should be used to obtain a more accurate calculation of the torsion constant and shear areas.

- Sections can be defined which overlap but no attempt is made to subtract the missing material from one section where it overlaps another. If this condition is detected a warning is given.

- Where materials are defined in the section for the calculation of modular ratios, the compound section should only be assigned to lines using the same material as the section's reference material.

- Where different materials are specified, the plastic neutral axis and plastic section modulus are not calculated because the ratio of yield strengths is not available.

- Where different materials are specified note that the self weight due to gravity is calculated based on the original areas and the associated densities of the individual components that comprise the compound section. However the

value for the Area as displayed on the Compound Section dialog is for the
transformed section that was created with and cannot be used directly for
simple hand-checking of reactions when carrying out basic checks for a model.

- Compound sections of arbitrary shapes can be defined using the **Arbitrary Section Property Calculator.**

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.

Material Library

❑ The **material library** contains a range of elastic material properties for
commonly used structural materials. For some materials in some regions a user-
defined grade of material can also be specified.

Composite Library

❑ The **composite library** contains a range of commonly used structural
composite material properties.

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.

❑ **Concrete (Nonlinear)** Define material to predict the mean behaviour of a
concrete section due to creep effects, with optional reference to design code.

❑ **Concrete (Reinforced)** Define reinforced concrete that references linear elastic
concrete and steel materials provided in the **Material Library**.

Tension / Compression Only properties

❑ **Tension / Compression Only (Bar)** uses the piecewise linear bar material model to define a simple no compression (tension-only) or no tension (compression-only) or material behaviour.

Specialised Material Properties

❑ **Interface** for use with **delamination**, **elasto-plastic** and **Mohr-Coulomb** material models.

❑ **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.

❑ **Duncan-Chang** A simple soil model whose parameters are calculated by curve fitting triaxial test data over a range of cell pressures.

❑ **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.

❑ **Piecewise 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.

Editing of Material Properties

Editing of pre-defined material data (such as that provided in the material library) allows for viewing both the original material definition input data, as well as modifying the values used. Editing of user-defined material properties only permits viewing and editing of the values used. Context menu entries for materials present in the Attributes Treeview include:

❑ **Edit Definition...** displays the original definition dialog with all the original input data intact.

❑ **Edit Attribute...** 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.

❑ **Edit** original definition dialog with all the original input data intact.

Defining Temperature Dependent Material Properties

Temperature dependent properties can be defined for each material in the Attributes Treeview by selecting the context menu item **Edit temperature dependence…** This option is not available for licences that only support linear analysis. Once defined, the attribute name will be appended with the letters (TD) to signify that Temperature Dependent properties have been assigned. These additional properties need to be removed from a material attribute prior to editing previously defined general material property data.

Care should be taken when using a table of temperature dependent nonlinear material properties. For example, a temperature rise which is subsequently followed by cooling could result in a step change increase in Young's modulus, yield stress and other plastic properties which violates the first law of thermodynamics. If the temperature monotonically increases throughout the analysis and sensible properties are defined in the table, then reasonable results should be obtained

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.

General notes

- Once assigned to geometry, material directions can be **visualised** using the Attributes layer in the Layers Treeview

- Material property attributes can be formed into a composite lay-up using the **composite** attribute facility.

- Rubber, crushing, and plastic material attributes cannot be combined.

Material Library

The Material Library contains standard elastic properties for common materials, with some defined in accordance with national or international Codes Of Practice, and for a range of grades. It can be accessed from the **Attributes> Material> Material Library** menu item.

Other materials can be defined by making an appropriate selection when using the
Attributes> Material menu item.

Materials supported

The following materials are currently available:

- **Aluminium**
- **Aluminium Alloy**
- **Cast Iron**
- **Concrete**
- **Epoxy**
- **Foam**

- **Glass**
- **Graphite**
- **Invar**
- **Polyurethane**
- **Steel - Prestressing bar**
- **Steel - Prestressing strand**

- **Steel - Prestressing wire**
- **Steel - Reinforcing bar**
- **Steel - Structural**
- **Timber**
- **Titanium**
- **Wrought iron**

Standard settings

By selecting the **Material type**, **Region**, **Standard** and, in some cases, **Grade**, a
material attribute is created using appropriate values for Young's modulus, Poisson's
ratio, Mass density and Coefficient of Thermal expansion.

The values shown in the material library dialog are in model units, as defined on the
Model Properties dialog, and therefore may have been converted from values in the
relevant Code Of Practice in other unit sets. For example, the units for Elastic modulus
of Steel may be given as 29,000ksi in a design code, and for models in kips and feet,
or Newtons and metres, the value would be converted as appropriate. Note that for
concrete the values of density displayed are for reinforced concrete and not plain
concrete, because in LUSAS an allowance is made for reinforcement.

Advanced settings

For some materials, and for some regions, a user-defined grade can also be specified
by using the **Advanced define** drop-list item. One example of this is for a selection of
Concrete, USA, AASHTO LRFD 7th, where a Compressive strength, Weight and
Aggregate correction factor can be specified, along with Prestress parameters. The
parameters listed will vary with each design code.

Prestress parameters

Prestress parameters can be entered via the Advanced define menu item for design
codes that are supported by this facility. This currently includes AASHTO (3rd to 8th),
EN1991-1-1:2014, EN1992-2:2005 and IRC:112-2011.

General notes

- For concrete, in general, Codes of Practice give a secant modulus of elasticity, which is strictly only applicable to transient (short term) loads. Under sustained loading, concrete creeps. While this may be handled analytically using the Concrete Creep materials in LUSAS, approximations may be made, for which many Codes Of Practice would require the derivation of an appropriate "long term" elastic modulus based on the age of concrete, length over which load is sustained, cement class etc. As such this is not generally covered by the Material Library. It is, however, possible to create a material attribute from the library and subsequently edit the value of the Elastic Modulus by right-clicking on the attribute in the 🔴 Treeview and selecting the Edit Attribute option, typically dividing the secant modulus by a factor to account for creep (typically dividing by a value in the range 2 to 4) as determined from the rules in the relevant Code to give a suitable "long-term" value.

- Some Codes Of Practice do not give complete data for elastic material properties, most commonly omitting a value for Poisson's ratio e.g. BD21/01 (UK), RT/CE/C/025 (UK), IRC:21 and IS456 (India). In such cases, a value has been obtained from another national Code from the same country or, where this is not possible, another suitable reference.

- Mass density is in units of mass/volume. In some Codes Of Practice e.g. AASHTO (USA), CAN/CSA-S6-06 (Canada), Eurocodes, and IS456 (India), the Code of Practice gives only a Unit Weight (units of force/volume). In such cases, the value shown in the dialog and used in the model is converted by dividing by the acceleration due to gravity in the appropriate units. As described above, the values are in model units, and may have been converted from the values in the Code Of Practice.

- The material attribute name that is provided will also automatically include a description formed from the selections made, when the material is added to the Attributes 🔴 Treeview.

- To define a user-defined linear elastic material model see "Isotropic/Orthotropic Material" on page 249

Usage

- Pick the material type and options required from the drop -list and click **OK** or **Apply** to add the material properties to the Attributes 🔴 Treeview. Values can be modified and other material properties can be defined once the material has been added to the Attributes 🔴 Treeview. The material properties may then be assigned to appropriate features of the model.

Composite Library

The composite library contains a range of structural composite material properties. It can be accessed from the **Attributes> Material> Composite Library** menu item. Composite material units will default to those specified on the Model Properties dialog but may be changed if desired.

Usage

- 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 selected features of the model.

Isotropic/Orthotropic Material

Isotropic material (properties are the same in all directions) and orthotropic material (properties in the two or three principal directions may differ) attributes can be defined using the **Attributes > Material** menu item. The following material properties can be specified.

For a structural analysis:

- ❑ **Elastic** is used to specify linear elastic material properties including Young's modulus, Poisson's ratio, and mass density. Optional thermal expansion and dynamic constants can be specified. Orthotropic material orientation is specified as global, relative to a local coordinate system or relative to the feature local axis system. Orthotropic models are **Plane stress**, **Plane strain**, **Thick**, **Axisymmetric**, **Solid**. Note that not all elements accept all the orthotropic models. Refer to the *Element Reference Manual* for full details of valid element/material combinations. For geotechnical analyses that do not include a two phase material definition, the density must relate to the total mass of the soil, including the mass of any water content in the voids. The soil may be saturated, partially saturated or dry. For geotechnical analyses that include a two phase material definition, the density must relate to the soil grains only.
- ❑ **Plastic** is used to model ductile yielding of nonlinear elasto-plastic materials such as metals, soils/rocks/sand. See headings below for range of materials supported.
- ❑ **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** is used to model the inelastic behaviour that occurs when the relationship between stress and strain is time dependent.
- ❑ **Damage** is used to model the initiation and growth of cavities and micro-cracks.

- ❑ **Shrinkage** is used to define the shrinkage properties of a material as a piecewise linear curve.

- ❑ **Viscosity** is used to model viscoelastic behaviour. Coupling of the viscoelastic with nonlinear elasto-plastic materials enables hysteresis effects to be modelled.

- ❑ **Two-phase** material properties are required when performing a consolidation analysis (or an undrained analysis), in which two-phase elements are used to define soil in either a fully or partially saturated state.

- ❑ **Ko Initialisation** is used to define initial stresses in soils.

Plastic Material Models (Isotropic)

The following Isotropic models are 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.

Plastic Material Models (Orthotropic)

- ❑ **Stress Potential** (**Hill and Hoffman models**) These material 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.

For a thermal analysis:

- ❑ **Isotropic** Specify properties for general thermal and concrete heat of hydration analysis.
- ❑ **Orthotropic** Specify properties for general thermal and concrete heat of hydration analysis.
- ❑ **Thermal Link (Linear)** Thermal links are used to analyses heat conduction at contacting interfaces.
- ❑ **Thermal Link (Nonlinear)** Thermal links are used to analyses heat conduction at contacting interfaces.
- ❑ **Thermal (User)** For advanced use. Used to enter material parameters for user nonlinear thermal material models.

For a hygro-thermal analysis:

- ❑ **Isotropic** Specify properties for a hygro-thermal analysis. 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.

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 isotropic materials can be defined accessed using the **Attributes> Material>** menu item, providing a thermal analysis entry is present in the Analyses Treeview. Isotropic thermal material is applicable to all thermal elements except thermal links (see the *Element Reference Manual*)

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.

- Use isotropic material for materials in which the conductivity is constant in all directions.
- Use orthotropic material when conductivity varies in different directions. The direction of orthotropy is defined relative to any local coordinate systems.

For transient thermal analysis the material properties require a volumetric heat capacity to be entered. This is the product of the density and the Specific Heat Capacity ($\rho*C$)

If phase change is to be modelled the latent heat (of phase change) 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.

Thermal Material (Isotropic)

Isotropic thermal material is used to represent thermal properties which are identical in each direction. It is applicable to all thermal elements except thermal link elements - for which thermal link material properties must be defined.

 For general thermal analysis:

❑ **Phase change state** is required to be specified if a material changes state, i.e. it boils, liquefies or solidifies in the temperature range under consideration. Options are None, DelGiudice, and Lemmon. If phase changes are indicated then the thermal mass effects of the material will be calculated from the latent heat (of phase change) values entered and not from the specific heat capacity values.

❑ **Thermal conductivity** is required in all cases

❑ **Specific heat capacity** is only required for transient analysis.

❑ **Density** is required in all cases

❑ **Latent heat (of phase change)** is only listed for phase change analysis.

Exothermic behaviour

For the specific case of concrete heat of hydration analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, exothermic behaviour settings and values can be specified:

❑ **None** No exothermal reaction is to take place.

❑ **Concrete heat of hydration** when selected, provides the means to specify the following cement types: **I**, **II**, **IV**, **V** and **General**, with each requiring additional parameters to be specified. See **Concrete Heat of Hydration** for more details.

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.

If stress in the concrete due to the hydration process is required a coupled analysis should be carried out.

Thermal Material (Orthotropic)

Thermal orthotropic material is valid for **2D membrane** plane field or **3D solid** field elements.

For general thermal analysis:

❑ **Phase change state** is required to be specified if a material changes state, i.e. it boils, liquefies or solidifies in the temperature range under consideration. Options are None, DelGiudice, and Lemmon. If phase changes are indicated then the thermal mass effects of the material will be calculated from the latent heat (of phase change) values entered and not from the specific heat capacity values.

❑ **Thermal conductivity in x, y and z directions** can be specified independently.

❑ **Angle of orthotropy** may be specified to rotate the material reference axes with respect to the reference axes.

❑ **Specific heat capacity** is only required for transient analysis.

- ❑ **Latent (heat of phase change)** is only required and listed for phase change analysis.
- ❑ **Density** is required in all cases.

Exothermic behaviour

For the specific case of heat of hydration analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, additional options and values can be set by selecting the **Concrete heat of hydration** option.

- ❑ **Concrete heat of hydration** when selected, provides the means to specify the following cement types: **I**, **II**, **IV**, **V** and **General**, with each requiring additional parameters to be specified. See **Concrete Heat of Hydration** for more details.

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.

If stress in the concrete due to the hydration process is required a structural/thermal **coupled analysis** should be carried out.

Thermal Link Material (Linear)

Thermal link material is defined from the **Attributes > Material** menu.

Thermal links are used to analyses heat conduction at contacting interfaces.

Specify the thermal link properties that will be used.

- ❑ **Gap conductance**
- ❑ **Convective heat transfer coefficient**
- ❑ **Radiative heat transfer coefficient**
- ❑ **Variation of gap conductance with distance**
- ❑ **Variation of gap convective coefficient with distance**
- ❑ **Variation of radiative coefficient with distance**

Thermal link material (Nonlinear)

Thermal link material is defined from the **Attributes > Material** menu

Specify the thermal link properties that will be used.

- ❑ **Conductance**
- ❑ **Convective coefficient**
- ❑ **Radiative coefficient**
- ❑ **Total distance**

Thermal User (Nonlinear)

For advanced use allowing entering of material parameters for user-defined 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

Hygro-thermal Material (Isotropic)

Hygro-thermal isotropic materials can be defined accessed using the **Attributes> Material>** menu item, providing a hygro-thermal analysis entry is present in the Analyses Treeview. If it is not, use the **Analyses > Hygro-Thermal Analysis** menu item to create one. Isotropic hygro-thermal material is applicable to all thermal elements except thermal links (see the *Element Reference Manual*)

For a hygro-thermal analysis two material types are available:

- ❑ **Hygro-thermal concrete** (for concrete)
- ❑ **Hygro thermal** (for general use, and for use with concrete)

Hygro-thermal material properties are applicable to hygro-thermal elements only. If a hygro-thermal element is to represent a non-porous material (e.g. steel) it should be assigned thermal isotropic properties.

Initial values must be specified explicitly; they cannot be computed by a steady state analysis as is the case for a transient field problem. To aid in specifying these values relevant equations, notes and references are provided.

Hygro-thermal concrete material

Density and specific heat capacity can be defined for either **Kiln dry concrete** or **Concrete at ambient conditions**.

Porosity can be **Automatically determined** or **Specify**. See the Porosity entry below for details.

General parameters:

- ❑ **Thermal conductivity**
$$k_T(T) = \begin{cases} \left[k_{T0} + k_{T1}(T - T_{Ref}) \right]\left(1 + 4\dfrac{\bar{\rho}_w}{\bar{\rho}_s} \right) & \text{if dependent on water saturation} \\ k_{T0} + k_{T1}(T - T_{Ref}) & \text{otherwise} \end{cases}$$

where

k_{T0} thermal conductivity (input in the main dialog)

k_{T1} change of thermal conductivity with temperature (input in Advanced dialog)

$\bar{\rho}_w$ mass of water in a unit volume of concrete. Varies during solution.

$\bar{\rho}_s$ mass of solid skeleton in a unit volume of concrete. Computed from density and porosity.

T_{Ref} reference temperature. Fixed as 20 °C (or its equivalent in °K)

Examples, units [J/s.m.K]

$$k_T(T) = k_{T0} + k_{T1}(T - T_{ref}) = 1.952 - 0.0024(T - 20)$$ high strength concrete (*)

$$k_T(T) = k_{T0} + k_{T1}(T - T_{ref}) = 1.333 - 0.00133(T - 20)$$ normal weight concrete (*)

$$k_T(T) = k_{T0} = 2.7$$ quartz aggregate, [B1]

$$k_T(T)|= k_{T0} = 1.5$$ bazalt aggregate, [B1]

$$k_T(T) = k_{T0} = 1.4$$ granite aggregate, [B1]

$$k_T(T) = k_{T0} = 1.25$$ anorthozite aggregate, [B1]

$$k_T(T) = k_{T0} = 0.3 \text{ to } 0.6$$ expanded shale aggregate, [B1]

$$k_T(T) = k_{T0} = 2.1 \text{ to } 2.7$$ wet concrete, [B1]

See **References**

❑ **Specific heat capacity**
   and

❑ **Density**

Specific heat capacity and density are calculated from:

$$C_s(T) = C_{T0} + C_{T1}(T - T_{Ref})$$
$$\rho_s(T) = \rho_{T0} + \rho_{T1}(T - T_{Ref})$$

where:

C_{T0}, ρ_{T0} input values in the main dialog

C_{T1}, ρ_{T1} change property with temperature

Examples, units [as shown]

$C_s(T) = C_{T0} = 900$ [J/kg.K]   dry concrete, siliceous or calcareous aggregates, [E1]

$\rho_s(T) = \rho_{T0} = 1.4$ [kg/m³]   expanded shale aggregate concrete, [B1]

$\rho_s(T) = \rho_{T0} = 2.1$ [kg/m³]   siliceous aggregate concrete, [B1]

$\rho_s(T) = \rho_{T0} = 2.3$ [kg/m³]   quartzite aggregate concrete, [B1]

$\rho_s(T) = \rho_{T0} = 2.5$ [kg/m³]   bazalt aggregate concrete, [B1]

See **References**

Specific heat and density are used to compute the volumetric heat capacity $\overline{\rho C}$. This property is assumed to be either experimentally determined for the concrete as a whole (option: Concrete at ambient condition) or it is computed via the rule of mixtures of the volumetric heat capacities of its constituents.

Concrete at ambient condition: $\overline{\rho C} = \rho_s C_s$

Kiln dry concrete: $\overline{\rho C} = \overline{\rho}_s C_s + \overline{\rho}_w C_w + \overline{\rho}_g C_g$

where

$\overline{\rho}_s = (1-n)\rho_s$	mass of dry skeleton per unit volume of concrete
$\overline{\rho}_w = nS_w\rho_w$	mass of liquid water per unit volume of concrete
$\overline{\rho}_g = n(1-S_w)\rho_g$	mass of gaseous mixture (air and water vapour) per unit volume of concrete
ρ_g	density gaseous mixture (computed from partial pressures)
ρ_w	density of water
S_w	water saturation (computed from capillary pressure)
n	porosity (either input or computed from the degree of hydration)
$C_g = 1400$ [J/kg.K]	specific heat of the gaseous mixture (assumed constant)
$C_w = 4180$ [J/kg.K]	specific heat of liquid water

❑ **Permeability**   The correct name is intrinsic permeability K_I and it has a large variation - from $10^{-16} \, m^2$ to $10^{-20} \, m^2$. It varies with temperature and degree of hydration, Γ:

$$k_I = K_I \times 10^{0.005(T-T_{ref})+2.25(1-\Gamma/T_\infty)}$$

It is used in Darcy's law and relates the gradient of the water pressure to the liquid water flux.

$$\mathbf{J}_w = -\rho_w \frac{K_w k_I}{\mu_w}(\nabla P_w - \rho_w \mathbf{g})$$

An alternative flux definition using permeability a [m/s] is also used:

$$\mathbf{J}_w = -\frac{a}{g}(\nabla P_w - \rho_w \mathbf{g})$$

Equating the fluxes and using material properties for water ($\rho_w = 1000$ kg/m³, viscosity $\mu_w = 0.001$ Pa.s, gravity g ≈ 10m/s²; and assuming a saturated flow, $K_w = 1$, and $k_I = K_I$) gives

$$K_I[m^2] \approx a[m/s] \times 10^{-7}$$

Care should be taken not to mix these two permeabilities when data is taken from the literature.

❑ **Entrapped air volume ratio**   This is only listed if the Exothermic behaviour option 'Concrete heat of hydration' is chosen on the same dialog.

❑ **Porosity**   This is only listed if the Porosity option 'Specify' is chosen.

For cured, old concrete a porosity value must be input and it stays constant during the analysis. A typical value lies between 0.08 to 0.17, with lower values being used for concrete with higher strength.

For curing concrete, i.e. one with heat of hydration, initial and final porosities are automatically determined. The initial porosity will accommodate the amount of liquid water and

entrapped air. The final porosity is calculated from the degree of hydration on the 28th day Γ_{28}, the composition of the concrete and empirical data. Intermediate values are linearly interpolated as function of the degree of hydration Γ.

References

(*) adapted from 'BS EN 1992-1-2:2004 Eurocode 2. Design of concrete structures. General rules. Structural fire design', to best fit given quadratic relation for the temperature interval 20 to 90 degrees Celcius.

(B1) Z. P. Bazant, M. F. Kaplan. Concrete at high temperatures, Longman 1996.

(E1) 'BS EN 1992-1-2:2004 Eurocode 2. Design of concrete structures. General rules. Structural fire design.'

Exothermic behaviour

For concrete heat of hydration analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, exothermic behaviour settings and values can be specified as:

- ❑ **None** No exothermal reaction is to take place.
- ❑ **Concrete heat of hydration** when selected, provides the means to specify the following cement types: **I**, **II**, **IV**, **V** and **General**, with each requiring additional parameters to be specified. See **Concrete Heat of Hydration** for more details.

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.

If calculation of stress in the concrete due to the hydration process is required a **coupled hygro-thermal/structural analysis** should be carried out.

Advanced properties

Calculation of thermal conductivity

- ❑ **Thermal conductivity depends upon water saturation** (when checked varies conductivity with water saturation. When unchecked is constant)

General parameters:

- ❑ **Tortuosity coeffiecient** Diffusion of water vapour in air is governed by the mean free path or average distance between collisions for the gas molecules. The tortuosity coefficient accounts for the reduced diffusion due to collisions with the walls of the concrete twisted pore network.

The diffusive vapour flux $\mathbf{J}_{v,diff}$ and the diffusion coefficient D_{va} are

$$\mathbf{J}_{v,diff} = \overline{\rho}_g D_{va} \nabla \left(\frac{\rho_v}{\rho_g} \right) \qquad D_{va} = f_z D_{va,0} \left(\frac{T}{T_{ref}} \right)^{1.667}$$

where

ρ_v	density of water vapour (computed from the partial pressure)
$D_{va,0} = 2.58 \times 10^{-5}$ [m^2/s]	diffusion coefficient of water vapour in air, at ambient conditions
f_z	tortuosity coefficient.

❑ **Latent heat of evaporation** The latent heat of water evaporation depends on the temperature. An averaged value of H=2.4x10^6 [J/kg] can be used for the usual temperature range of concrete hygro-thermal analyses. When water vapour is transferred into the concrete by environmental loading and subsequently condenses and increase the water saturation, S_w, heat is released proportional to H. When liquid water is transferred to the concrete by a prescribed loading with RH=1.0 the increase of saturation should not require release of latent heat. For these elements, during the duration of this load, the latent heat is set to zero, H=0.

❑ **Change of thermal conductivity with temperature** See note.

❑ **Change of specific heat capacity with temperature** See note.

❑ **Change of density with temperature** See note.

Note. Change of thermal / specific heat capacity / density with temperature options allow for increase / decrease of parameters according to temperature (linear variation). For example, change in thermal conductivity / degree Celcius, the units are J/s.m K/K = J/s.m.

Darcy's equation

Darcy's equation describes the flow of fluid through a porous medium. Choose either **Use gravity**, **Exclude** gravity or **Specify** a value for acceleration.

Hygro-thermal material

Hygro-thermal material properties are assumed to be constant with the exception of the relative water permeability, which is a function of the water saturation.

Darcy's equation

Darcy's equation describes the flow of fluid through a porous medium. Choose either **Use gravity**, **Exclude** gravity or **Specify** a value for acceleration.

General parameters:

❑ **Thermal conductivity** is calculated from:

$$k_T(T) = \begin{cases} \left[k_{T0} + k_{T1}(T - T_{Ref})\right]\left(1 + 4\dfrac{\overline{\rho}_w}{\overline{\rho}_s}\right) & \text{if dependent on water saturation} \\ k_{T0} + k_{T1}(T - T_{Ref}) & \text{otherwise} \end{cases}$$

where

k_{T0}	thermal conductivity (input in the main dialog)
k_{T1}	change of thermal conductivity with temperature (input in Advanced dialog)
$\overline{\rho}_w$	mass of water in a unit volume of concrete. Varies during solution.
$\overline{\rho}_s$	mass of solid skeleton in a unit volume of concrete. Computed from density and porosity.
T_{Ref}	reference temperature. Fixed as 20 °C (or its equivalent in °K)

Examples, units [J/s.m.K]

$$k_T(T) = k_{T0} + k_{T1}(T - T_{ref}) = 1.952 - 0.0024(T - 20) \quad \text{high strength concrete (*)}$$
$$k_T(T) = k_{T0} + k_{T1}(T - T_{ref}) = 1.333 - 0.00133(T - 20) \quad \text{normal weight concrete (*)}$$
$$k_T(T) = k_{T0} = 2.7 \quad \text{quartz aggregate, [B1]}$$
$$k_T(T)|= k_{T0} = 1.5 \quad \text{bazalt aggregate, [B1]}$$
$$k_T(T) = k_{T0} = 1.4 \quad \text{granite aggregate, [B1]}$$
$$k_T(T) = k_{T0} = 1.25 \quad \text{anorthozite aggregate, [B1]}$$
$$k_T(T) = k_{T0} = 0.3 \text{ to } 0.6 \quad \text{expanded shale aggregate, [B1]}$$
$$k_T(T) = k_{T0} = 2.1 \text{ to } 2.7 \quad \text{wet concrete, [B1]}$$

See **References**

❑ **Specific heat capacity** and

❑ **Density** are calculated from:

Specific heat capacity and density are calculated from:

$$C_s(T) = C_{T0} + C_{T1}(T - T_{Ref})$$
$$\rho_s(T) = \rho_{T0} + \rho_{T1}(T - T_{Ref})$$

where:

C_{T0}, ρ_{T0}	input values in the main dialog
C_{T1}, ρ_{T1}	change property with temperature

Examples, units [as shown]

$$C_s(T) = C_{T0} = 900 \ \text{[J/kg.K]} \quad \text{dry concrete, siliceous or calcareous aggregates, [E1]}$$
$$\rho_s(T) = \rho_{T0} = 1.4 \ \text{[kg/m}^3\text{]} \quad \text{expanded shale aggregate concrete, [B1]}$$
$$\rho_s(T) = \rho_{T0} = 2.1 \ \text{[kg/m}^3\text{]} \quad \text{siliceous aggregate concrete, [B1]}$$
$$\rho_s(T) = \rho_{T0} = 2.3 \ \text{[kg/m}^3\text{]} \quad \text{quartzite aggregate concrete, [B1]}$$
$$\rho_s(T) = \rho_{T0} = 2.5 \ \text{[kg/m}^3\text{]} \quad \text{bazalt aggregate concrete, [B1]}$$

See **References.**

Specific heat and density are used to compute the volumetric heat capacity $\overline{\rho C}$. This property is assumed to be either experimentally determined for the concrete as a whole (option: Concrete at ambient condition) or it is computed via the rule of mixtures of the volumetric heat capacities of its constituents.

Concrete at ambient condition: $\qquad \overline{\rho C} = \rho_z C_z$

Kiln dry concrete: $\qquad \overline{\rho C} = \overline{\rho}_z C_z + \overline{\rho}_w C_w + \overline{\rho}_g C_g$

where

$\overline{\rho}_z = (1-n)\rho_z$	mass of dry skeleton per unit volume of concrete
$\overline{\rho}_w = nS_w\rho_w$	mass of liquid water per unit volume of concrete
$\overline{\rho}_g = n(1-S_w)\rho_g$	mass of gaseous mixture (air and water vapour) per unit volume of concrete
ρ_g	density gaseous mixture (computed from partial pressures)
ρ_w	density of water
S_w	water saturation (computed from capillary pressure)
n	porosity (either input or computed from the degree of hydration)
$C_g = 1400$ [J/kg.K]	specific heat of the gaseous mixture (assumed constant)
$C_w = 4180$ [J/kg.K]	specific heat of liquid water

❑ **Latent heat (of phase change)**

The latent heat of water evaporation depends on the temperature. An averaged value of $H=2.4\times10^6$ [J/kg] can be used for the usual temperature range of concrete hygro-thermal analyses. When water vapour is transferred into the concrete by environmental loading and subsequently condenses and increase the water saturation, S_w, heat is released proportional to H. When liquid water is transferred to the concrete by a prescribed loading with RH=1.0 the increase of saturation should not require release of latent heat. For these elements, during the duration of this load, the latent heat is set to zero, H=0.

❑ **Porosity of medium**

For cured, old concrete a porosity value must be input and it stays constant during the analysis. A typical value lies between 0.08 to 0.17, with lower values being used for concrete with higher strength.

For curing concrete, i.e. one with heat of hydration, initial and final porosities are automatically determined. The initial porosity will accommodate the amount of liquid water and entrapped air. The final porosity is calculated from the degree of hydration on the 28th day Γ_{28}, the composition of the concrete and empirical data. Intermediate values are linearly interpolated as function of

the degree of hydration Γ.

❑ **Permeability** The correct name is intrinsic permeability K_I and it has a large variation - from $10^{-16}\ m^2$ to $10^{-20}\ m^2$. It varies with temperature and degree of hydration, Γ:

$$k_I = K_I \times 10^{0.005(T-T_{ref})+2.25(1-\Gamma/\Gamma_\infty)}$$

It is used in Darcy's law and relates the gradient of the water pressure to the liquid water flux.

$$\mathbf{J}_w = -\rho_w \frac{K_w k_I}{\mu_w}(\nabla P_w - \rho_w \mathbf{g})$$

An alternative flux definition using permeability a [m/s] is also used:

$$\mathbf{J}_w = -\frac{a}{g}(\nabla P_w - \rho_w \mathbf{g})$$

Equating the fluxes and using material properties for water ($\rho_w = 1000$ kg/m^3, viscosity $\mu_w = 0.001$ Pa.s, gravity g $\approx$ 10m/s^2; and assuming a saturated flow, $K_w = 1$, and $k_I = K_I$) gives

$$K_I[m^2] \approx a[m/s] \times 10^{-7}$$

Care should be taken not to mix these two permeabilities when data is taken from the literature.

❑ **Tortuosity coefficient** Diffusion of water vapour in air is governed by the mean free path or average distance between collisions for the gas molecules. The tortuosity coefficient accounts for the reduced diffusion due to collisions with the walls of the concrete twisted pore network.

The diffusive vapour flux $\mathbf{J}_{v,diff}$ and the diffusion coefficient D_{va} are

$$\mathbf{J}_{v,diff} = \overline{\rho}_g D_{va} \nabla \left(\frac{\rho_v}{\rho_g} \right) \qquad D_{va} = f_z D_{va,0} \left(\frac{T}{T_{ref}} \right)^{1.667}$$

where

ρ_v	density of water vapour (computed from the partial pressure)
$D_{va,0} = 2.58 \times 10^{-5}$ [m^2/s]	diffusion coefficient of water vapour in air, at ambient conditions
f_z	tortuosity coefficient.

❑ **Acceleration X/Y/Z component** This is only listed if the option 'Specify Accelartion' is chosen for Darcy's equation. Define acceleration components for X, Y and Z.

References

(*) adapted from 'BS EN 1992-1-2:2004 Eurocode 2. Design of concrete structures. General rules. Structural fire design' to best fit given quadratic relation for the temperature interval 20 to 90 degrees Celcius.

(B1) Z. P. Bazant, M. F. Kaplan. Concrete at high temperatures, Longman 1996.

(E1) 'BS EN 1992-1-2:2004 Eurocode 2. Design of concrete structures. General rules. Structural fire design.'

Concrete Models (Nonlinear)

Nonlinear concrete material models can be accessed from using the **Attributes> Material> Concrete (Nonlinear)** menu item.

Concrete models

Selection of a nonlinear concrete material model provides the means to define concrete with or without respect to a design code, and whether to exclude or include creep and/or shrinkage if a design code supports it.

- ❑ **Linear with creep/shrinkage (model 86)** - This material model uses a simplified linear approach to represent creep and is nonlinear in the time domain. The model 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.

- ❑ **Transient smoothed multi-crack (model 105)** This concrete model is fundamentally the same as concrete model 109, but can only be accessed if a hygro-thermal/structural coupled analysis is specified. The material model takes into account the effects of heat of hydration, creep and shrinkage, along with the hydration dependent evolution of properties E, fc and ft.

- ❑ **Smoothed multi-crack (model 109)** - This 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.

All three material models are for use with 2D and 3D continuum elements, 2D and 3D explicit dynamics elements, solid composite elements and semiloof or thick shell elements only.

Superceded concrete models

When a pre-version 17 model is loaded that uses older versions of concrete models such as the Cracking concrete model (model 82), the Cracking with crushing concrete model (model 84), and the Multi-crack concrete model (model 94) - those models will

be listed in the concrete material droplist in addition to the current valid concrete models.

Creep and shrinkage material models

Creep and shrinkage material models are nonlinear in the time domain. They may be used in conjunction with certain concrete models. See table for support by concrete material model.

- ❑ **AASHTO LRFD 7th Edition** - Applicable for specified concrete strengths up to 15.0 ksi. Article 5.4.2.3.1 contains a commentary with a caveat stating the accuracy of results that can be expected to be obtained in certain circumstances. Not supported by Transient smoothed multi-crack (model 105).
- ❑ **CEB-FIP Model Code 1990** Valid for ordinary structural concrete (12-80 MPa) where the compressive stress in concrete is less than 40% of its compressive strength at the time of loading.
- ❑ **EN1992-1-1:2004 Eurocode 2** - Appropriate for cases where the compressive stress in concrete is less than 45% of its compressive strength at the time of loading.
- ❑ **IRC:112-2011** - Appropriate for cases where the compressive stress in concrete is less than 36% of its characteristic compressive strength at the time of loading. Not supported by Transient smoothed multi-crack (model 105).
- ❑ **User** - Only supported by Transient smoothed multi-crack (model 105). Permits user-defined parameters to be specified.

For linear elastic concrete material models see those provided in the Material Library.

Creep only models

- ❑ **Chinese Code** Based upon equations taken from the Chinese Creep Code for Dams. This 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. Shrinkage is not included.

Creep and shrinkage support by concrete material model

Concrete model	AASHTO LRFD 7th Edition	CEB-FIP Model Code 1990	Chinese Code (Draft 2007)	EN1992-1-1:2004 Eurocode 2	IRC:112-2011	User
Linear with creep and shrinkage (model 86)	Creep and shrinkage supported	Creep and shrinkage supported	Creep only supported	Creep and shrinkage supported	Creep and shrinkage supported	Not supported
Transient smoothed multi-crack (105)	Not supported	Creep and shrinkage supported	Not supported	Creep and shrinkage supported	Not supported	Creep and shrinkage supported
Smoothed multi-crack (model 109)	CCreep and shrinkage supported	Creep and shrinkage supported	Creep only supported	Creep and shrinkage supported	Creep and shrinkage supported	Not supported

Notes

- The creep and shrinkage models are only applicable to beams, but 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.

- When a **Structural analysis** is being carried out the Linear with creep/shrinkage concrete, and the Smoothed multi-crack concrete models are available.

- When a **Coupled Hygro-Thermal/Structural analysis** is being carried out, the Transient smoothed multi-crack concrete model (Model 105) should be used.

- Refer to the help pages for each design code for more information.

See the *Solver Reference Manual* and the *Theory Manual* for further details of all models.

Concrete Creep and Shrinkage to AASHTO LRFD 7th edition

Concrete material properties to AASHTO LRFD 7th edition are defined from the **Attributes> Material> Concrete (Nonlinear)**menu item.

This material model is based upon the concrete creep and shrinkage rules set out in AASHTO LRFD Bridge Design Specifications 7th Edition Article 5.4.2.3. Note that

this code and associated commentary offer different instructions for calculating creep coefficients. LUSAS allows you to choose between them:

- Creep coefficient can be calculated according to Article 5.4.2.3.2, or to the corresponding Commentary, adjusted according to AASHTO LRFD 2nd Edition.

- Shrinkage strain can be determined using the factor for the effect of volume-to-surface ratio (ks) from equation 5.4.2.3.2-2, or from the Commentary equation C5.4.2.3.2-2.

- Early age elastic modulus and compressive strength can be determined using the approximation of article 5.4.2.3.2, or by reference to CEB-FIP Model Code 1990 clause 2.1.6.1

As per Article 5.4.2.3.1, the AASHTO provisions may be regarded as applicable for specified concrete strengths up to 15.0 ksi, with the caveat that the Commentary indicates that "Without specific physical tests or prior experience with the materials, the use of the empirical methods ... cannot be expected to yield results with errors less than ±50 percent".

References cited are to AASHTO LRFD Bridge Design Specifications 7th Edition unless otherwise stated.

Material Properties

The following material parameters are required depending upon other settings made on the dialog. All parameters relate to when 'Elastic modulus from strength and weight class' is selected, unless where noted:

- ❑ **Young's modulus** This value is only required to be entered if the option 'Input elastic modulus and density' is used. In other cases, the modulus of elasticity is determined from the compressive strength, weight class and aggregate correction factor, using equation 5.4.2.4-1. A default value of E_c=4070 ksi is given, corresponding to normal weight concrete with f'_c = 5.0ksi and an aggregate correction factor of K_1=1.0. The value entered may either be used directly or modified for early age stiffness gain by selecting the option "Modify using CEB-FIP 1990".

- ❑ **Poisson's Ratio** - The default is v=0.2 as per article 5.4.2.5.

- ❑ **Mass density** ρ. This value is only required to be entered if the option 'Input elastic modulus and density' is used. In other cases, the density is determined from the weight class using Table 3.5.1-1. A default value of ρ =0.145/32.2 kslug/ft³ is given, corresponding to normal weight concrete with f'_c = 5.0ksi.

- ❑ **Coefficient of thermal expansion** (available when Thermal properties are defined) may be taken as α=6E-6/°F according to Article 5.4.2.2.

- ❑ **Mass Rayleigh damping constant** (available when Dynamic properties are specified)
- ❑ **Stiffness Rayleigh damping constant** (availabel when Dynamic properties are specified)

Properties for use in creep and shrinkage calculations:

- ❑ **Weight class** need only be specified if the option 'Elastic modulus from strength and weight class' is selected. The droplist options then correspond to Table 3.5.1-1 i.e. **Lightweight**, **Sand-Lightweight** and **Normal Weight** (the default). The appropriate value unit weight, w_c, is determined from Table 3.5.1-1 and used for calculation of the modulus of elasticity using equation 5.4.2.4-1, and also to determine the density, ρ, for structural loading purposes.

- ❑ **Uniaxial compressive strength** f'_c. The value entered is used in the determination of the early age compressive strength for the concrete (using the approximation of article 5.4.2.3.2 or by reference to CEB-FIP Model Code 1990 clause 2.1.6.1, depending on the option selected). It is also used in the determination of Elastic modulus from equation 5.4.2.4-1 when the option "Elastic modulus from strength and weight class" is selected. A default value of f'_c=5.0ksi is given.

- ❑ **Aggregate correction factor** This value is only required to be entered if the option "Elastic modulus from strength and weight class" is selected. The value of K_1 would then be used for calculation of the modulus of elasticity using equation 5.4.2.4-1. A default of K_1=1.0 is given as per Article 5.4.2.4.

- ❑ **Cement type** need only be specified if the early age elastic modulus and compressive strength is to be determined according to the option "Modify using CEB-FIP 1990". The properties in question are determined from CEB-FIB Model Code 1990 clauses 2.1.6.3 and 2.1.6.1 respectively and accordingly the droplist options are: **SL (slow)** (slow hardening cements), **N and R (normal and rapid)** (normal and rapid hardening cements), or **RS (rapid high strength)** (rapid hardening high strength cements). Cement type 'N and R (normal and rapid) is the default option.

- ❑ **Relative humidity** - is used in the calculation of both creep coefficient and, where required, shrinkage. Typical percentage values would lie in the range $30 \le H \le 81$ from Figure 5.4.2.3.3-1. The default value is H= 70%.

- ❑ **Internal perimeter factor** or **Volume to surface ratio** appear depending upon selections made on the Options dialog. See 'Calculation of volume-to-surface ratio' for more details.

Options

Creep coefficient

The creep coefficient is calculated using one of two available methods, described below. It may be appropriate to compare the two approaches for structures which have a high sensitivity to creep.

- ❑ **Using article 5.4.2.3.2** This option calculates the creep coefficient using eqn 5.4.2.3.2-2 to determine k_s (the factor for the effect of the volume-to-surface ratio of the component), rather than using the alternative commentary clause. This option is fully compliant with AASHTO 7th but is using an approach described in the commentary as 'an approximation' for the value of k_s.

- ❑ **Using commentary and 2nd Edition** This option calculates the creep coefficient using the commentary clause C5.4.2.3.2-2 to determine k_c, which is used in place of k_s. This approach generally leads to values of $k_c<1.0$ and, in tests, gave creep effects somewhat lower than the expected range. Comparison with the 2nd edition identified the original use of k_c with a 3.5 multiple in the creep coefficient, rather than 1.9 as in later editions. Accordingly this option adopts the 3.5 multiple and has been found give creep effects within the expected range for the tests undertaken. This option is therefore not fully compliant with AASHTO 5th to 7th but may be deemed to be a more accurate approach.

In either case, coefficients k_f and k_{td} for creep may be evaluated using the approximation for f'_{ci} of article 5.4.2.3.2 or by reference to CEB-FIP Model Code 1990 clause 2.1.6.1, depending on the option selected.

Shrinkage strains

Shrinkage strains can also be determined by one of two available methods, described below. It may be appropriate to compare the two approaches for structures which have a high sensitivity to shrinkage.

- ❑ **Using equation 5.4.2.3.2-2** This option calculates the strain due to shrinkage using eqn 5.4.2.3.2-2 to determine k_s (the factor for the effect of the volume-to-surface ratio of the component), rather than using the alternative commentary clause. Eqn 5.4.2.3.2-2 leads to values of $k_s=1.0$ and, in tests, gave shrinkage effects ranging from those within the expected range to those considerably larger. This option is fully compliant with AASHTO 5th to 7th but is using an approach described in the commentary as 'an approximation' for the value of k_s.

- ❑ **Using commentary** This option calculates the strain due to shrinkage using the commentary clause C5.4.2.3.2-2 to determine k_s. The commentary generally leads to values of $k_s<1.0$ and, in tests, gave shrinkage effects ranging from those in the expected range to those considerably lower. This option is fully compliant with the commentary approach set out in AASHTO 5th to 7th, and therefore may be deemed to be a more accurate approach, however, caution is recommended.

In either case, coefficients k_f and k_{td} for shrinkage are evaluated using the approximation $f'_{ci}=0.8*f'_c$ as the appropriate age for determining f'_{ci} in the context of shrinkage is not properly defined in the Code Of Practice.

Calculation of volume-to-surface section size

Volume-to-surface ratio can be calculated automatically for beams from the section dimensions using either a full perimeter (outer and inner, if voids are present) or by specifying an internal perimeter factor. As per Article 5.4.2.3.2, "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.". This Article indicates that an internal parameter factor of 0.5 is generally appropriate.

The computation is carried out for each element, and for tapered sections the value will vary along the beam. Alternatively a volume-to-surface ratio may be specified directly.

- ❑ **Calculate using section dimensions or thickness (lines and surfaces)** causes the volume-to-surface ratio to be calculated as $V/S = A_c/u$ 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. For shells, the V/S ratio is half the assigned geometric thickness.

- ❑ **Adjusted using internal perimeter factor for voids (lines only)** (for Linear with creep/shrinkage (Model 86) only). Causes 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 $V/S = A_c/u$. For solid sections the value of internal perimeter factor is irrelevant. The default value is 0.5 as per Article 5.4.2.3.2, as quoted above.

- ❑ **Specified directly (any geometry type)** The volume-to-surface ratio stated should be compatible with the modelling units in use.

Elastic modulus and compressive strength for creep calculation for age < 28 days

- ❑ **Modify using CEB-FIP 1900** When this option is selected, coefficients k_f and k_{td} for creep are calculated using CEB-FIP Model Code 1990 clause 2.1.6.1, and the elastic modulus for concrete of less than 28 days age using clause 2.1.6.3. This option requires definition of 'Cement type' in the main parameters.

- ❑ **Use 28 day values** When this option is selected, coefficients k_f and k_{td} for creep are calculated using the approximation for f'_{ci} of Article 5.4.2.3.2, and the elastic modulus for concrete used is not adjusted from that entered in the main parameters (or determined from equation 5.4.2.4-1).

Notes

- The AASHTO LRFD 7th edition creep and shrinkage model can be used with beam elements, 2D and 3D continuum elements, semi-loof and thick shells, and composite solid elements. AASHTO LRFD 7th 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 "notional size" for continuum elements.
- The AASHTO LRFD 7th edition 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, along with the age at onset of drying shrinkage, must be defined using an **Age** attribute.

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> Concrete (Nonlinear)** menu item

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.

- ❑ **Young's modulus** - 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 $27 \times 10^3 \le E_{ci} \le 44 \times 10^3$ MPa. Taking into account the

adjustment for aggregate, the expected range of values is therefore $18 \times 10^3 \leq E_{ci} \leq 53 \times 10^3$ MPa. The default value of $E_{ci} = 36 \times 10^3$ 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 $\rho = 2500$kg/m³.

❑ **Coefficient of thermal expansion** (when Thermal properties are specified) - may be taken as 10E-6 per degree K according to clause 2.1.8.3.

❑ **Mass Rayleigh damping constant** (when Dynamic properties are specified)

❑ **Stiffness Rayleigh damping constant** (when Dynamic properties are specified)

Properties for use in creep and shrinkage calculations:

❑ **Uniaxial compressive strength** - 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, $f_{cm} = F_{ck} + 8$MPa. For grades C12 to C80, as per clause 2.1.1.2, this would give a likely range $20 \leq f_{cm} \leq 88$MPa. The default value of $f_{cm} = 48$MPa, 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 **Notional size** appear depending upon selections made on the Options dialog. See 'Calculation of volume-to-surface ratio' for more 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 $\sigma_c \leq 0.4 f_{cm}$ (t_0).

Shrinkage

The effects of shrinkage can be included in accordance with clause 2.1.6.4.4.

Thermal properties

When specified requires the coefficient of thermal expansion to be defined.

Dynamic properties

When specified requires Mass Rayleigh and Stiffness Rayleigh damping constants to be defined.

Damage-based options

Specification of **Fracture energy** or **Strain at end of softening curve** via selection of the appropriate radio button on the dialog may also be applicable for certain selections.

Options

Notional section size

Notional 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 notional section size may be specified directly. For beams the notional size of member is computed automatically by LUSAS for each element, and for tapered sections will vary along the beam. The availability of the following options is concrete model dependent:

- ❑ **Calculate using section dimensions or thickness (lines and surfaces)** When applied to lines, causes the notional 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. When applied to surfaces, causes the notional size to be equal to the surface thickness. This is a good approximation of $2A_c/u$ when the surface thickness is negligable compared to the other dimension of the surface cross-section. If this is not the case, calculate $2A_c/u$ by hand and use the option 'Specified directly (any geometry type).'

- ❑ **Adjusted using internal perimeter factor for voids (lines only)** (for 'Linear with creep/shrinkage (Model 86)' only). Causes 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 notional 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.

❑ **Specified directly (any geometry type)** For this the notional size entered should be entered in the model units for length.

Notes

- The use of an internal perimeter factor to calculate a notional 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 continuum 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 a notional 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 are specified in Timescale units (as defined initially on the New Model dialog, and as can be checked using 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. All time values are tabulated to a Solver data file as seconds when this file is created.

- 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: Chinese Code for Concrete Hydraulic Structures(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:

$$^{\infty}E = 1.05\,^{360}E \text{ or } ^{\infty}E = 1.20\,^{90}E, \text{ A} = 0.40, \text{ B} = 0.34$$

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

Concrete Creep and Shrinkage to EN1992 (2004-2014)

Concrete material properties to EN1992-1-1:2004 are defined from the **Attributes> Material> Concrete (Nonlinear)** menu item.

The EN1992-1-1 creep and shrinkage material 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 characteristic compressive strength at the time of loading, as per EN1992-1-1 clause 3.1.4(2). This material model also includes options appropriate to EN1992-2:2005 Annex B, section B.103.

Clause references cited are to EN1992-1-1:2004+A1:2014 unless otherwise stated.

Material Properties

The following material parameters are required:

- ❑ **Young's modulus** - the tangent modulus of elasticity of concrete at 28 days, E_c. This value is typically determined from the secant modulus, E_{cm}, in Table 3.1, adjusted for aggregate type according to the commentary to clause 3.1.3(2) and using $E_c = 1.05E_{cm}$ 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 $19 \times 10^3 \leq E_c \leq 56 \times 10^3$ MPa. The default value of $E_c = 36.75 \times 10^3$ MPa, corresponds to concrete with $f_{ck} = 40$MPa 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 $v_c = 0.2$, 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 $\rho = 25$ kN/m³ = 2548.4kg/m³ for normal weight concrete according to EN1991-1-1:2002 Table A.1
- ❑ **Coefficient of thermal expansion** (when Thermal properties are defined) - may be taken as 10E-6 per K according to clause 3.1.3(5).
- ❑ **Mass Rayleigh damping constant** (when Dynamic properties are specified)

275

❑ **Stiffness Rayleigh damping constant** (when Dynamic properties are specified)

Properties for use in creep and shrinkage calculations:

❑ **Uniaxial compressive strength** - 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 $20 \leq f_{cm} \leq 98MPa$. The default value of $f_{cm} = 48MPa$, corresponds to $f_{ck} = 40Mpa$.

❑ **Cement type** - refers to cement type in clauses 3.1.2(6), 3.1.4(5) and Annex B, clause B.1(2) and may be: **Class S** (slow hardening cements), **Class N** (Normal or rapid 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 would lie in the range $20 \leq RH \leq 80$ from Table 3.2, and Figure 3.1 indicates a value of RH=50% may represent inside conditions, while RH=80% may represent outside conditions

❑ **Internal perimeter factor** or **Notional size** appear depending upon selections made on the Options dialog. See 'Calculation of notional section size' for more details.

Creep calculations

Creep calculations are carried out to EN1992-1-1 clause 3.1.4 and Annex B.1 with the following limitations and modifications:

- The effect of elevated or reduced temperatures on the maturity of concrete (clause B.1(3) and expression B.10) are not considered, meaning that the modulus of elasticity (exp. 3.5) and the notional creep coefficient (exp. B.5) are not adjusted. Likewise the effect of cement type on the creep coefficient (clause B.1(2) and exp. B.9) are not considered.

- Nonlinear effects of high stresses are not considered. According to clause 3.1.4(4), a nonlinear notional creep coefficient (exp. 3.7) should be used for areas where compressive stress $\sigma_c > 0.45 f_{ck}$ (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.

- EN1992-2:2005 Annex B, section B.103 gives expressions for creep in high-strength concrete ($f_{ck} > 50MPa$). Concretes with or without silica fume are treated differently as lower creep strains are observed in concrete with silica fume. The option may be switched on by the user as required; as well as being used for high strength concretes, the methods given in Section B.103 may be preferred for thick members, in which the kinetics of basic creep and drying creep are quite different, as per EN1992-2 Annex B. On the contrary, they would not be applicable when working strictly to EN1992-1-1.

Thermal properties

When specified requires the coefficient of thermal expansion to be defined.

Dynamic properties

When specified requires Mass Rayleigh and Stiffness Rayleigh damping constants to be defined.

Damage-based options

Specification of **Fracture energy** or **Strain at end of softening curve** via selection of the appropriate radio button on the dialog may also be applicable for certain selections.

Options

Notional section size

Notional section size can be calculated automatically for beams from the section dimensions using either a full perimeter (outer and inner, if voids are present) or by specifying an internal perimeter factor, which factors down the contribution of the internal perimeter in the calculation of the perimeter exposed to drying. The computation is carried out for each element, and for tapered sections the value will vary along the beam. Alternatively a notional section size may be specified directly. For shells the notional size is the assigned geometric thickness. The availability of the following options is concrete model dependent:

- ❑ **Calculate using section dimensions or thickness (lines and surfaces)** causes the notional 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.

- ❑ **Adjusted using internal perimeter factor for voids (lines only)** (for Linear with creep/shrinkage (Model 86) only). Causes 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 notional size, h_0, 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.

- ❑ **Specified directly (any geometry type)** The notional size stated should be compatible with the modelling units in use.

High strength concrete

❑ **Use EN1992-2 equations for high strength concrete (>50MPa)** This option determines the creep and shrinkage parameters using the alternative expressions of EN1992-2:2005 Annex B, section B.103 and may also be of use for thick members as described above.

❑ **Silica Fume ≥5% of cementitious content** This option is only available when the EN1992-2 equations for high strength concrete are in use. The option affects the calculation of drying shrinkage, basic creep and drying creep according to EN1992-2 Section B.103 as appropriate.

Shrinkage

❑ **Include / Exclude autogenous shrinkage** When code based shrinkage is selected, the effects of shrinkage can be included in accordance with clause 3.1.4(6) and Annex 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".

EN1992-2:2005 Annex B, section B.103 gives expressions for shrinkage in high-strength concrete (f_{ck}>50MPa). Concretes with or without silica fume are treated differently in the drying shrinkage calculation. The option may be switched on by the user as required; as well as being used for high strength concretes, the methods given in Section B.103 may be preferred for thick members, as described in EN1992-2 Annex B. On the contrary, they would not be applicable when working strictly to EN1992-1-1.

Notes

- The EN1992 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 "notional size" for continuum elements.

- The EN1992 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, along with the age at onset of drying shrinkage, must be defined using an **Age** attribute.

- Nonlinearity of creep due to high compressive stress (exp. 3.7); the effect of type of cement and curing temperature (exp. B.9); and the effect of temperature on the maturity of concrete (exp. B.10) 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 IRC:112-2011

Concrete material properties to IRC:112-2011 are defined from the **Attributes> Material> Concrete (Nonlinear)**menu item.

The IRC:112-2011 creep and shrinkage model uses a simplified linear approach to represent creep, which is deemed appropriate for cases where the compressive stress in concrete is less than 36% of its characteristic compressive strength, as per clause 6.4.2.7.

Material Properties

The following material parameters are required:

❑ **Modulus of elasticity of concrete** - the secant modulus of elasticity of concrete, E_{cm}. This value is typically determined from E_{cm} in Table 6.5 or by use of the equation in Annexure A2, Eq. A2-5, adjusted for aggregate type by Table 6.5 Note (2). The expected range of values is therefore $19 \times 10^3 \leq E_{cm} \leq 49 \times 10^3$ MPa. The default value of $E_{cm} = 33 \times 10^3$ MPa, corresponds to concrete with $f_{ck} = 40$MPa and quartzite/granite aggregate.

❑ **Poisson's Ratio** - may be taken as 0.2 for uncracked concrete and 0 for cracked concrete according to clause 6.4.2.5(4)(ii). A value of $v_c = 0.2$, corresponds to an uncracked section, acknowledging the primary use of the creep material in prestressed concrete where cracking is minimised.

❑ **Mass density** - is not specified in IRC:112-2011. A useful reference may be EN1992-1-1:2004, Table A.1, where a value of $\rho = 25$ kN/m³ = 2548.4kg/m³ for normal weight concrete is given

❑ **Coefficient of thermal expansion** - (when Thermal properties are specified) This is not specified in IRC:112-2011. A useful reference may be EN1992-1-1:2004, clause 3.1.3(5) where a value of 10E-6 per K is given.

❑ **Mass Rayleigh damping constant** (when Dynamic properties are specified)

❑ **Stiffness Rayleigh damping constant** (when Dynamic properties are specified)

Properties for use in creep and shrinkage calculations:

- ❑ **Uniaxial 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 Annexure A2, Eq. A2-1, i.e. $f_{cm} = f_{ck} + 10\text{MPa}$ or referring to Table 6.5. This would give a likely range $25 \le f_{cm} \le 100\text{MPa}$. The default value of $f_{cm} = 50\text{MPa}$, corresponds to $f_{ck} = 40\text{MPa}$.

- ❑ **Cement type** - refers to cement types described in Annexure A2, clauses A2.3(1) and A2.5(1) and may be: Rapid hardening, Normal hardening or Slow hardening. Normal hardening is the default option.

- ❑ **Relative humidity** - is used in the calculation of both creep coefficient and, where required, shrinkage. Typical percentage values would lie in the range $20 \le RH \le 80$, from Tables 6.8 and 6.9. The default of RH=50%, would represent dry atmospheric conditions.

- ❑ **Internal perimeter factor** or **Notional size** appear depending upon selections made on the Options dialog. See 'Calculation of notional section size' for more details.

Thermal properties

When specified requires the coefficient of thermal expansion to be defined.

Dynamic properties

When specified requires Mass Rayleigh and Stiffness Rayleigh damping constants to be defined.

Damage-based options

Specification of **Fracture energy** or **Strain at end of softening curve** via selection of the appropriate radio button on the dialog may also be applicable for certain selections.

Creep calculations

Creep calculations are carried out to IRC:112-2011 clause 6.4.2.7 and Annexure A2 with the following limitations:

- Elevated or reduced temperatures during curing are not considered. According to clause A2.5(1), the concrete age should be adjusted according to equation Eq. A2-13. This affects the characteristic strength (Eq. 6.3) and the notional creep coefficient (Eq. A2-19). Likewise, the effect of cement type on creep coefficient in Eqn. A2-25 is not considered.

Options

Notional section size

Notional section size can be calculated automatically for beams from the section dimensions using either a full perimeter (outer and inner, if voids are present) or by specifying an internal perimeter factor, which factors down the contribution of the internal perimeter in the calculation of the perimeter exposed to drying. The computation is carried out for each element, and for tapered sections the value will vary along the beam. Alternatively a notional section size may be specified directly. For shells the notional size is the assigned geometric thickness. The availability of the following options is concrete model dependent:

- ❑ **Calculate using section dimensions or thickness (lines and surfaces)** causes the notional size of section to be calculated according to clause 6.4.2.6(4) 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.

- ❑ **Adjusted using internal perimeter factor for voids (lines only)** (for Linear with cree /shrinkage (Model 86) only). Causes 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 notional size, h_0, following clause 6.4.2.6(4). 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.

- ❑ **Specified directly (any geometry type)** The notional size stated should be compatible with the modelling units in use.

Shrinkage

- ❑ **Include / Exclude autogenous shrinkage** When code based shrinkage is selected, the effects of shrinkage can be included in accordance with clause 6.4.2.6 and Annexure A2 clause A2.6. Clause 6.4.2.6 describes that "the total shrinkage strain is composed of two components, the drying shrinkage strain and the autogenous shrinkage strain... 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... The drying shrinkage strain develops slowly, since it is a function of the migration of the water through the hardened concrete"

Notes

- The IRC:112-2011 creep and shrinkage model can be used with beam elements, 2/3D continuum elements, semi-loof and thick shells, and composite solid elements. IRC:112-2011 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 "notional size" for continuum elements.

- The IRC:112-2011 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, along with the age at onset of drying shrinkage, must be defined using an **Age** attribute.

- The effect of type of cement on the creep coefficient in Eq. A2-25, and the effect of temperature curing on the maturity of concrete (Eqn. A2-13) are not currently considered and should be assessed using code requirements.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Transient Smoothed Multi Crack Concrete Model (Model 105)

The transient smoothed multi-crack concrete model is fundamentally the same as the **smoothed multi-crack concrete model (Model 109)** but it is intended for use in a hygro-thermal-structural coupled analysis, and takes into account the effects of heat of hydration, creep and shrinkage, along with the time (hydration) dependent evolution of properties E, fc and ft. Modelling units of N and m are enforced with teh use of this concrete model.

The hygro-thermal component considers the mass balance equation of moisture as well as the enthalpy balance equation, and uses two primary variables, namely the capillary pressure and temperature. The basic mechanical model simulates directional cracking, rough crack closure and crushing using a plastic-damage-contact approach which is hydration dependent.

- For this concrete model cracking and crushing 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 using star symbols.

- General crack width calculation is possible and can be plotted for the Contour and Values layers as component of CWMax (Maximum crack width) under the Plastic strain - Plane stress entity. Component EFSMax (Maximum equivalent fracture strain) is also available. Note that these components are only applicable for elements that support the multi-crack concrete model. See Appendix K: Results Tables in the *Element reference Manual* for details. See also the general notes relating to crack width calculation at the end of this topic.

- 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.

See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Staged Construction of a Concrete Dam' for a worked example.

Specification of Concrete model, Creep Code and Shrinkage

See **Concrete Models (Nonlinear)** for concrete model, creep and shrinkage options.

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 $E > 1.2\ f_c / \varepsilon_c$

❑ **Poisson's ratio**

❑ **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.

❑ **Coefficient of thermal expansion**

❑ **Mass Rayleigh damping constant** (if 'Dynamic properties' is selected).

❑ **Stiffness Rayleigh damping constant** (if 'Dynamic properties' is selected).

Plastic Material Properties

These primary material properties apply to both unreinforced (plain concrete) and reinforced concrete except where stated.

- ❑ **Uniaxial compressive strength** (f_c) e.g. 40E6 N/m^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. 3E6 N/m^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.

- ❑ **Strain at peak uniaxial compression** (ε_c) e.g. 2.2E-3. 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 2.2E-3. It is important that the Young's modulus, E, (defined on the Elastic properties page) is consistent with the strain at peak uniaxial compression, ec. A reasonable check is to ensure that E > 1.2 fc/ ec

Specification of **Fracture energy** or **Strain at end of softening curve** via selection of the appropriate radio button on the dialog is required. Specifying fracture energy is usually the more convenient option:

- ❑ **Fracture energy per unit area** (G_f) e.g. 130 N/m. 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 = 130 N/m is suggested. Typical values of G_f are: for 16mm aggregate, 100 N/m; for 20 mm aggregate, 130 N/m; and for 32 mm aggregate, 160 N/m.

- ❑ **Strain at end of softening curve** (ε_o) e.g. 3.5E-3.In some cases it may be deemed more suitable to specify the strain at end of softening curve. In such a case and where no data is available then a value for ε_0 of 3.5E-3 is a reasonable first estimate for most concrete. 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.
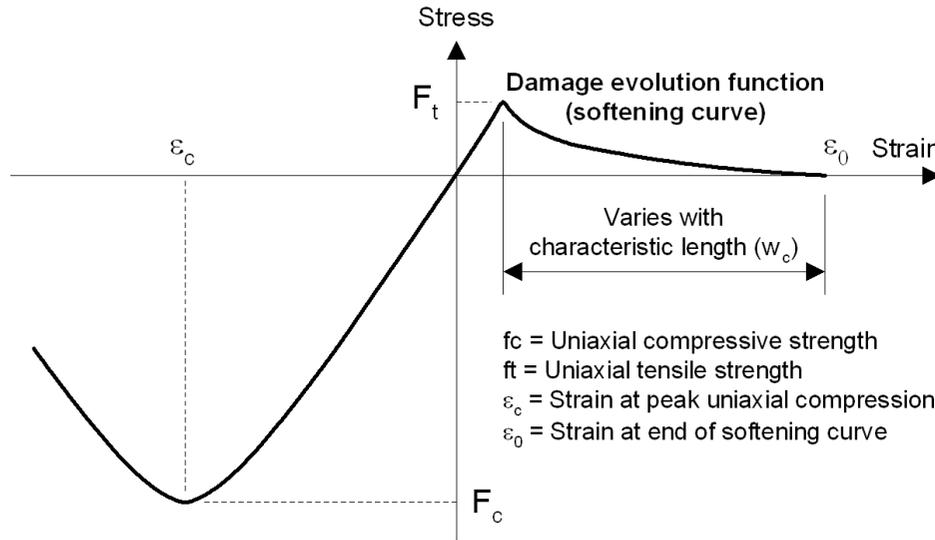
Stress / strain curve for multi crack concrete model (Model 105)

❑ **Specify short and long term viscous/elastic coefficients** This option is only available if Creep or User-defined creep settings have been chosen and requires definition of one or more of the following parameters: **28 day creep factor**; **Proportion short term viscous**; **Proportion long term elastic**.

❑ **Coefficient drying shrinkage** Describe.

❑ **Coefficient chemical shrinkage** Describe.

Note relating to Uniaxial Compresive strength

- 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 concrete properties

These more advanced concrete model 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.

- ❑ **Biaxial to uniaxial stress ratio** (β_r) e.g. 1.15. A range of 1.05 to 1.3 is suggested and a default of 1.15 is provided.

- ❑ **Initial relative position of yield surface** (Hz_o) e.g. 0.6. This parameter 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** (giving plastic potential slope relative to that of yield surface) (ψ). e.g. -0.1The parameter ψ is used to control the degree of dilatancy. Associated plastic flow is achieved if $\psi=1$, 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) e.g. 0.425. This constant 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_o** for the 1st opening stage (m_{hi}) e.g. 0.5. This multiplier 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_o** (m_{ful}) e.g. 5.0. 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** (ratio for local damage surface) ($r\sigma$) e.g. 1.25. 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** (μ) e.g. 1.0. A range of 0.5 to 1.5 is suggested and a default value of 1.0 is provided.

The use of non-default values for these remaining parameters should only be considered with knowledge of the fundamental equations used by the concrete model, and experimental data.

❑ **Crack fixity strain** (default 0.05)

❑ **Fracture process zone width** (e.g. about 3 times aggregate size) (default 0.06)

❑ **Number of iterations** after which cracks are fixed (default 2). 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

❑ **Maximum iterations** for fracture strain update (default 2)

❑ **Residual Softening Factor** (default 0.001)

❑ **Material constant Ap** (default 0.60)

❑ **Material constant An** (default 0.75)

❑ **Contact Sharpness Z1** (default 8.00)

❑ **Contact Sharpness Z2** (default 8.00)

❑ **Limit of Contact Transition** (default 3.00)

❑ **Angular limit between crack planes** (Radians) e.g.1.0 (legacy support for Multi Crack Concrete Model (model 94) only)

❑ **Relaxation multiplier** (default 1.0)

❑ **Creep factor multiplier** (default 1.0)

❑ **Pickett factor multiplier** (default 1.0)

❑ **Degree of percolation** (default 0.05)

❑ **Degree of hydration first cure** (default 0.5)

❑ **Moisture rate Pickett effect** (default 10.0E-12)

❑ **Saturation start Pickett effect** (default 0.85)

❑ **Short relaxation time** (default 844.0E3)

❑ **First long relaxation time** (default 12.96)

❑ **Second long relaxation time** (default 95.04)

❑ **Temperature when degradation starts** (default 40.0)

❑ **Temperature when degradation finishes** (default 80.0)

❑ **Finishing degradation coefficient for E** (default 0.9)

❑ **Finishing degradation coefficient for fc** (default 0.8)

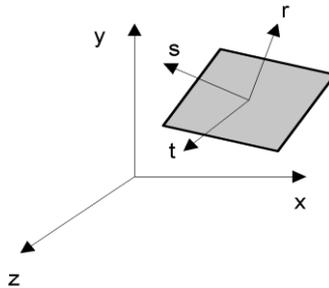❑ **Finishing degradation coefficient for ft** (default 0.9)

Evolution of properties E, FC, Ft

❑ **Default** When selected, set the 'Finishing degradation coefficient' entries for E, Fc and Ft to be 0.9, 0.8 and 0.9 respectively.

❑ **Define power coefficients** When selected, replaces the 'Finishing degradation coefficient' entries for E, FC and Ft with 'Logistic model coefficient for E', 'Power model coefficient for fc', and 'Power model coefficient model ft' entries, with deafults of 12.5, 3.0 and 1.0 respectively
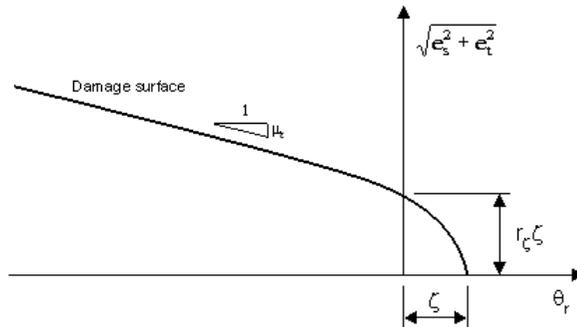
Planes of degradation

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 $r_\sigma = c / f_t$ 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

- LUSAS nonlinear concrete material models 105 is based on the smeared crack concept. This imposes limitations on the maximum size of a mesh element based on material properties elastic modulus, tensile strength, and fracture energy – which should not exceed the equivalent of approximately 0.35m. If it does, the actual value for the fracture energy is increased internally.

- When the purpose of an analysis is to check if cracking would occur and compute the crack widths mesh refinement using a denser mesh, and time step convergence studies using a sufficient number of load steps should be done. Start with small load steps as the initial orientation of cracks tends to propagate, as in real life.

- 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 minimum element length of greater than or equal to the reinforcement cover is suitable for linear elements. A minimum element length of greater than or equal to twice the reinforcement cover is suitable for quadratic elements.

- Care should be taken when choosing an increment size for analysis with multi-crack concrete models as inaccurate results can be obtained if the load

increments are too large. More accurate results are obtained when smaller increment sizes are used.

- 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.

- It is recommended that the following LUSAS options are used with these concrete models:

  252 Suppress pivot warnings.

  62 Allow negative pivots.

  Please contact LUSAS technical support if these options are required.

See the *Solver Reference Manual* and *Theory Manual* for further information relating to concrete material modelling.

Smoothed Multi Crack Concrete Model (Model 109)

The smoothed 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.

Capabilities

- Cracking and crushing 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 using star symbols.

- General crack width calculation is possible and can be plotted for the Contour and Values layers as component of CWMax (Maximum crack width) under the Plastic strain - Plane stress entity. Component EFSMax (Maximum equivalent fracture strain) is also available. Note that these components are only applicable for elements that support the multi-crack concrete model. See Appendix K: Results Tables in the *Element reference Manual* for details. See

also the general notes relating to crack width calculation at the end of this topic.

- When 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.

- LUSAS models that used previous versions of this concrete model will automatically be updated to use this concrete model, and when solved will use defaults to ensure that results obtained will be similar to those previously obtained.

See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Nonlinear analysis of a concrete beam' for a worked example.

Specification of Concrete model, Creep Code and Shrinkage

See **Concrete Models (Nonlinear)** for concrete model, creep and shrinkage options.

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 $E > 1.2\ f_c/\varepsilon_c$
- ❑ **Poisson's ratio**
- ❑ **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.
- ❑ **Coefficient of thermal expansion** (if 'Thermal properties' is selected).
- ❑ **Mass Rayleigh damping constant** (if 'Dynamic properties' is selected).
- ❑ **Stiffness Rayleigh damping constant** (if 'Dynamic properties' is selected).

Plastic Material Properties

These primary material properties apply to both unreinforced (plain concrete) and reinforced concrete except where stated.
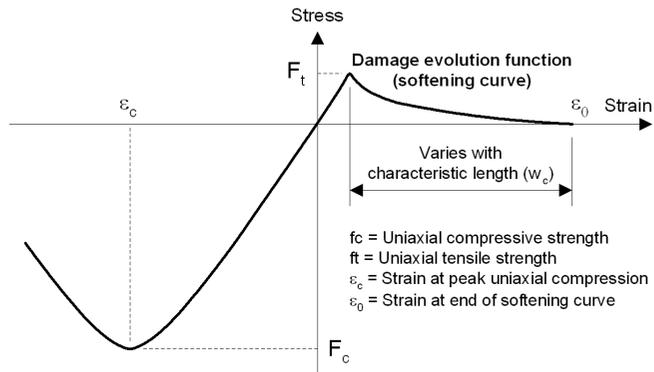
- ❑ **Uniaxial compressive strength** (f_c) e.g. 40 N/mm^2. The generally available "characteristic compressive strength" may not be appropriate for use in a finite element analysis with a concrete material model. This is because cube strengths

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.

❑ **Strain at peak uniaxial compression** (ε_c) e.g. 0.0022. 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, ec. A reasonable check is to ensure that E > 1.2 fc/ ec

Specification of **Fracture energy** or **Strain at end of softening curve** via selection of the appropriate radio button on the dialog is required. Specifying fracture energy is usually the more convenient option:

❑ **Fracture energy per unit area** (G_f) e.g. 0.13N/mm. 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 $G_f = 0.13$N/mm is suggested. Typical values of G_f are: for 16mm aggregate, 0.1 N/mm; for 20 mm aggregate, 0.13 N/mm; and for 32 mm aggregate, 0.16 N/mm.

❑ **Strain at end of softening curve** (ε_o) e.g. 0.0035.In some cases it may be deemed more suitable to specify the strain at end of softening curve. In such a case and where no data is available then a value for ε_0 of 0.0035 is a reasonable first estimate for most concrete. 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.

Stress / strain curve for multi crack concrete model (Model 94)

Note relating to Uniaxial Compresive strength

- 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 concrete properties

These more advanced concrete model 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. The use of non-default values for the remaining parameters should only be done with a knowledge of the fundamental equations used by the concrete model, and experimental data.

- ❑ **Biaxial to uniaxial stress ratio** (β_r) e.g. 1.15. A range of 1.05 to 1.3 is suggested and a default of 1.15 is provided.
- ❑ **Initial relative position of yield surface** (Hz_o) e.g. 0.6. This parameter 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** (giving plastic potential slope relative to that of yield surface) (ψ). e.g. -0.1The parameter ψ is used to control the degree of dilatancy. Associated plastic flow is achieved if $\psi=1$, 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) e.g. 0.425. This constant 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_o** for the 1st opening stage (m_{hi}) e.g. 0.5. This multiplier 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_o** (m_{ful}) e.g. 5.0. 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** (ratio for local damage surface) ($r\sigma$) e.g. 1.25. 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** (μ) e.g. 1.0. A range of 0.5 to 1.5 is suggested and a default value of 1.0 is provided.

❑ **Angular limit between crack planes** (Radians) e.g.1.0 (for Multi Crack Concrete Model (model 94) only)

❑ **Crack fixity strain** (default 0.05)

❑ **Fracture process zone width** (e.g. about 3 times aggregate size) (default 0.06)

❑ **Number of iterations** after which cracks are fixed (default 2). 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
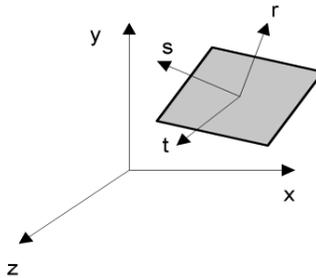
iterations is increased , then the Maximum number of iterations on the Nonlinear and Transient Control dialog should also be increased

❑ **Maximum iterations** for fracture strain update (default 2)

❑ **Residual Softening Factor** (default 0.001)

❑ **Material constant Ap** (default 0.60)

❑ **Material constant An** (default 0.75)

❑ **Contact Sharpness Z1** (default 8.00)

❑ **Contact Sharpness Z2** (default 8.00)
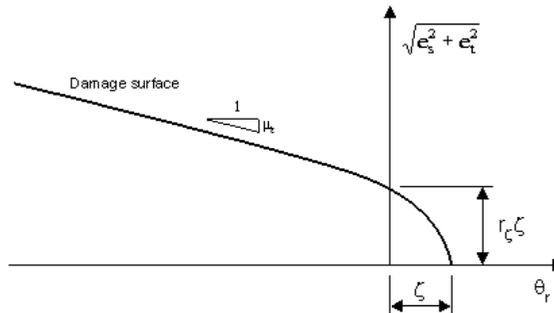
❑ **Limit of Contact Transition** (default 3.00)

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 $r_\sigma \square = c / f_t$ 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

- LUSAS nonlinear concrete material models 105 is based on the smeared crack concept. This imposes limitations on the maximum size of a mesh element based on material properties elastic modulus, tensile strength, and fracture energy – which should not exceed the equivalent of approximately 0.35m. If it does, the actual value for the fracture energy is increased internally.

- When the purpose of an analysis is to check if cracking would occur and compute the crack widths mesh refinement using a denser mesh, and time step convergence studies using a sufficient number of load steps should be done. Start with small load steps as the initial orientation of cracks tends to propagate, as in real life.

- 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 minimum element length of greater than or equal to the reinforcement cover is suitable for linear elements. A minimum element length of greater than or equal to twice the reinforcement cover is suitable for quadratic elements

- Care should be taken when choosing an increment size for analysis with multi-crack concrete models as inaccurate results can be obtained if the load increments are too large. More accurate results are obtained when smaller increment sizes are used.

- 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.

- It is recommended that the following LUSAS options are used with these concrete models:

    252 Suppress pivot warnings.

    62 Allow negative pivots.

    Please contact LUSAS technical support if these options are required.

See the *Solver Reference Manual* and *Theory Manual* for further information relating to concrete material modelling.

Concrete (Reinforced) Material Model

Reinforced concrete material for use with the **RC frame design** facility is defined by selecting the **Attributes > Material > Concrete (Reinforced)** menu item. This material references linear elastic concrete and steel materials provided in the **Material Library**. The entries are filtered to show those that are valid according to the Code of Practice selected.

To carry out reinforced concrete frame design checks, a concrete material and a reinforcement material need to be assigned to the lines representing reinforced concrete members in the model. This cannot be done by defining and assigning separate concrete and steel material attributes (since only one material can be assigned to a line feature within Modeller) so the concrete (reinforced) material model, which provides a compound material attribute, must be defined and assigned instead.

When created, three material entries appear in the Attributes 🐝 Treeview, one for each concrete and steel isotropic material defined, and one compound material entry that references them both. To carry out reinforced concrete frame design checks only the compound material attribute should be assigned to lines in the model representing reinforced concrete members.

Concrete Heat of Hydration

Concrete heat of hydration properties can only be defined for a Thermal or Hygro-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 by selecting the General option in the 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 mass 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:

- ❑ **Mass of cement per unit volume** This should not include the mass of slag or the mass of fly ash and must be defined in kg/m^3)
- ❑ **Water/Cementitious ratio** =(mass per unit volume of cement and any slag and any fly ash)/(mass per unit volume of water)
- ❑ **Mass of slag per unit volume** Mass of ground granulated blast furnace slag per unit volume (must be defined in kg/m^3)
- ❑ **Mass of fly ash per unit volume** Also known as flue ash (must be defined in kg/m^3)
- ❑ **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%).
- ❑ **Delay for heat of hydration** A (negative) factor that can be specified to delay the onset of heat of hydration to get a best fit of temperature curve data with experimental data.

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.

Worked example

See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Staged Construction of a Concrete Dam', for a typical use of this facility.

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.
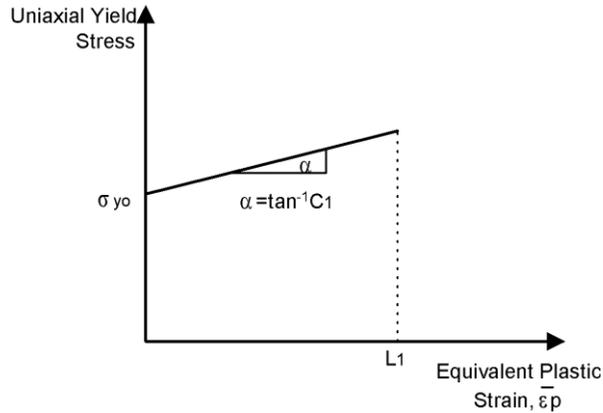
Hardening Curve Definition for the Tresca Yield Model

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, and selecting the Plastic option.

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, σzz, 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.

- The stresses specified in each direction are related in defining the yield surface. Randomly specified values can lead to convergence difficulties. As a guide, to avoid a concave yield surface, stresses should comply with the following relationship:

$$\sigma_{ii} > \frac{\sigma_{jj}\sigma_{kk}}{\sigma_{jj}+\sigma_{kk}} \quad i \neq j \neq k, \ i, j, k = x, y, z$$

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:

- ❑ **Total strain** Uniaxial yield stress v Total strain
- ❑ **Plastic strain** Uniaxial yield stress v Effective plastic strain
- ❑ **Hardening gradient** Hardening gradient (slope) v Effective plastic strain

Hardening curves are defined as follows

- ❑ **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.
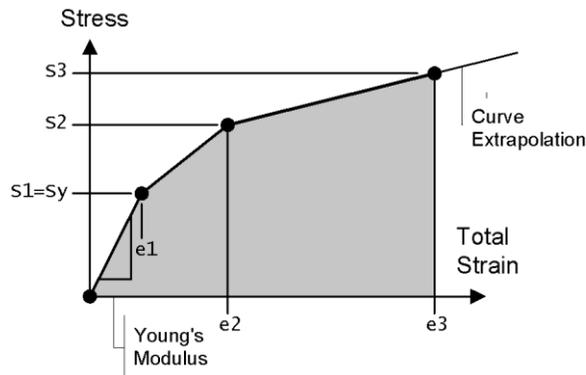
- ❑ **Uniaxial yield stress vs. Effective plastic strain** Requires input of coordinates at the ends of straight line approximations to the uniaxial yield stress vs. effective plastic strain curve.

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.
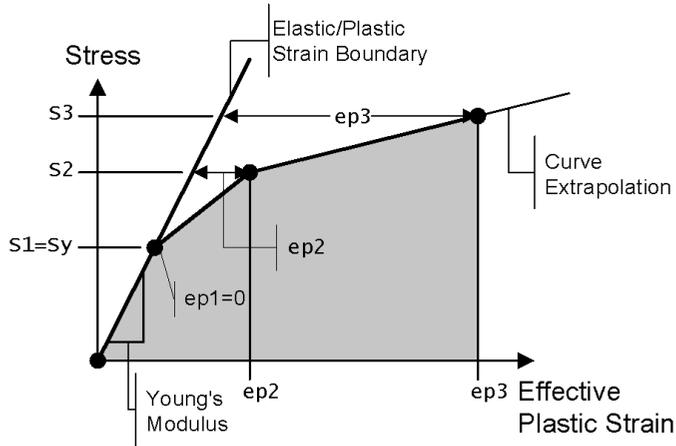
❑ **Hardening gradient (Slope) vs. Effective plastic strain** Requires specification of gradient and limiting strain values for successive straight line approximations to the stress vs. effective plastic strain curve.

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.
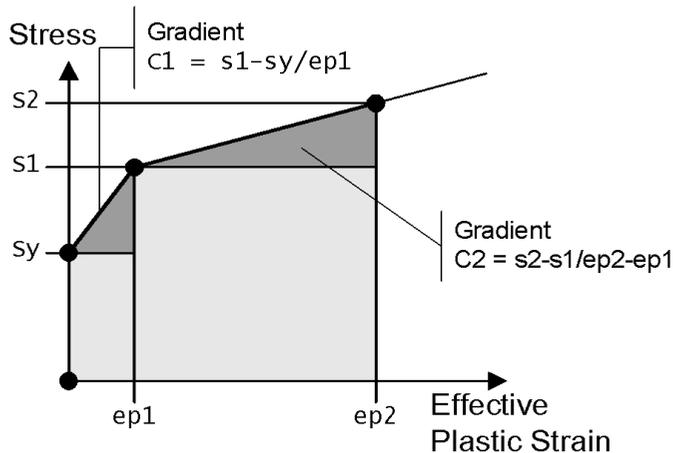
Optimised von Mises (Model 75)

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

❑ **Initial unixial yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.

❑ **Heat fraction coefficient** 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.
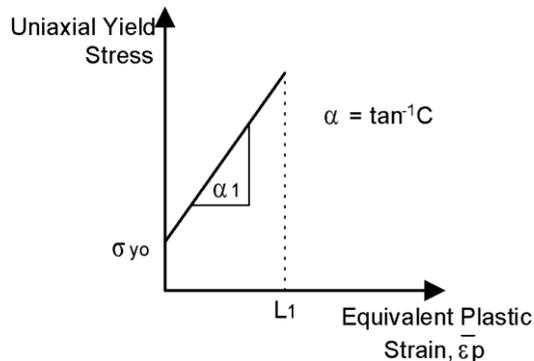
Nonlinear Hardening Curve for the von Mises Yield Model (Model 75)

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 dilettante frictional materials that exhibit increasing shear strength with increasing confining stress (for example, granular soils

or rocks). The model incorporates isotropic hardening and dilatancy, and only allows tension if cohesion is included.

A **Modified Mohr Coulomb** model is also available which provides tensile and/or a compressive cut-off limits to prevent unrealistic stresses from developing and to improve solution convergence.

Material Properties

- ❑ **Initial Cohesion** [Ci] defining the degree of granular bond and a measure of the shear strength.
- ❑ **Initial and Final Friction Angle** [ϕi and ϕf]defining the angles of shearing resistance
- ❑ **Dilation Angle** defining the magnitude of plastic volume strains.

Hardening input

- ❑ **Cohesion hardening parameter** [h_c]
- ❑ **Limiting equivalent plastic strain** [ε_f]
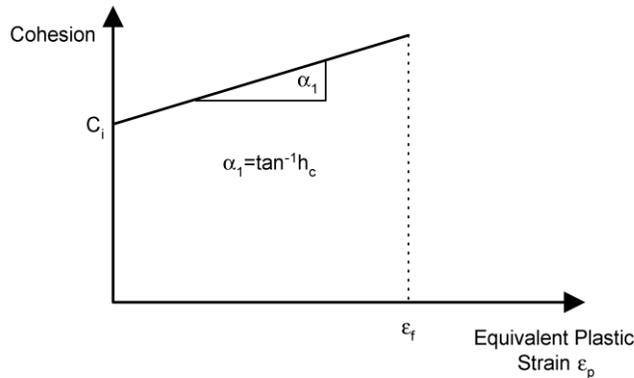
Cohesion Definition for the Non-Associated Mohr-Coulomb Model (Model 65)
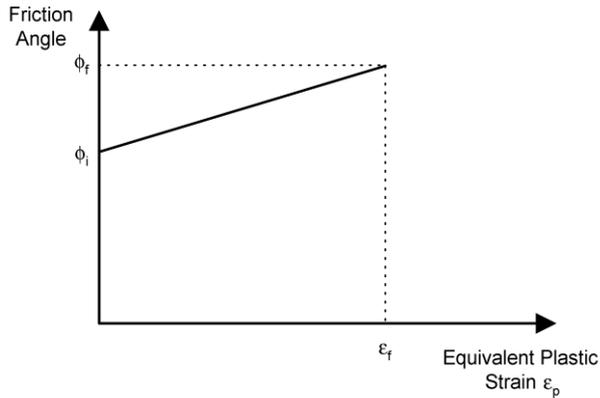
Friction Angle Definition for the Non-Associated Mohr-Coulomb Model (Model 65)

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.

Differences between the Non Associated and Modified Mohr Coulomb models

In the Non Associated Mohr-Coulomb model, the yield surface goes to a singularity in the tensile direction. As a result, at "very low" confining stresses, convergence problems may be met. With the Modified Mohr-Coulomb model the tension cut-off removes the singularity with the intention of improving the numerical solution and making convergence easier.

In the Non Associated Mohr-Coulomb model, the yield surface becomes larger indefinitely in the compression direction, inferring that for any value of confining stress, the soil is getting stronger. With the Modified Mohr-Coulomb model the compressive cut-off prevents the strength in compression increasing indefinitely; it reaches the specified value and gets no stronger.

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** [C_0] 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** [f_0] A material property of granular materials, such as cohesive soils and rocks.

❑ **Heat fraction cofficient** [h_f] 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 values

❑ **Slope of cohesion** The slope of the cohesion against the equivalent plastic strain.

❑ **Slope of friction angle** The slope of the friction against the equivalent plastic strain.

❑ **Plastic strain** The limit of equivalent plastic strain up to which the hardening curve is valid.
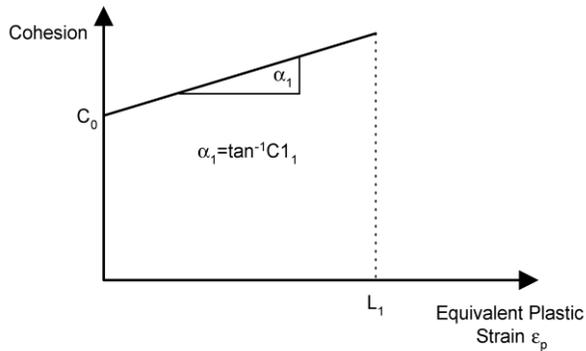
Cohesion Definition for the Drucker-Prager Yield Model (Model 64)
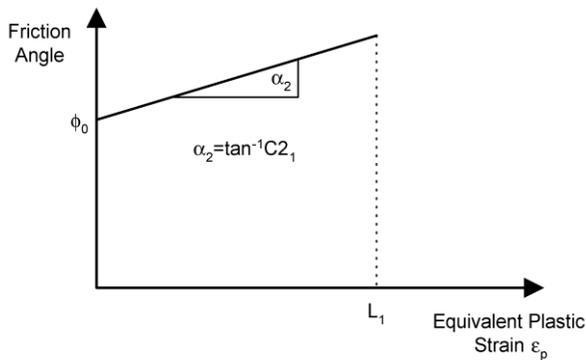
Friction Angle Definition for the Drucker-Prager Yield Model (Model 64)

See the*Solver Reference Manual* for further details.

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, with a time hardening form 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 fully or partially saturated 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 soil in either a fully or partially saturated state. LUSAS can model the deformation of fully or partially saturated porous media during the consolidation process, under isothermal conditions and small strain assumptions, as well as in the limiting drained and undrained conditions. 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:

❑ **Fully saturated** - In addition to elastic properties, it is necessary to specify the Bulk Modulus values for solid and fluid phases, the porosity of the medium, the unit weight of fluid, hydraulic conductivity values of the medium in X, Y and Z directions, and the fluid density. It is possible to specify the solid phase of the material to be "incompressible", by selecting the 'Incompressible solids' option, which negates the need to input the corresponding bulk modulus.

❑ **Partially saturated** - requires additional saturation properties, namely 'Irreducible saturation' and 'Degree of saturation to be considered fully saturated' to be specified. It is also required to specify a draining and filling (absorption) curve using the "Define curves" button (although it is possible to define only the draining curve and have the filling curve assumed identical).

❑ **Piecewise linear partially saturated** - requires the same parameters as for partially saturated material, above, with the draining and filling curve defined in a piecewise fashion and a 'Curve tolerance' (used to judge whether the point to be interpolated is on a segment) needs to be specified.

❑ The **Define curves** button is only enabled for partially saturated materials and allows the draining and filling curves to be defined.

❑ **Incompressible solids** Selection of this option excludes the Bulk modulus of solid phase entry from the data input table.

See the *Solver Reference Manual* for details of all input parameters.

Draining and filling curves

Draining and filling curves must be specified for partially saturated materials. This may be by defining the rate of water extraction, weight factor, air entry and permeability values or manually by selecting "piecewise linear" and specifying pore pressure. 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.
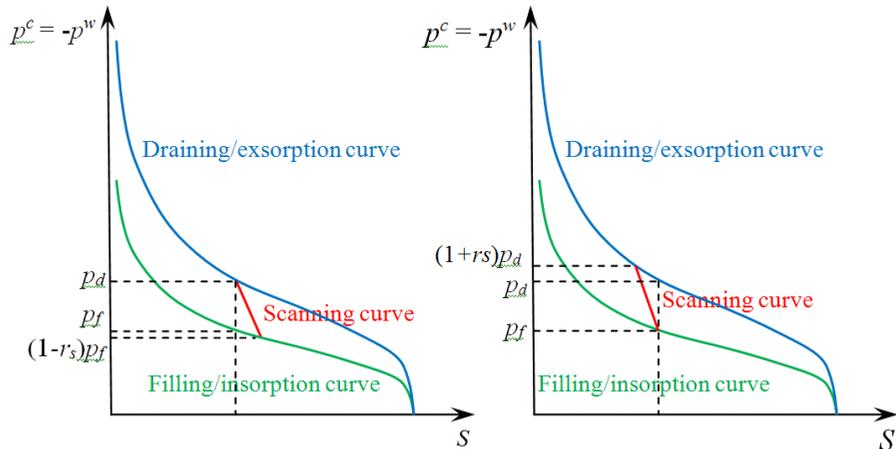
Typical draining and filling curve showing jump from draining/exsorption to filling/insorption (left) and jump from filling/insorption to draining/exsorption (right)

Notes

- For geotechnical analyses that do not include a two phase material definition, the mass density (specified as part of the linear elastic properties) must relate to the total mass of the soil, including the mass of any water content in the voids. The soil may be saturated, partially saturated or dry. For geotechnical analyses that include a two phase material definition, the mass density must relate to the soil grains only.

- Hydraulic Conductivity is often referred to as the Coefficient of Permeability.

- 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.

K_0 Initialisation

K_0 initialisation properties allow initial stresses in soil to be calculated. K_0 data can be defined for material models used in soil analyses, principally, elastic, **Modified Mohr-Coulomb**, **Modified Cam-clay** and **Duncan-Chang** material models, via dedicated pages on those material properties dialogs. The Modified Cam-clay and Duncan-Chang models both need the definition of an initial stress state to calculate an initial stiffness.

❑ **Coefficient of lateral earth pressure**, K_0, relates vertical stresses, σ_v, in the soil to horizontal stresses, σ_H, via the relationship $\sigma_H = K_0 \sigma_v$ where it is assumed that shear stresses are negligible. On element activation, the effective vertical stresses in the soil are calculated from the soil weight and any additional loads such as overburden. Horizontal stresses are then calculated using the K_0 value. As long as strict K_0 conditions of horizontal soil layers and phreatic surface are maintained, the generated stress field will be in balance with the loads. In the case that this is not true, further iterations will be required to bring the soil into equilibrium. K_0 can be greater than 1.0 during the ground initialisation phase at the start of the analysis but must be less than 1.0 when applied to elements which are activated at later stages.

K_0 initialisation of Modified Cam-clay

The Modified Cam-clay model requires the input of two state variables. The pre-consolidation pressure, P_c, determines the size of the yield surface and the initial void ratio, e_o, the amount of compression the soil has undergone. These parameters are input in the Cam clay material parameters and, using material spatial variations, may be easily varied with depth.

Alternatively, they can be calculated during the K_0 initialisation. For a normally consolidated clay, the **coefficient of lateral earth pressure**, K_0 needs to be defined. Once, the stresses are calculated, the pre-consolidation pressure is known as the soil lies on the yield surface. If the **specific volume of the soil of the Normal Consolidation Line**, N, at a pressure of 1.0 is provided, the specific volume on the stable state boundary surface for the stress state can be calculated during K0 initialisation and hence the value of the initial void ratio, e_o. In the absence of N, the value from the material property input will be used.

For over-consolidated soils the value of K_0 can be calculated using the equation:

$$K_0 = ocr.K_{nc} - \frac{\nu_{ul}}{1 - \nu_{ul}}(ocr - 1)$$

where ocr is the **over-consolidation ratio** (ocr=σ_{vmax}/σ_v where $\sigma_v$max is maximum historic vertical stress applied to the soil), K_{nc} the coefficient of lateral earth pressure for normal consolidation (i.e. K_0 for a normally consolidated soil) and ν_{ul} is the **Poisson's ratio of the soil during unloading.**

Material spatial variations can be applied to the K_0 parameters and to the ocr in particular. Close to the surface the ocr changes very rapidly, going to infinity, when the vertical stress goes to zero. In this case a much more reduced value of ocr must be input to avoid problems with its interpolation at the element Gauss points.

Usage

The initialisation of K_0 ground conditions requires a first loading increment in which just the soil loads and groundwater conditions are applied. For K_0 conditions to be valid, both the soil layers and the phreatic surface must be horizontal. Also, the vertical boundaries at the edges of the model must allow free movement in the vertical direction but be restrained in the horizontal direction. If these conditions are strictly met the internal stress field applied to the soil will exactly balance the loads and the soil will not displace.

Delamination Interface Material (Model 25)

The delamination interface material model can be specified by selecting the **Attributes> Material> Specialised> Interface** 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

❑ For a model with a 2D analysis category two fracture modes are present

❑ For a model with a 3D analysis category three fracture modes are present

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.

Elasto-Plastic Interface (Model 27, 26)

Elasto-plastic interface models can be specified by selecting the **Attributes> Material> Specialised> Interface** menu item.

Elasto-plastic interface models are used to represent the friction-contact relationship within planes of weakness between two discrete bodies.

- ❑ For a model with a 2D analysis category 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.
- ❑ For a model with a 3D analysis category 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 Elasto-Plastic Interface 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 Elasto-Plastic Interface 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

Mohr Coulomb Friction Interface Material

Mohr-Coulomb friction interface material can be specified by selecting the **Attributes> Material> Specialised> Interface** menu item.

Whilst 'elasto-plastic interface material' (Model 26 for volumes, Model 27 for plane stress and plane strain) may be used to model the soil-structure interface, as it has a limited tension criterion normal to the interface plane, and a Mohr-Coulomb criterion tangential to the interface plane, and it must be assigned to continuum elements. This gives the failure plane an appreciable width, while failure at a soil-structure interface is observed to occur in a very localised shear band only a few grains in thickness.

The Mohr-Coulomb interface material can be used in conjunction with 2D and 3D interface elements. It permits the modelling of a standard elastic-plastic sliding interface as well as strain hardening, softening and dilation. The classic Mohr-Coulomb relationship is used to define failure conditions on the interface, and the slip at the interface is used to control the rate of dilation as well deterioration in the angle of friction. Cohesion can also be linked to the slip. It can also be used in a two-phase analysis.

It is assumed that the resistance of the interface to movement in any direction is a function of the amount of slippage which has occurred but that it is not dependent on the particular path which may have been taken to get to the current point.

Constant values

A friction interface can be defined by specifying constant values for either:

❑ **Tangential slip** This requires the **Angle of friction**, **Dilatancy angle**, and **Cohesion** to be specified

❑ **Tangential displacement** This requires **Normal compressive pressure**, **Dilatancy angle**, and **Cohesion** to be specified.

For either option a cut-off tension limit can be specified by selecting **Specify cut-off tension limit**.

Variation of values using piecewise linear curves

Alternatively, piecewise linear curves may be defined for each of the parameters required by the Tangential slip or Tangential displacement options. Note that defining a piecewise line curve for a required parameter removes the corresponding entry from the Input grid.

For Tangential slip:

- ❑ **Angle of friction curve** Specify values of Tangential slip against stress.
- ❑ **Dilatancy angle curve** Specify values of Tangential slip against dilatency.
- ❑ **Cohesion curve** Specify values of Tangential slip against cohesion.

For Tangential displacement:

- ❑ **Stress/strain curve** Specify values of Tangential displacement against stress.
- ❑ **Dilatancy angle curve** Specify values of Tangential displacement against dilatency.
- ❑ **Cohesion curve** Specify values of Tangential displacement against cohesion.

Two phase material properties

- ❑ **Hydraulic conductivity** Once basic material properties have been entered, a value for hydraulic conductivity can be entered by pressing the Two phase radio button. This option controls the flow of fluid across an interface element. A value of zero closes the interface. An interface element separating two two-phase elements connects the pore-water pressures on opposite nodes. A high value of κ compared to the hydraulic conductivity in the elements will limit the affect of the interface on the fluid following through it. The hydraulic conductivity may be changed at any point in the analysis by assigning new properties to the interface element.

Advanced

For Tangential slip:

- ❑ **Normal stiffness factor** The normal stiffness factor specified should be set to be between 10 and 100 to reduce penetration normal to the surface.

For Tangential displacement:

- ❑ **Normal stiffness factor** The normal stiffness factor specified should be set to be between 10 and 100 to reduce penetration normal to the surface.
- ❑ **Tangential stiffness factor** For soils, values of tangential stiffness should be set to be between 0.1 and 0,.8.
- ❑ **Update calculated stiffness** The calculated stiffness can be updated either Never, Iteratively (at the end of every iteration, or Incrementally (at the end of every increment or loadstep.

For more details see *Mohr-Coulomb Interface Model* in the *Solver Reference Manual*.

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.
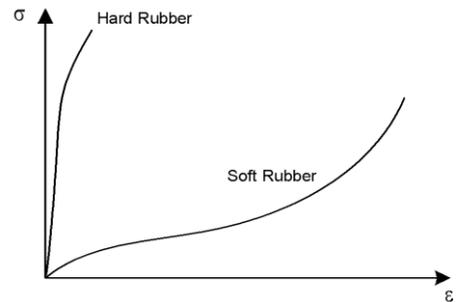
Rubber materials are also practically incompressible, that is, they retain their original volume under deformation. This is equivalent to specifying a Poisson's ratio approaching 0.5.

The strain measure used in LUSAS to model rubber deformation is termed a **stretch** and is measured in general terms as:

λ = dnew/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.
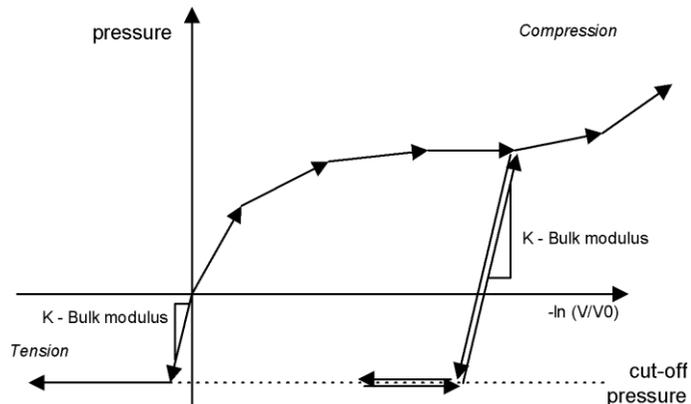
Pressure - Logarithm of Relative Volume Curve

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 = (syld2)/3, then classical von Mises yield criterion is obtained.

Yield Surface Representation For Different Pressure Dependent Yield Stress Values
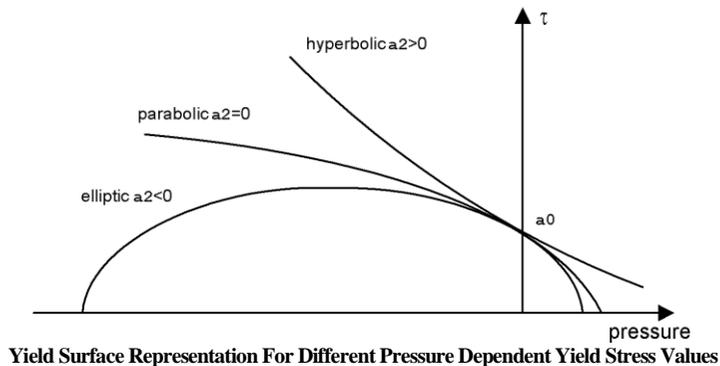
Notes

• **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: $K = \dfrac{E}{3(1 - 2v)}$

• **Shear modulus** The relationship between the elastic modulus values in shear, G, and tension, E, assuming small strain conditions, is given by: $G = \dfrac{E}{2(1 + v)}$

- **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 (or an undrained analysis), in which two-phase elements are used to define soil in either a fully or partially saturated state.

K_0 initialisation of Modified Cam-clay

K_0 **initialisation properties** allow initial stresses in soil to be calculated.

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.

Duncan-Chang Model

The Duncan-Chang model is defined using the **Attributes> Material> Specialised** menu.

Duncan-Chang is a simple soil model whose parameters are calculated by curve fitting triaxial test data over a range of cell pressures, σ_3. It takes an incremental elastic approach in which Young's modulus and Poisson's ratio are evaluated from current stresses. A predictor step is used to determine whether the soil is loading or unloading, following which, corresponding values of Young's modulus and Poisson's ratio are calculated for use in the incremental elastic modulus matrix.

The model can be used with standard continuum elements as well as the two-phase elements. The modelling parameters required can be evaluated in standard laboratory tests.

Material Properties

The material properties to be defined vary according to the Poisson's ratio model used.

The implementation allows for three descriptions of Poisson's ratio, ν:

- ❑ **Constant**
- ❑ **Direct (E-ν) method**
- ❑ **Indirect (E–b) method**

These material properties are listed for a Constant Poisson's ratio model unless otherwise noted.

- ❑ **Cohesion** - the void ratio of the soil at the start of the analysis
- ❑ **Friction angle** in degrees
- ❑ **Change of angle of friction** - with respect to cell pressure (in degrees)
- ❑ **Failure ratio**
- ❑ **Density**
- ❑ **Atmospheric pressure**
- ❑ **Initial Young's modulus number**
- ❑ **Young's modulus number**
- ❑ **Initial Young's modulus exponent**
- ❑ **Bulk modulus number** (Indirect (E-b) method
- ❑ **Bulk modulus exponent** (Indirect (E-b) method
- ❑ **Poisson's ratio**
- ❑ **Poisson's ratio tangent** (Direct (E-v) method
- ❑ **Cell pressure moderation component** (Direct (E-v) method
- ❑ **Poisson's ratio at atmospheric pressure** (Direct (E-v) method

❑ **Young's modulus when soil fails** (Indirect (E-b) method

❑ **Poisson's ratio when soil fails** (Indirect (E-b) method

For further information on these parameters. *See Solver Reference Manual : Duncan-Chang model*

Notes

- The initial stress state can be input by using residual stress load types, by using an initial stress with gravity or by defining K_0 properties from the K_0 tab.

- If used with two-phase properties, the density property refers to the density of the dry soil particles rather than that of the soil mass including water content.

- This model provides a very simple description of soil behaviour. The incremental linear elastic formulation, in the absence of nonlinearities, will always provide a solution, but this solution may not be valid. This model is best used in situations for which it has been previously demonstrated as providing acceptable answers.

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.

❑ **Mass Rayleigh damping coefficient**
❑ **Stiffness Rayleigh damping constant**
❑ **Coefficient of thermal expansion**

Two-Phase Properties

Two-phase material properties are required when performing a consolidation analysis or groundwater flow problem in which two-phase elements are used to define soil in either a fully or partially saturated state.

K_0 initialisation Properties

K_0 **initialisation properties** allow initial stresses in soil to be calculated.

Varying Attributes

Material properties varying with depth can be used with this model. 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.

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 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.
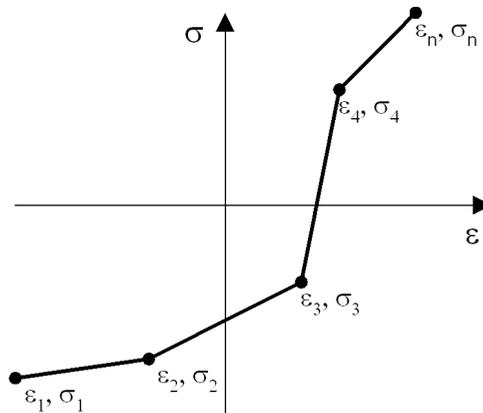
General piecewise linear definition

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.
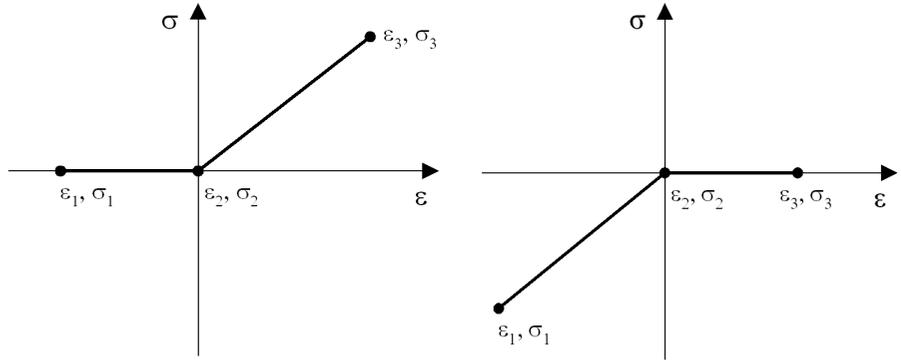
Tension-only bar material   **Compression-only bar material**

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 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.

Joint Material Models

Joint material properties can be defined by selecting the **Attributes > Material > Joint** menu item. 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.

- ❑ **Frictional Contact** with an initial gap. Friction contact joints can be specified to permit lift-off in a user-defined direction.
- ❑ **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, Elastic** 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.
- ❑ **Piecewise Linear, Hysteretic** A range of inelastic hysteretic joint models for basic and complex hysteresis.
- ❑ **Nonlinear user-defined** A user-defined joint model. This allows user-supplied joint property subroutines to be used from within LUSAS. This facility provides controlled access to the pre- and post-solution constitutive processing and nonlinear state variable output via these user-supplied subroutines.
- ❑ **Matrix properties** Definition of symmetrical matrix properties (stiffness, mass or damping) to specify a joint representing a pilecap sitting on a group of piles in a global (3D) 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.

General settings required for joints

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. Refer to the *Solver Reference Manual* for a full description of the joint material input parameters required for these joint models.

- ❑ **Assignment to** defines / shows the feature(s) to which the joint material has been defined. Options (according to joint type selected_ are 'Points', 'Point and line ends', 'Lines', and 'Surfaces'. Once assigned to a feature type, the joint cannot be assigned to different feature types.
- ❑ **Joint type** selection of a joint type adds the properties that need to be defined for that joint type to the tabular grid on the dialog.
- ❑ **Mass position** (when required) options are used to specify that the joint mass will be 'Between nodes', or 'At first node', or 'At second node'

Additional **Thermal expansion**, **Varying friction coefficient**, **Damping**, **Coupled**, and **Compound hysteresis** options are selectable according to joint type.

Note that selection of one of the 'Assignment to' options controls the expected units for each degree of freedom as displayed in the tool tip when the cursor is hovered over an input cell. When defining a new joint material 'Point' or 'Point and line ends' is the default, again according to joint type. When pre-version 15 models are being edited no selection is automatically made and for these models there is no requirement to make a selection.

Properties for each degree of freedom / common to all freedoms

Properties for each degree of freedom must be specified in the tabular grid. Some joint materials have properties that can be entered to be common for all freedoms.

Usage

See **Joint and Interface Elements** for general information about using joints.

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.

Elasto-plastic joint model force response curves

Elasto-plastic
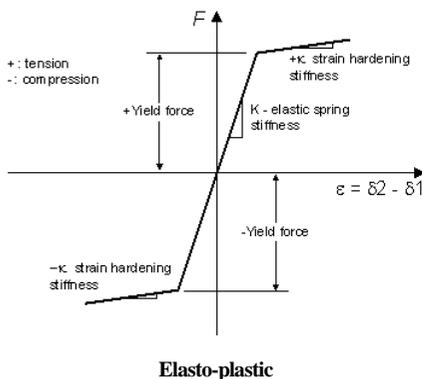
Contact joint model force response curves
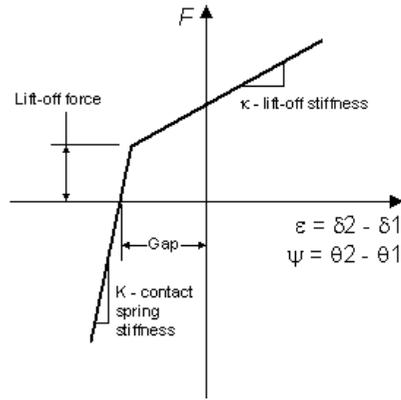
Smooth contact
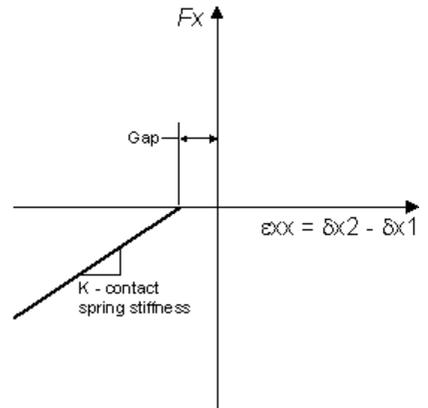
Frictional contact 1
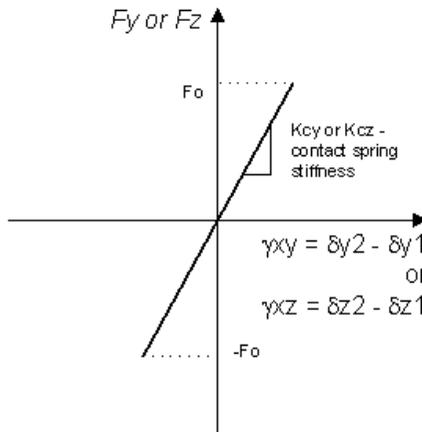
Frictional contact 2
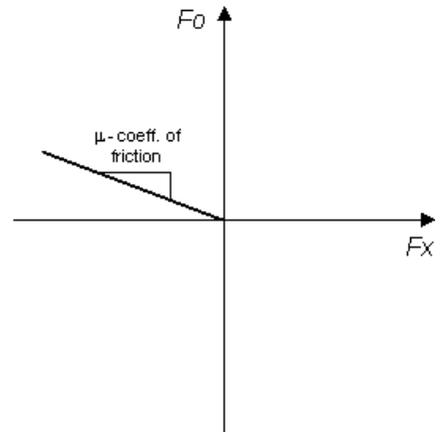
Frictional contact 3

Notes relating to Contact and Frictional Joints

- Initial gaps are measured in units of length for translational freedoms and in radians for rotational freedoms

- For the Smooth Contact model, 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.

- Friction contact joints can be specified to permit lift-off in a user-defined direction.

- Both Smooth Contact and Frictional Contact joints can be used for lift-off or hook contact by using appropriate stiffnesses, gap and yield force.

Seismic isolator joint model force response curves
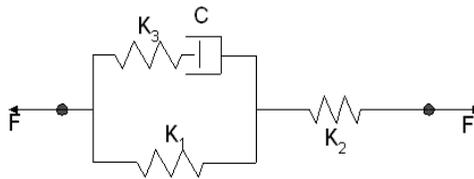
Visco-elastic damper
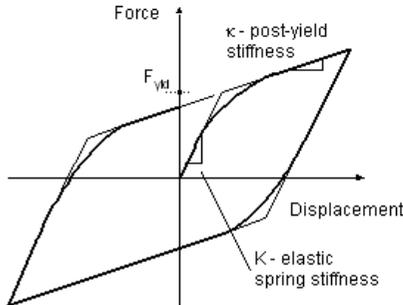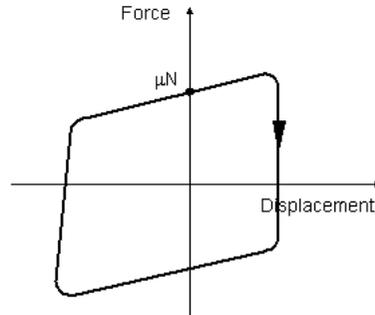
Lead Rubber Bearing   **Friction Pendulum System**

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.

- ❑ **Analysis category** Shows the **analysis category** in use.
- ❑ **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 <u>local y axis</u> must be aligned to the vertical axis. For 3D applications the joint element <u>local z axis</u> must be aligned with the vertical axis. **Local coordinate systems** may be required to achieve correct element orientation.
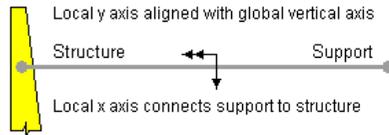
Joint element axes required for 2D modelling applications
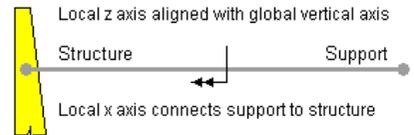
Joint element axes required for 3D modelling applications

- ❑ **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 of the trilinear Earth Pressure dialog. Diagrams and equations (as shown below) are provided also to clarify the meaning of the various material parameters that must be defined.
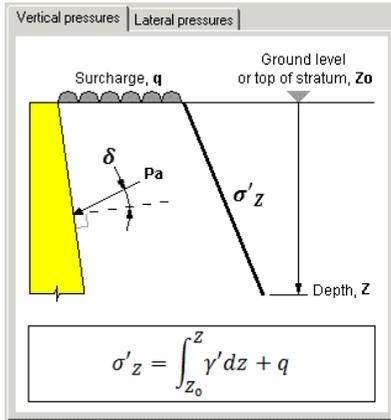
Vertical pressures
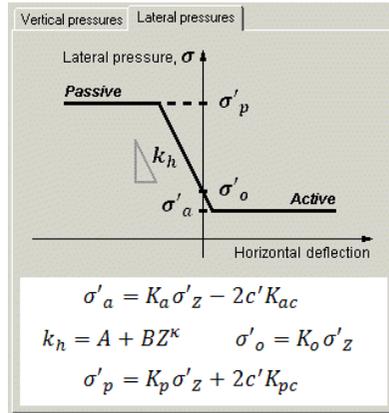
Lateral pressures

Modulus of subgrade reaction

The constant of proportionality, known as the Modulus of Subgrade Reaction, is a measure of the soil stiffness as defined in the lateral pressures image on the dialog, where it is represented with a single slope labelled k_h. It is generally conceded that for over-consolidated cohesive soils, k_h remains approximately constant with depth. In granular soils and also some normally-consolidated clays, k_h is usually assumed to increase linearly with depth. Whilst a number of methods for obtaining suitable values of k_h are available, including field and triaxial tests which can provide results which may be converted into an appropriate value, in some circumstances it may be expressed in polynomial form as $k_h = A + Bz^\kappa$ where A, B and κ are empirical constants. Usually a safe design may be obtained with a low k_h value and a high value of K_o. However, as it may be difficult to obtain reliable values for either, it is generally recommended that the sensitivity of the design is checked with regard to variations of these parameters.

$\kappa = 0$ for cohesive soil under moderate loads

$\kappa = 0.5$ for medium cohesive soil and non-cohesive soil above water table

$\kappa = 1.0$ for non-cohesive soil below the groundwater level or under greater loads

$\kappa = 1.5$-2.0 for loose non-cohesive soil under very high loads
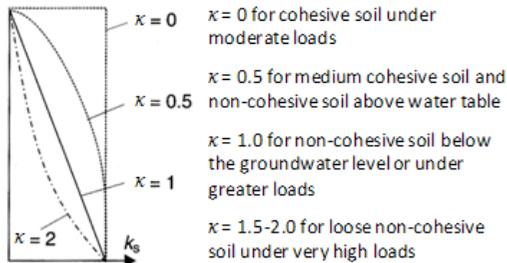
Typical variations of modulus of subgrade reaction with depth

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)**.

Other options

❑ **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.

❑ **Non recoverable** When ticked, displacements which occur while the joint is in a fully passive or fully active state will not be recovered. Instead, if displacement is reversed, then earth pressures will change according to the Franchin model i.e. parallel to the initial stiffness. This (Franchin) model is implemented using the 'parallel reversal' hysteresis implementation of the piecewise linear joint with the 'Initial stiffness' unloading rule and a flat (zero) stiffness beyond the active and passive limits. When unticked, displacements which occur while the joint is in a fully passive or fully active state must be recovered, and earth pressures are modified according to the initial stiffness, passing through the zero displacement/Ko origin, i.e. the joint is effectively nonlinear-elastic (recoverable).

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, if a different soil weight is used for each layer the overburden pressure from any soil layers above must be calculated manually and entered as a surcharge. Where all layers have the same unit weight the datum for each layer can be the same (i.e ground level) remembering that any additional surcharge (q) at ground level needs to be included in the definition of all lower layers.

- 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).

Worked example

- See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > ' Embedded Retaining Wall'.

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, Elastic Joint Material

Piecewise Linear, Elastic Joint Material is defined using the **Attributes> Material> Joint** menu.

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. 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.
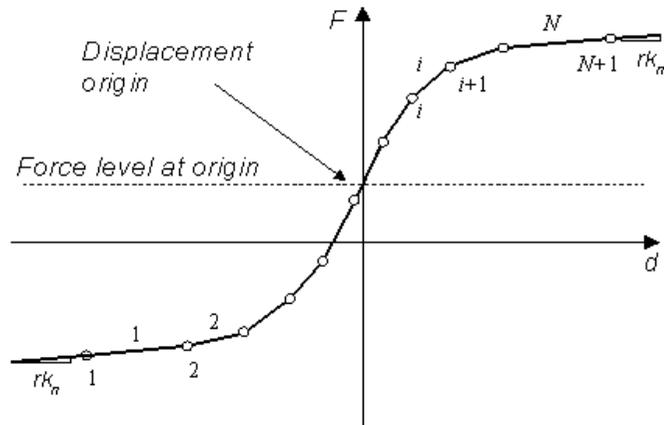
Piecewise linear joint material definition

General settings

For details of general settings to be made see **Joint Material Models**.

Piecewise linear settings

❑ **Coupled** When selected, defines the force / deformation relationship for all degrees of freedom with respect to the deformation in the axial or normal direction. When unset, the relationship in each degree of freedom is defined independently.

Properties for each degree of freedom / common to all freedoms

Properties for each degree of freedom must be specified in the tabular grid.

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.

- Variations can also be used in the definition of curves.

Piecewise Linear, Elastic (Axial Force Dependent) Joint Material

Piecewise Linear, Elastic (Axial Force Dependent) Joint Material is defined using the **Attributes> Material> Joint** menu.

This joint material is similar to the generic 'Piecewise Linear, Elastic' joint material model, but allows any number of Force/Displacement curves to be optionally specified by defining a number of linear curve segments for each curve for a stated axial force. It can be used when inelastic material behaviour is chosen to be modelled using an elastic material model and an associated force/displacement curve.

A typical set of piecewise linear, elastic (axial force dependent) load-deflection curves with N linear segments is shown below.
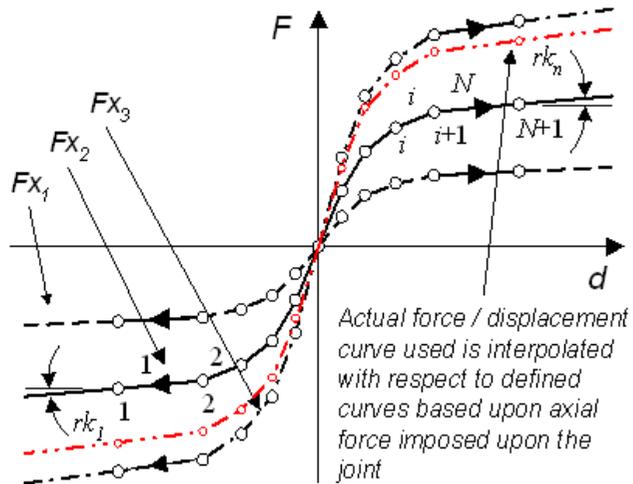
Piecewise linear, elastic (axial force dependent) load-deflection curves

If multiple user-defined curves are specified in LUSAS Modeller, these curves are automatically interpolated by LUSAS Solver when a solve is requested to arrive at suitable force/displacement values for a particular axial force imposed upon a joint to which the joint material has been assigned. The bounding piecewise linear curve with the lowest or highest axial force will be used respectively if the axial force in the joint is smaller or larger than the bounding values. If only one curve for a particular axial force is specified, this curve will be used for all axial forces. A stiffness factor governs how force/displacement values on a curve 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

- When defining a set of piecewise linear (axial force dependent) load-deflection curves each curve must contain the same number of points (and hence curve segments).

- A force/displacement curve can be defined for an axial component (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 curve that matches the axial force in the joint 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 have non-unique negative sides. 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, Hysteretic Joint Material

The Piecewise Linear, Hysteresis, and Compound Hysteresis Joint Models are generic extensions of the 'Piecewise Linear, Elastic' joint material model within LUSAS, and generalisations of well-known named hysteresis models that are in use in industry.

The primary backbone, or envelope, or skeleton under monotonic loading is a piecewise linear curve as used by the Piecewise Linear, Elastic joint material model, where the whole loop for all multi-linear hysteresis joint models can be above or below the zero force axis, and the origin of the loop may have any displacement and force, not necessarily zero displacement and force.

A typical piecewise linear, hysteretic load-deflection curve with N linear segments is shown below.
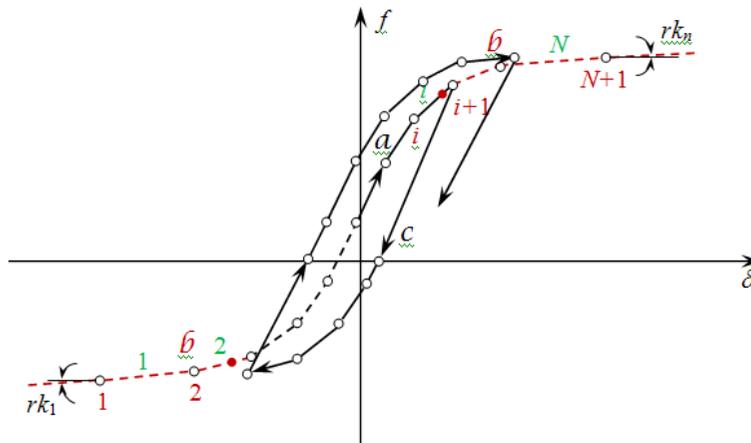
Piecewise linear curve with hysteresis behaviour.

Hysteretic model types supported

Hysteretic models commonly used in industry can be defined by specifying a LUSAS hysteresis model type and a suitable unloading rule(s).

Industry models supported include:

- Degrading bi-
  linear
- Bi-linear with
  slackness
- Origin-centred bi-
  linear
- Peak oriented

- Kivell degrading
  stiffness
- Pivotal plastic model
- Modified Takeda
- Muto degrading tri-
  linear

- Fukada degrading tri-
  linear
- Sina degrading
  trilinear
- Stewart degrading
  stiffness
- Franchin

It should be noted that, where industry-named models consist of a fixed number of piecewise linear segments to describe the primary backbone of the curve, the implementation of the named hystereris models within LUSAS allows for multiple segments to be specified (if required) for each required line segment of a named model. In addition, various unloading rules can be chosen for a single hysteresis model, and any two basic hysteresis models may be combined with different loading and unloading rules to create compound hysteresis loops. This provides many additional options and permutations for assessing a wide variety of hysteretic joints.

General joint settings

The Piecewise Linear, Hysteretic joint dialog is similar to the Piecewise Linear, Elastic dialog, but with additional controls to specify the hysteresis loops. For details of general settings that can be made see **Joint Material Models**.

Hysteresis settings

For piecewise linear (hysteretic) joints, by default, basis hysteresis settings and curve data for at least one hysteresis loop need to be defined.

❑ **Compound hysteresis** When selected, allows hysteresis settings and curve data for a second hysteresis loop to be defined.

❑ **Coupled** When selected, defines the force / deformation relationship for all degrees of freedom with respect to the deformation in the axial or normal direction. When unset, the relationship in each degree of freedom is defined independently.

Hystereris models

The generic hysteresis models implemented in LUSAS are:

❑ **Parallel reversal loading**
❑ **Kinematic-hardening**
❑ **Kinematic-hardening with slackness**
❑ **Origin centred**
❑ **Peak oriented**
❑ **Kivell**
❑ **Pinching point**

❑ **Modified Takeda**

❑ **Muto outer**

❑ **Fukada outer**

❑ **Sina outer**

❑ **Stewart**

❑ **Franchin** - available by selecting Parallel reversal loading with an initial stiffness unloading rule. See table below.

See **Hysteretic Joint Model Force Response Curves** for loading and unloading forms of these models.

See *LUSAS Theory Manual 1* for more details on these models.

Unloading rules

For the 'Origin centred', 'Peak oriented' and 'Stewart' hystereris models the 'Unloading rule' drop-down is disabled as these model types only use their inherent unloading rules. For the other hysteresis models the following unloading rules may be chosen:

❑ **Initial stiffness**

❑ **Secant stiffness**

❑ **Scaled stiffness**

❑ **Emori-Schnobrich**

❑ **Drain-2D**

To make hysteresis and unloading rule settings to directly match industry-named hysteresis models see the mapping table below.

Properties for each degree of freedom / common to all freedoms

Force / Displacement curve data to define each linear segment defining the backbone of a hysteresis model and hysteresis parameters must be specified for each chosen degree of freedom. Selecting the appropriate radio button and the accompanying **Specify** button will launch a separate sub-dialog on which the data can be defined. See **Joint Material Models** for details.

Note that it is possible to select different unloading rules for each hysteresis loop.

Settings required to directly match industry named hysteresis models

Hysteresis loop and LUSAS Hysteresis Model	Unloading rule	Linear segments *	Equivalent industry named Hysteresis Model
H1: Kinematic hardening	Emori and Schnobrich	1	Degrading bi-linear
H1: Kinematic hardening with slackness	Initial stiffness	3	Bi-linear with slackness
H1: Origin centred	(inherent)	1	Origin-centred bi-linear
H1: Peak oriented	(inherent)	1	Peak oriented
H1: Kivell	Emori and Schnobrich	1	Kivell degrading stiffness
H1: Pinching point	(Any)	1	Pivotal plastic model
H1:Modified Takeda	Emori and Schnobrich or Drain-2D	1	Modified Takeda
H1: Origin-centered H2 :Muto outer	(inherent) Initial stiffness	3	Muto degrading tri-linear
H1: Kinematic hardening H2: Fukada outer	Initial stiffness Emori and Schnobrich	3	Fukada degrading tri-linear
H1: Peak oriented H2: Sina outer	(inherent) Emori-Schnobrich	3	Sina degrading trilinear
H1: Stewart	Scaled stiffness	3	Stewart degrading stiffness
H1: Parallel reversal loading	Initial stiffness	1	Franchin

* Number of linear segments defining backbone of model

For example, to make settings for a Degrading bi-linear hysteresis model, as generally named and used in industry, select LUSAS hysteresis model 'Kinematic hardening' and the unloading rule 'Initial stiffness', and define a single line segment for the primary backbone of the curve.

Hysteretic joint model force response curves
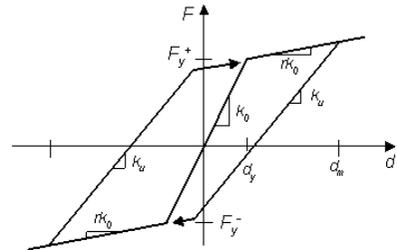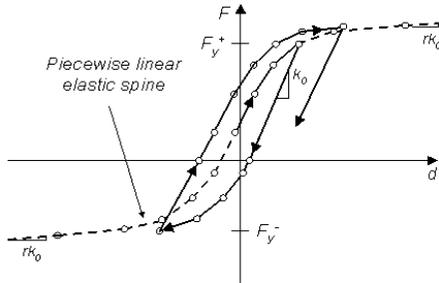
Parallel reversal loading
The Franchin model is implemented using a flat (zero) stiffness beyond the active and passive limits

Kinematic hardening
This simplest form is the same as the Degrading bi-linear model)
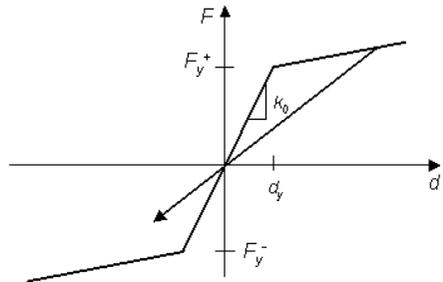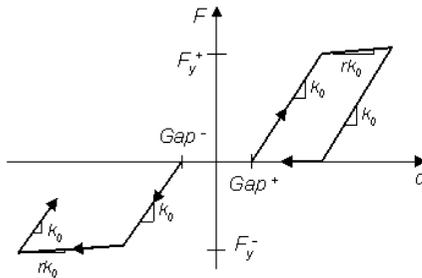
Kinematic hardening with slackness
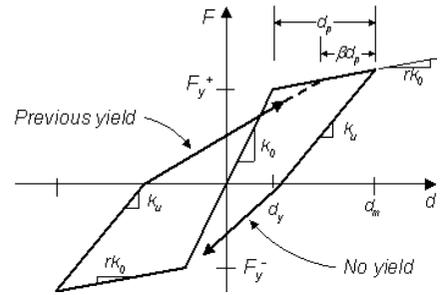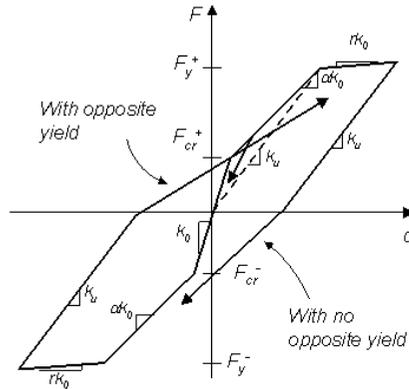**(This simplest form is the same as the Bi-linear with slackess model)**
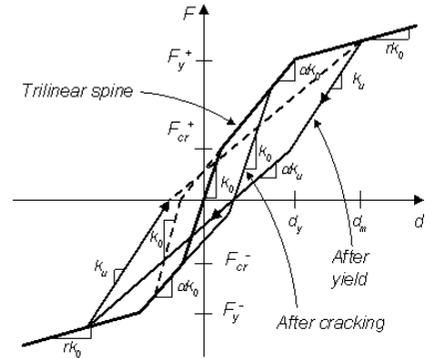
Origin centred
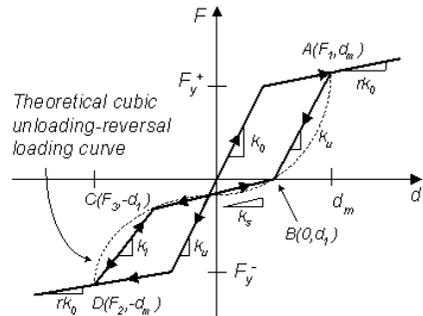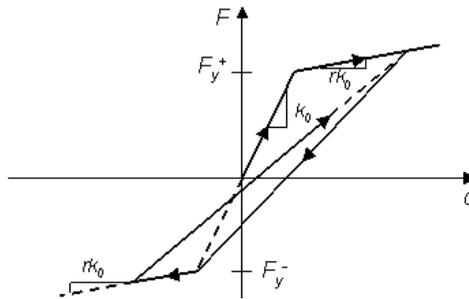
Peak oriented

Kivell
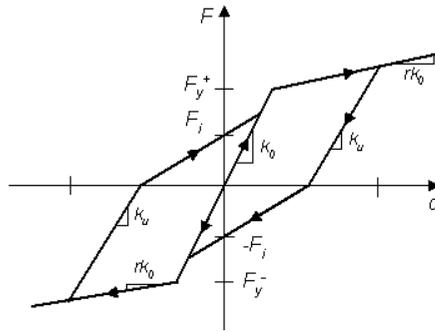
Pinching point

Modified Takeda

Muto outer

Fukada outer
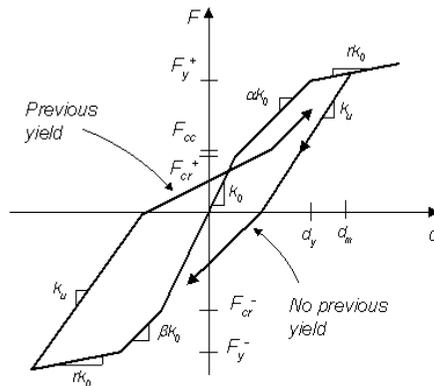
Sina outer
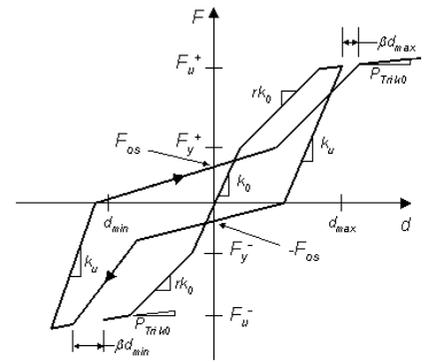
Stewart

See *LUSAS Theory Manual 1* for more details on these joint models.

Matrix properties for joint elements

For joints defined by a matrix of properties the following options are available according to the analysis category in use.

- ❑ **Assignment to** Points and Line ends, Lines, or Surfaces
- ❑ **Joint type** Joints for beams, Joint for semiloof, Joints no rotational stiffness
- ❑ **Matrix type** options are **Stiffness** (as typically used to specify a joint representing a pilecap sitting on a group of piles in a 3D model), **Mass** (for use with Non-structural mass elements), **Mass Rayleigh Damping** (for assigning damping characteristics to a joint with no stiffness).

Note that it is only possible to assign one matrix joint property (i.e. stiffness or mass or mass rayleigh damping) to a joint. Multiple assignments are not supported, but multiple coincident joints may be created to model such behaviour.

Two input options are provided:

- ❑ **First quadrant** permits entries in the top-right triangle of the top-left quadrant only, and all other entries (greyed out) are reflected automatically.
- ❑ **Full matrix** permits entries in the top-right triangle of the full matrix and the bottom-left triangle (greyed out) is reflected automatically.

Example input cell layouts for Joint type: Joint no rotational stiffness:

	u1	v1	w1	u2	v2	w2
u1						
v1						
w1						
u2						
v2						
w2						

First quadrant

	u1	v1	w1	u2	v2	w2
u1						
v1						
w1						
u2						
v2						
w2						

Full matrix

Example input for Joint type: Joint for beams (3D) is shown below where values of K1 to K8 are entered in one quadrant and reflected automatically in other quadrants. K7 and K8 are coupling terms in this case.

	u1	v1	w1	THx1	THy1	THz1	u2	v2	w2	THx2	THy2	THz2
u1	K1				K8		-K1				-K8	
v1		K2		K7				-K2		-K7		
w1			K3						-K3			
THx1		K7		K4				-K7		-K4		
THy1	K8				K5		-K8				-K5	
THz1						K6						-K6
u2	-K1				-K8		K1				K8	
v2		-K2		-K7				K2		K7		
w2			-K3						K3			
THx2		-K7		-K4				K7		K4		
THy2	-K8				-K5		K8				K5	
THz2						-K6						K6

Data entry for a symmetrical matrix

For more information please refer to the *Solver Reference Manual*.

Support Conditions

Support conditions describe the way in which the model is supported or restrained and are defined from the **Attributes > Support** menu.

A support is defined for use with a particular **analysis category**.

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. For axisymmetric models stiffness/unit length is a stiffness per unit length, per radian.

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 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

Single lift-off force	Upper and lower values of lift-off force	Upper and lower values of displacement
Simple Lift-off	**Advanced lift-off contact control**	**Gap displacement contact control**
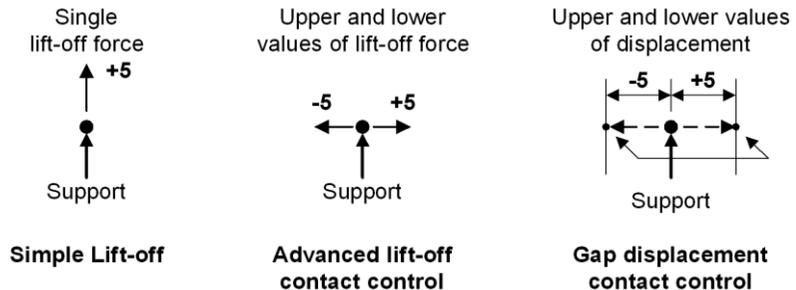
Notes

- 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 / Hygro-thermal Supports

Thermal and Hygro-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. For both:

- ❑ **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).

The following support types are for use with hygro-thermal analysis only:

- ❑ **Temperature**
- ❑ **Capillary pressure**

Note that while a structural analysis cannot be carried out without supports, transient thermal or hygro-thermal analysis can be done without 'supports'. For instance an environmental temperature load could be used instead of a prescribed temperature at a boundary node.

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.

Line supported in X

Point automatically supported in X & Y

Line supported in Y
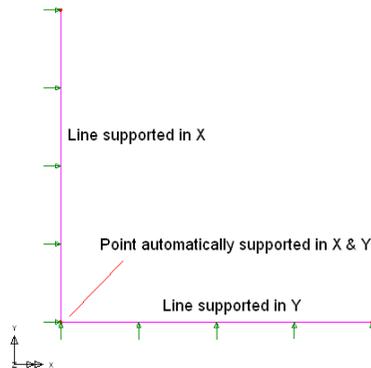
Support dependency across analyses / loadcases

- 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 [icon] support visualisation button.

Support attributes can be visualised individually by selecting an attribute name in the Attributes [icon] 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 [icon] 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 [icon].

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.
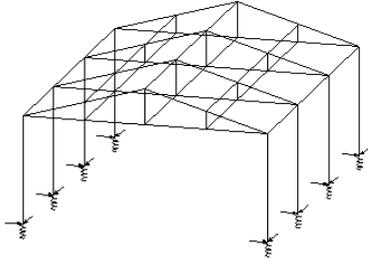
Example: Translational Supports

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.
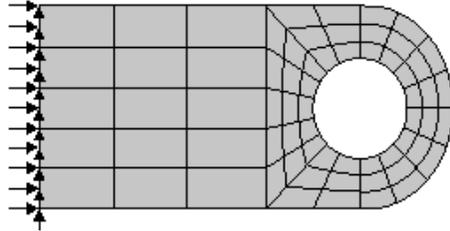
Loading Attributes

Loading attributes describe the external influences to which the model is subjected. Structural, Thermal and Hygro -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 and Initial 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.

❑ **Discrete Thermal Loads** are used to distribute a given loading pattern over full or partial areas of the model, independent of the model geometry.

Hygro-Thermal Load Types

❑ **Hygro-Thermal Loads** describe the temperature/heat and moisture input to a hygro-thermal analysis.

❑ **Discrete Hygro-Thermal Loads** are used to distribute a given loading pattern over full or partial areas of the model, independent of the model geometry.

All Structural, Thermal and Hygro-Thermal loads are feature-based loads that, once defined, can be assigned to the model geometry (or to **mesh objects** in a mesh-only model) for a stated analysis and loadcase, and they 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.

Advanced loading assignment

The Loading Assignment dialiog provides additional options to assign loading to a range of loadcases, or to one or more specified loadcases within an analysis.

❑ **Range of loadcases from** specifies the starting loadcase in the Analyses 🔍 Treeview of a range of loadcases to which the loading will be assigned. One of the next two settings will also be required.

- **Until specified loadcase** Specifies the end of the loadcase range in the Analyses 🔍 Treeview to which the loading will be assigned.
- **Until end of analysis** Assigns the loading to all loadcases that follow the specified starting loadcase in the Analyses 🔍 Treeview.

❑ **Specified loadcases** permits individual selection of one or more loadcases to which the loading will be assigned.

Notes

- Load factors of assigned loads can be changed by selecting the **Change load factor** menu item accessed from the loading name context menu.

- Subsequent editing of a load factor for a loadcase where the load factor was initially defined for a range of loadcases will change the factor used on each of the loadcases in that range. If this is not what is required, load case factors should be assigned on a per-loadcase basis.

- If a single loadcase, that was defined outside of a range of loadcases, is moved in the Analyses 🔍 Treeview to a position within a range of loadcases its load factor will be retained in addition to any load factor already specified for the range of loadcases in which it now sits, and both load factors will be applied during an analysis.

- 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 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.

Load visualisation can be toggled on and off using the [icon] load visualisation button.

Viewing loads by definition and by effect on the mesh is also possible via the [icon] load visualisation button's drop-down menu

Viewing of assigned loads on a loadcase basis requires each loadcase in the Analyses ⌚ Treeview to be set active in turn using the **Set Active** context menu item.

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.

Editing of multiple loading attributes

For particular structural loading attributes, editing of the properties of multiple load attributes of the same loading type is possible. Structural load types supported by this facility are:

- Concentrated load
- Body force load
- Global distributed load
- Face load
- Local distributed load
- Temperature load
- Tendon load

Note that other load types and other non-load attributes require editing individually.

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**
- ❑ **Body Force**
- ❑ **Global Distributed**
- ❑ **Face**
- ❑ **Local Distributed**
- ❑ **Distributed to Perimeter**
- ❑ **Temperature**
- ❑ **Stress and Strain**
- ❑ **Internal Beam Point**
- ❑ **Internal Beam Distributed**
- ❑ **Tendon**

In the following headings the LUSAS Solver name for the loading is shown in parentheses (when available).

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.
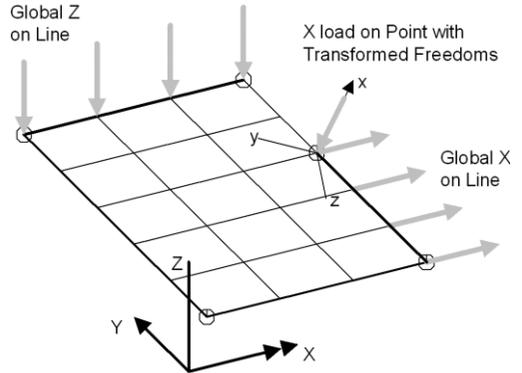
Concentrated loads are defined relative to the nodal coordinate system. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

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).
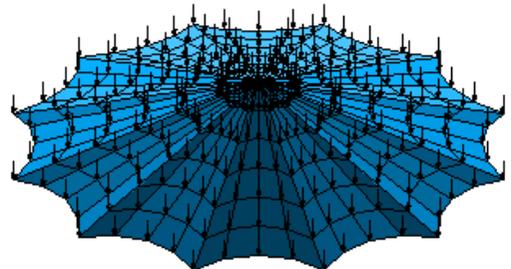
By default, Body Forces define accelerations, but an option on the **Attributes** tab of the **Model Properties** dialog, can be set so that Body Forces define a force per unit volume.

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.

In a coupled hygro-thermal analysis, when using concrete model 105 and heat of hydration (i.e. modelling the setting of concrete) the gravity load is increased from zero to its final value, depending upon the volume solidification ratio, to avoid computational difficulties when concrete is not yet fully solid. A full gravity load is attained after approximately 24 hours.

Global Distributed Load (CL)

This 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)

This 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.
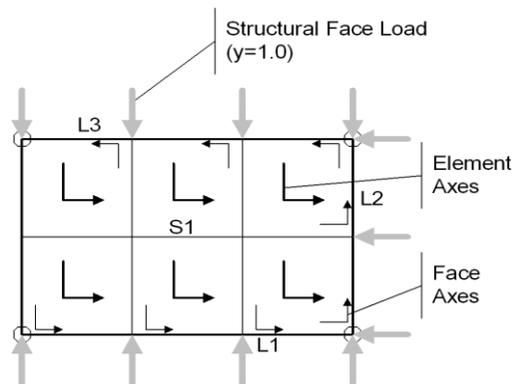
Where a loaded Line or Surface feature is common to two or more higher order features, it is possible to specify to which higher order feature elements the load is assigned.

Note that in a structural analysis the direction of the normal face load is not consistent between 2D and 3D continuum elements. For 2D continuum elements it is from the face towards the interior of the element. For 3D elements it is in the opposite direction - from the face of the element outwards.

See the *Element Reference Manual* for details of element face directions.

Local Distributed Load (UDL)

This 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).

Distributed Perimeter Load

Distributed to Perimeter loading distributes a uniformly distributed load that would otherwise be applied to a surface, as equivalent loads applied to the lines that form the perimeter of the surface. It is primarily of use in loading concrete floor slabs in buildings, where the floor slab is not modelled in full in order to reduce the complexity of the model. Surfaces representing the extent of each floor slab must be defined but no mesh assignment is required. Lines forming the Surface must, however, be assigned a suitable line mesh. Sub-beams that may support a floor slab (other than those beams that support its perimeter) can be defined without the need to model those beams as line features or elements in the model. Loading on sub-beams as a result of distributed slab loading and their self weight is represented as equivalent loads at the locations where the sub-beams meet the supporting beams at the perimeter of a surface.

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.
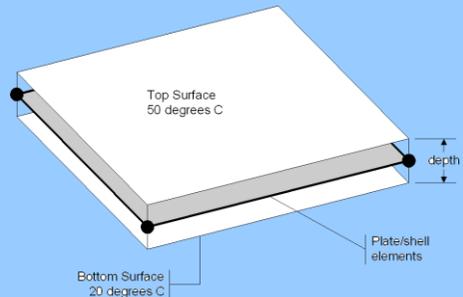
In this example (which assumes no slab eccentricity) a 0.5m thick concrete slab is at 20 degrees Celcius. The top surface is subjected to a temperature of 50 degrees Celsius and the bottom surface remains at 20 degrees Celsius.

 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 Loads (TTSIE, 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 an element name in the Stress and Strain loading dialog.

- **Target** is used to define a value for a selected results component for a feature that will be obtained when a solve has taken place. This facility is primarily for use when a specified jacking force is required to be applied to one end of a line representing a cable. To assign a target value a point must be selected in addition to any meshed feature (a line for instance) containing that point. If a

stress component is selected, all strain components are disabled and vice versa. Note that target values are only appropriate for linear materials.

❑ **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. Note that specifying a value for an initial stress or force will not necessarilyresult in that actual stress or force being obtained when a model is solved. When this is required, specify a target value (see above) instead.

❑ **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 target, 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 targets, 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.
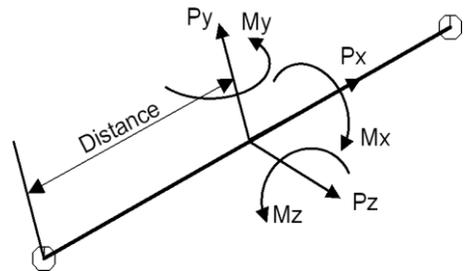
See the *Element Reference Manual - Appendix A* for examples of internal beam loading (ELDS) types. Refer to each element's main listing to see if an element supports this loading type.

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.
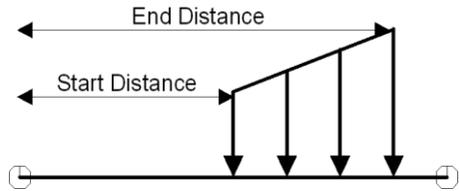
Note that for projected loading on a line not sitting in a primary plane the start and end points of each load set are based upon projecting the line back into either an XY, YZ, or ZX plane (depending upon the load component selected) with the start of the internal loading pattern being measured from 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.

Tendon Loading

Definition of tendon loading is only available in selected LUSAS Bridge and LUSAS Civil & Structural software products. The **Tendon** load attribute associates previously defined tendon properties and tendon profiles with a specified prestress force and associated jacking and loss details. Once defined, the tendon loading attribute is stored in the Attributes ⚙ Treeview and can be directly assigned to selected relevant features in a model. Tendon loading is converted into discrete structural loading for prestress calculation purposes.

See **Prestress loading** in the Application Manual (Bridge, Civil & Structural) for more details.

Prescribed and Initial 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.
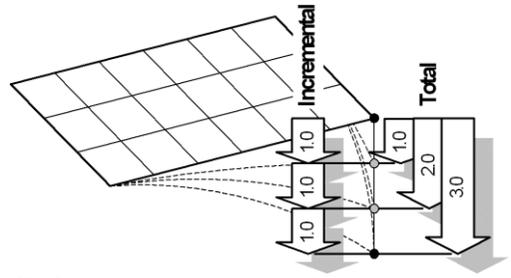
Notes

- 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. Large prescribed rotations will require a geometrically nonlinear analysis.

- 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 superseded 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.

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. An initial velocity can be input direct. To input an initial acceleration an Advanced setting must be specified. Please contact technical support in this instance.

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 loading facility to define the target to be loaded.

Discrete structural loads generated by LUSAS

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 the prestress loading facility. Manual manipulation of discrete loads generated by these facilities is disabled by default, with the intention that they should be edited or deleted using the facility that created them. This restriction can however be removed, if desired, to allow manual editing and deletion of these loads, by selecting the File> Model Properties>Attributes menu item and selecting Allow modification of LUSAS-generated attributes.

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.
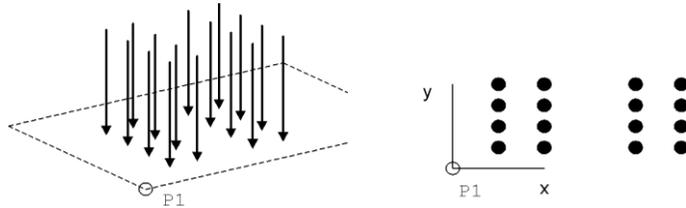
Patch Load

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.
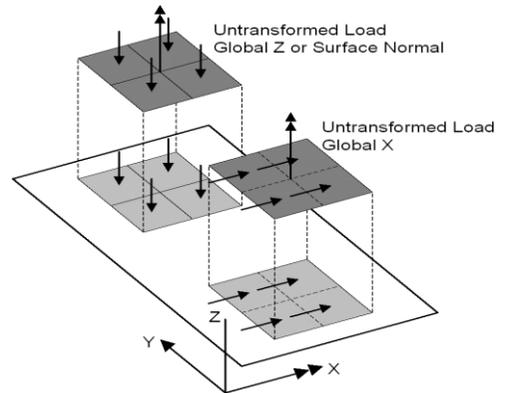
In the example shown, untransformed global loads are projected onto a model normal to the patch definition. The projection vector is denoted by a double-headed arrow on a visualised patch. The direction of the load applied to the model is defined using the untransformed load direction.

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.
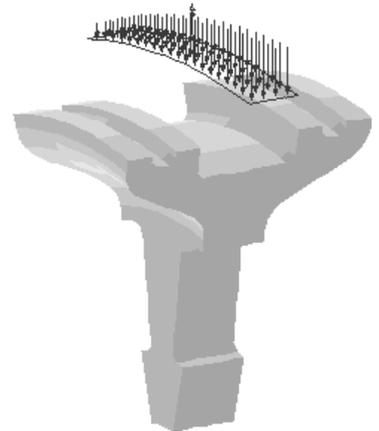
This example shows a typical 3D patch load where the patch is defined in space and projected onto the model.

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.
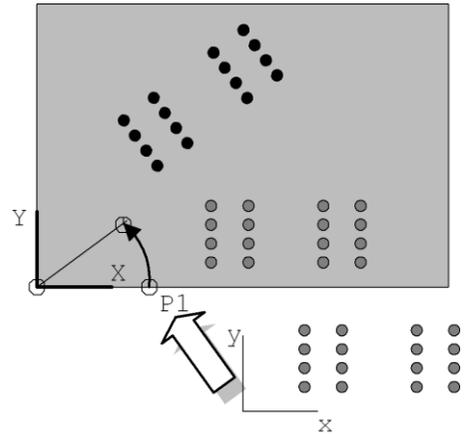
In the example shown right the Point load defined about local xy axes is assigned to Point 1 subject to a patch direction transformation using a 30 degree xy rotation about the global origin. Note that the local origin of the patch load is rotated and repositioned as well as the patch itself.

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 loading facility to define the target to be loaded.

In addition, the discrete load can be specified to:

❑ **Project onto line (2D line beam and frame models)** Discrete loading is applied to the beam as corresponding forces and moments along the beam.

- **Ignore projection direction moments** When point or patch loads are applied at an offset to a line beam and the projection direction

is in the same plane as the applied loading, the associated moment induced on the line beam can be ignored by checking this option.

❑ **Project over area (grillages, slabs and 3D space frames)** where the area may be defined by a grid of beam elements (for grillages and space frames), a thick plate structure (for slabs), or the face of a solid model.

❑ **Project over area (shell models)** where the area may be defined by a shell structure (for slabs).

❑ **Project into volumes (solid models)** Primarily for use with tendon loading with 3D models.

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.

Notes

When a search area is used as part of assigning a discrete load, the search area icon in the Attributes ✦ Treeview will change from grey to coloured to show the search area is in use. Whilst this looks the same as when a search area is assigned to geometry features in a model, for the former no assignment-related context menu options are available. The search area is associated with the assigned loading and can only be disassociated from it by visiting the discrete loading assignment dialog and choosing a different search area, or deassigning the discrete loading from the model.

Editing of Discrete Loading Data

User-defined discrete loading

Each discrete loading attribute in the Attributes Treeview has a context menu item named Edit...

- Selecting the **Edit...** 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 of patch at any time.

Pre-defined discrete loading

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 loading facility) 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 Attributes 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	Hydrostatic Patch Load
Default number of divisions showing insufficient discrete point loads.	Increased number of divisions on assignment. The double arrow vector indicating patch orientation.
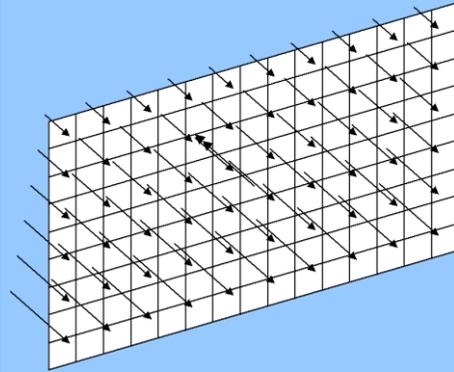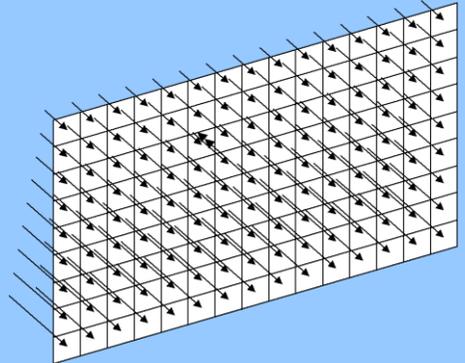
Thermal Loading

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** or a **Load Curve**. For information on which load types can be applied to which element types, see the *Element Reference Manual.*

Thermal loading is accessed via the **Attributes > Loading** or **Attributes > Loading > Thermal** menu items depending upon the type of user interface in use.

Thermal Load Types

Thermal load types available are:

- ❑ **Prescribed**
- ❑ **Environmental conditions**
- ❑ **Rate of Heat Inflow (Concentrated)**
- ❑ **Rate of Heat Inflow/Unit Length)**
- ❑ **Rate of Heat Inflow/Unit Area (Flux)**
- ❑ **Rate of Heat Inflow/Unit Volume**
- ❑ **Rate of Heat Input/Unit Volume (user)**

Prescribed

Defines a prescribed temperature for an element.

❑ The **Total** prescribed load type defines the total temperature at a given node at a specified increment.

❑ The **Incremental** prescribed load type adds to any temperatures present from a previous increment.

Environmental Conditions

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.

❑ **Environmental temperature** is the temperature of the surrounding air/fluid to which the object is losing heat to (or gaining heat from).

❑ **Convection heat transfer coefficient** is in units of 'power per unit area per unit temperature' (e.g. $J.s^{-1}.m^{-2}.K^{-1}$).

❑ **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. If a non-zero radiation heat transfer coefficient is specified, the analysis becomes nonlinear and **Nonlinear Control** must be used.

❑ **Reference temperature** (only available when **Temperature dependent** is chosen) refers to the model temperature, not the environmental temperature. It 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 other properties will be constant for all temperatures.

Defining temperature dependent properties will turn a linear thermal field problem into a nonlinear thermal problem.

Notes

- If heat transfer coefficients vary on a specified face the values will be interpolated using the shape functions to the Gauss points.

- **Load curves** can be used to maintain or increment the environmental temperature, or the Rate of Heat Inflow as a nonlinear analysis progresses

- Automatic load incrementation within the **Nonlinear Control** can be used to increment environmental loading.

- If external air/fluid temperature and associated convection and radiation heat transfer coefficients vary in time, pre-defined load curves for the Hygro-thermal analysis entry can be specified on the Environmental conditions assignment dialog at the load assignment stage in order to vary the values accordingly.

Rate of Heat Inflow (Concentrated)

Rate of Heat Inflow (Concentrated) 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.

Rate of Heat Inflow/Unit Length

- Requires single value of heat inflow per unit length to be defined.

Rate of Heat Inflow/Unit Area (Flux)

- Requires single value of heat inflow per unit area to be defined.

Note that for both 2D and 3D continuum elements a positive flux is from the face towards the interior, that is, a positive thermal flux raises the temperature of the continuum.

Rate of Heat Inflow/Unit Volume

Defines the internal heat generation for an element. Positive loading values indicate heat generation and negative values indicate heat loss.

- ❑ **Rate of Heat Inflow** a single value of heat inflow per unit volume.
- ❑ **Reference temperature** (only available when **Temperature dependent** is chosen). Must be specified for each set of properties.

Defining temperature dependent properties turns a linear thermal field problem into a nonlinear thermal problem.

Rate of Heat Inflow/Unit Volume (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.

Notes

- **Load curves** can be used to maintain or increment the Rate of Heat Inflow as a nonlinear analysis progresses

- Automatic load incrementation cannot be used with Rate of Heat Inflow loading.

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, h, may be given by $h = C [\delta t]^{(1/3)}$, where C is a constant and $[\delta t]$ is the temperature difference between the surface and the environment 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. The convective coefficients calculated are entered into the grid of the thermal loading attribute, along with the appropriate reference and environmental temperatures in each row. This single attribute 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 to the model and solve.

5. Since the problem is one-dimensional the solution may be checked to ensure that the convection coefficient has been correctly interpolated.

Hygro-Thermal Loading

Hygro-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** or a **Load Curve**. For information on which load types can be applied to which element types, see the *Element Reference Manual.*

Hygro-thermal loading is accessed via the **Attributes > Loading** or **Attributes > Loading > Hygro-thermal** menu items depending upon the type of user interface in use.

Hygro-Thermal Load Types

Hygro-thermal load types available are:

- ❑ **Initial conditions**
- ❑ **Prescribed**
- ❑ **Environmental conditions**
- ❑ **Rate of Heat and Moisture Inflow (Concentrated)**
- ❑ **Rate of Heat and Moisture Inflow/Unit Length)**
- ❑ **Rate of Heat and Moisture Inflow/Unit Area (Flux)**
- ❑ **Rate of Heat and Moisture Inflow/Unit Volume**

Initial Conditions

In a hygro-thermal analysis initial conditions are assigned directly to the model (on a per element basis) as a 'load'. This means that neighbouring elements can have discontinuous initial conditions. This allows for modelling the staged pouring of mass concrete in situations where the previous stage poured has a temperature close to the environmental temperature (or higher if there was not enough time for the heat of hydration to dissipate), and a newly poured stage has been mechanically cooled to a lower temperature.

Specifying values for old, cured concrete

- ❑ **Temperature** This can be defined in degree Kelvin (K) or degrees Celsius (C) as specified during model startup.

- ❑ **Concrete relative humidity** For old, cured concrete the value for relative humidity should lie between 0 and 1. Note that it is not given as a percentage. A concrete relative humidity of, say, 0.7, defines a moisture content in the concrete for the given initial temperature when the concrete is in a state of equilibrium with the environment that has the same temperature and relative humidity. In other words, concrete has a given relative humidity if it has stayed for a sufficiently long time in the environment with the same relative humidity. The entered value for relative humidity is used to compute the initial value for the 2nd degree of freedom (capillary pressure Pc from Kelvin's equation:

$$P_c = -\rho_w R_v T \ln(RH)$$

where:

ρ_w =1000 [kg/m^3]- density of water,

R_v = 461.915 [J/kg.K] - specific gas constant for water vapour.

T = 273.14+20 [K] - the given initial temperature converted to Kelvin

The explicit input of RH is required for old, already cured concrete.

- ❑ **Cured concrete** Check this box to allow input of values for cured concrete.

Specifiying values for new concrete

❑ **Temperature** is the only input required. Relative humidity cannot be defined for a liquid state of concrete

New concrete starts in a liquid state. This condition is set automatically when Concrete Heat of Hydration is specified in the material properties. For new concrete, initial, final porosities and initial water saturation are computed automatically. The initial porosity is computed from the relative volume required to contain the liquid water of the concrete mix plus the volume ratio of the entrapped air. The required initial capillary pressure is computed by solving the Sorption curve equation:

$$S_w = \left[1 + \left(\frac{P_c}{a_c P_{atm}} \right)^{\frac{b}{b-1}} \right]^{\frac{-1}{b}}$$

Prescribed

Prescribed loading is not used often in hygro-thermal analysis because it is rarely the case that the temperature or the relative humidity at a point or surface or volume remain constant, with a known value for some period of time covered by the analysis. Prescribed loading is equivalent to environmental temperature loading with a very high convection coefficient. Even when concrete is in contact with a body with a fixed temperature it might be more accurate to model it as an environmental load which can account for the thermal and hygro barrier imposed by the rough contact between them.

Hygro-thermal prescribed loading requires the following to be specified:

❑ **Temperature** This can be defined in degree Kelvin (K) or degrees Celsius (C) as specified during model startup.

❑ **Concrete relative humidity** This can be defined for concrete having:

- **Contact with water** For this, a value of 1.0 is used for concrete relative humidity. Water saturation = 1, and from Kelvin's equation the capillary pressure = 0.

- **Contact with air** A typical use would be when concrete is subjected to steam curing. If a value of 1.0 is specified for concrete relative humidity, water saturation = 1, and capillary pressure = 0. If a value of <1 is specified, from Kelvin's equation, a prescribed capillary pressure >0 is obtained. Such as prescribed loading would only be used for academic reasons. Better engineering results would be achieved if environmental temperature loading is used.

The capillary pressure is the same for both cases because the preservation of mass (or mass transfer) equation considers the <u>moisture</u> in concrete which is the sum of liquid water and water vapour. The result is that, in both cases, the water saturation in the concrete will increase.

- For the 'Contact with water' situation there is no condensation and the increase of water saturation in the concrete does not bring about a release of latent heat.

- For the 'Contact with air' situation the increase of the water saturation in the concrete is due to condensation. Condensation releases the latent heat of phase change and it raises the concrete temperature.

Note that prescribed relative humidity is converted to capillary pressures at element nodes. When solved, all concrete results are computed at element Gauss Points. In Modeller, these values are extrapolated to the element nodes. Because of this extrapolation the nodal value reported by the Modeller can differ from the prescribed value.

Note

- When a Prescribed loading is assigned to a **load curve**, the load curve is applied to both specified components of the load (unlike the Environmental Conditions loading types where each component can be assigned its own load curve). If both components of the prescribed load are non-zero, each component should be applied in a separate load case.

Environmental Conditions

Environmental Conditions load types model external air/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.

- ❑ **Environmental temperature** is the temperature of the surrounding air/fluid to which the object is losing heat to, or gaining heat from.
- ❑ **Convection heat transfer coefficient** (e.g. $J.s^{-1}.m^{-2}.K^{-1}$). For early age concrete its value depends upon the curing conditions, environmental temperature and wind velocity.
- ❑ **Radiation heat transfer coefficient** (e.g. $J.s^{-1}.m^{-2}.K^{-4}$) It is determined by multiplying the emissivity of the material by the Stefan-Boltzmann constant. If a non-zero radiation heat transfer coefficient is specified, the analysis becomes nonlinear and **Nonlinear Control** must be used.
- ❑ **Relative humidity** is the relative humidity of the environment. Enter a value between 0 and 1. Note that it is not given as a percentage.
- ❑ **Vapour mass coefficient** can be calculated by means of the Chilton-Colburn analogy if there is no reliable experimental data. The calculation is invoked by a negative value for the coefficient, after a load curve, if present, is applied. The Chilton-Colburn analogy applies for a solid, smooth surface:

$$\alpha_{v,env}^{analogy} = 0.913 \frac{\alpha_{T,env}}{\rho C_{air}} = 55.74 \times 10^{-5} \times \alpha_{T,env}$$

where

$\rho C_{air} = 1.17 \times 1400$ [J/m³.K] is the specific heat of air per unit volume.

and 0.913 is the ratio of the mass and thermal diffusivities for air–water vapour mixture at 298 K.

For a porous surface, heat is still convected through the whole surface while water vapour is convected via the pores exposed to the environment.

Thus it is reasonable to scale $\alpha_{v,env}^{analogy}$ by the porosity n

$$\alpha_{v,env} = 55.74 \times 10^{-5} \times n \times \alpha_{T,env}$$

Notes

- If heat transfer coefficients vary on a specified face the values will be interpolated using the shape functions to the Gauss points.

- **Load curves** can be used to maintain or increment the environmental temperature as a nonlinear analysis progresses.

- If environmental air/fluid temperature and associated convection and radiation heat transfer coefficients vary in time, pre-defined **load curves** for the Hygro-thermal analysis entry can be specified on the Environmental conditions assignment dialog at the load assignment stage in order to vary the values accordingly.

- For concrete, as it dries out because of a decrease of water saturation due to self desiccation (that is the liquid water becomes chemically bonded in the hydrated product), the drying out is included in the Heat of Hydration 'load' that is defined via the material property dialog. Curing at higher relative humidity can be used to reduce and prevent cracking. However, it should be noted that if the environmental relative humidity is higher than the concrete relative humidity this does not guarantee transfer of moisture from the environment to the concrete. If the concrete temperature is higher than the environmental temperature, the water vapour content in the concrete pores would be higher than the environmental water vapour content, for the same and even lower relative humidity of concrete.

Rate of Heat and Moisture Inflow (Concentrated)

❑ **Rate of Heat Inflow** a single value of heat inflow.

❑ **Rate of Moisture Inflow** a single value of moisture inflow.

Example use: 2D model with heating wire running along Z

Rate of Heat and Moisture Inflow/Unit Length

❑ **Rate of Heat Inflow** a single value of heat inflow per unit length.

❑ **Rate of Moisture Inflow** a single value of moisture inflow per unit length.

Rate of Heat and Moisture Inflow/Unit Area (Flux)

❑ **Rate of Heat Inflow** a single value of heat inflow per unit area.

❑ **Rate of Moisture Inflow** a single value of moisture inflow per unit area.

Note that for both 2D and 3D continuum elements the positive flux is from the face towards the interior, that is, a positive thermal flux raises the temperature of the continuum.

Rate of Heat and Moisture Inflow/Unit Volume

❑ **Rate of Heat Inflow** a single value of heat inflow per unit volume.

❑ **Rate of Moisture Inflow** a single value of moisture inflow per unit volume.

❑ **Reference temperature** (only available when **Temperature dependent** is chosen). Must be specified for each set of properties.

Defining temperature dependent properties turns a linear thermal field problem into a nonlinear thermal problem.

Example Applications of Rate of Heat and Moisture Inflow

Prescribed loading type	Rate of heat inflow	Rate of moisture inflow
Rate of heat and moisture inflow at a node (concentrated)	Heating wire running along Z axis in a 2D model	None
Rate of heat and moisture inflow per unit length	Heating wire.	None
Rate of heat and moisture inflow per unit area	Heating blanket, or cooling from surface water evaporation, or sun radiation flux.	None
Rate of heat and moisture inflow per unit volume	LUSAS heat of hydration (applied via the hygro-thermal material dialog for several types of cements).	Concrete self desiccation(*).
Rate of heat and moisture inflow per unit volume	Manually computed heat inflow rate from an experimental temperature evolution in an adiabatic test.	Interaction of hydrating cement with dry or wet aggregate.

Notes:

- To model pipe cooling where the pipe water temperature is assumed to stay constant, use a Prescribed temperature on the lines representing the cooling pipes.

- **Load curves** can be used to maintain or increment the Rate of Heat and Moisture Inflow as a nonlinear analysis progresses.

- When a Rate of Heat and Moisture loading is assigned to a **load curve**, the load curve is applied to both specified components of the load (unlike the Environmental Conditions loadtypes where each component can be assigned its own load curve). If both components of the Rate of Heat and Moisture loading are non-zero, each component should be applied in a separate load case.

- There is no time restriction on thermal rate of heat inflow. Temperature can increase /decrease indefinitely. For moisture rate of inflow there can be inflow only until full water saturation (1.0) is reached; similarly, rate of moisture outflow can be applied only up to saturations when the liquid network becomes disconnected (usually when water saturation <0.2).

- Flux load (rate of heat inflow per unit area) can model only the radiation heat received from the sun. The radiation heat lost to the environment cannot be modelled accurately.

- Rate of Heat and Moisture Inflow loading can be applied at a point (concentrated load), line, surface or volume. A positive value represents heat generation or moisture inflow; a negative value represents a heat sink or moisture outflow.

- Automatic load incrementation cannot be used with Rate of Heat Inflow loading.

Discrete Thermal / Hygro-Thermal Loads

Discrete thermal / hygro-thermal loads are used to distribute a given loading pattern over full or partial areas of the model, independent of the model geometry. Discrete 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 loads differ from feature - based loads in that they are not limited to application over whole features. Discrete 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.

Discrete point or patch loads are defined in relation to their own local coordinate system and are always assigned to points. Discrete loads differ from feature based loads in that they are not limited to application over whole features and may be effective over full or partial areas of the model. Search areas may be used to restrict the area of application of a discrete load.

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. When applied, 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.

Discrete Thermal / Hygro-Thermal Load Types

A discrete thermal / hygro-thermal 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:

- ❑ **Rate of Heat (and Moisture) Inflow (Point)**
- ❑ **Flux (Patch)**

Rate of Heat (and Moisture) Inflow (Point) Load

Rate of Heat (and Moisture) Inflow (Point) loads provide concentrated loading at specified locations. These 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. Each discrete point requires a flux load to be specified (in Watts if SI units are being used).

This example uses 16 distinct load values. The loads are applied to the model as distinct loads.
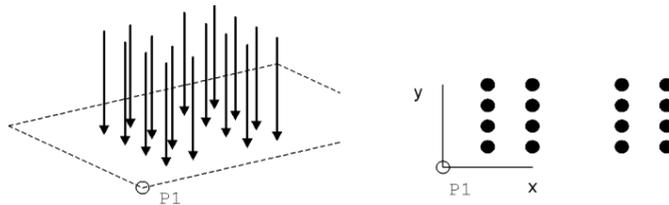
Flux (Patch) Load

Flux (Patch) loads provide a loading per unit length or per unit area. Each point of a discrete patch flux loading needs to be specified (in Watts/m^2 if SI units are being used). Flux Patch 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 straight sided patch 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

Note that for both 2D and 3D continuum elements a positive flux is from the face towards the interior, that is, a positive thermal flux raises the temperature of the continuum.

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.

- For both 2D and 3D continuum elements the positive flux is from the face towards the interior, that is, a positive thermal flux raises the temperature of the continuum.

Defining Discrete Thermal / Hygro-Thermal 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 **Rate of Heat Inflow (Point)** load each attribute defines multiple loads, one concentrated load at each given vertex.

❑ For **Flux (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.

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 accordingly. 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.
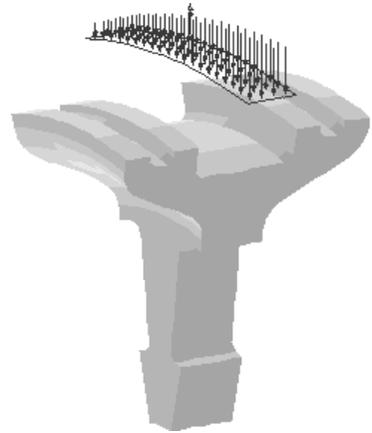
❑ **Patch load divisions** specify the numbers of divisions in the local x and y directions of the patch being assigned. The divisions are used to split the applied patch into individual component loads before they are in turn used to calculate equivalent nodal loads on the model. By default, the patch 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 Thermal / Hygro-Thermal Loads

Discrete loads are independent of features therefore their application is quite flexible. The load assignment parameters are explained below:

❑ **Patch Transformation** Changes the patch orientation. For example, a flux patch load may be skewed by applying a rotation transformation when assigning the load.

In the example shown right the Flux Patch 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.
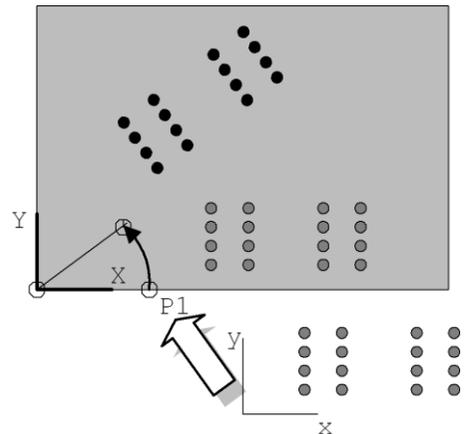
To rotate a patch about its centre, define the patch with its local origin at its centre

❑ **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.

In addition, a discrete patch load can be specified to:

❑ **Project over surface of volume** This option is used to project discrete loads over the surface of a volume.

❑ **Project into volume (solid models)** This option is used to project discrete loads into volumes (solid models).

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 Thermal / Hygro-Thermal Loading Data

Each discrete loading attribute in the Attributes Treeview has a context menu item named 'Edit'.

- Selecting the **Edit...** 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 of patch at any time.

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.

Search Areas

Search areas may be used to restrict the area of application of **discrete loads** (point and patch) on both 2D and 3D models. This is useful for several reasons:

❑ **Improved Control of Load Application.** For 2D models 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 loading facility to define the target to be loaded.

In the 2D 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.
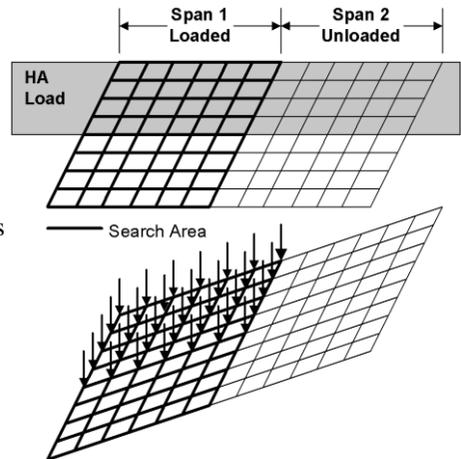
Tip. Search areas should normally always be used if the model is three-dimensional as, for example, for box-section or cellular construction decks, and discrete loads are applied.

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

Grillage members, represented by a line meshed with a single line mesh division, and forming a 4-sided 'cell' are automatically included in a search area. Overhanging grillage members, such as those seen on 3-sided or 2-sided 'bays' within a grillage model, and multi-edged cells and multiple overhanging line members affect how a search area is calculated, and in different ways. The differences can be explained as follows:

3-sided bays

Overhanging grillage members, such as those seen on 3-sided 'bays' of a grillage, where the outer side of a 'cell' is missing, and where the opposite side of the cell is represented by a line meshed with a single line mesh division, are included in the search area. In these cases, to save you from having to connect many point pairs together with dummy lines to form a closed 'cell', LUSAS automatically 'closes' the cell for the purposes of calculating the search areas, but does not create any additional features or assignments to the model in order to do so.
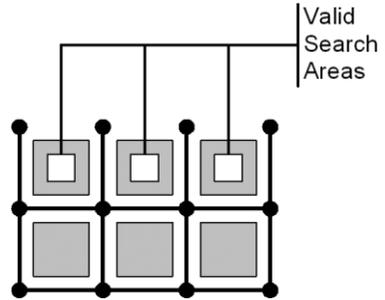
Valid Search Areas

In cases where the outer side of a 'cell' is missing, and the opposite side of the cell is represented by more than one line a dummy Line needs to 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 (1 DIV).

2-sided bays

Overhanging grillage members, such as those seen on 2-sided 'bays' of a grillage, are not included in the search area by default. For these, LUSAS does not automatically 'close' the cell for the purposes of calculating the search areas. Instead, a dummy Line needs to 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 (1 DIV).

Invalid: 2 edges overhanging from same node

Made valid by connecting points with a null line mesh
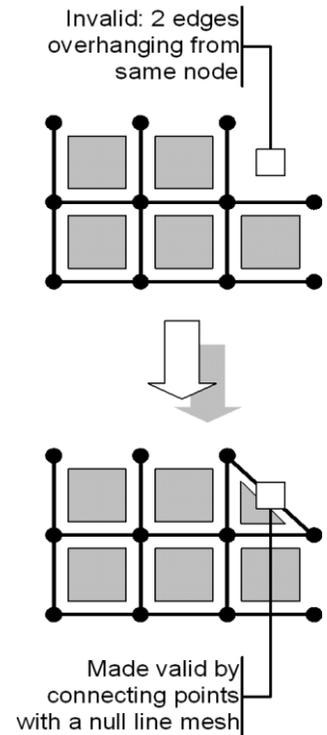
Multi-edged cells

Grillage 'cells' of more than four edges (line members) are automatically subdivided into triangles, but overhanging elements are only included if divided by no more than one edge.

For the invalid region shown, 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 (1 DIV). If done, the resulting cell of five edges will be subdivided into 4 triangles.
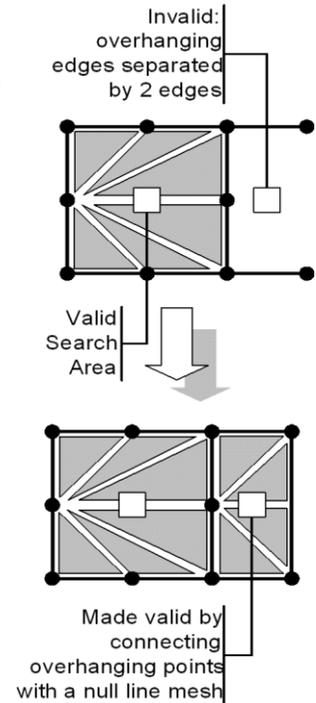
Multiple overhanging line members

There is no limit to the number of edges (or line members) that may hang over the main body if the overhanging members are only separated by one edge (see right).

In this cases LUSAS automatically 'closes' the cell for the purposes of calculating the search areas, (and does not create any additional assignments to the model in order to do so) and either sub-divides the resulting cells into triangles or uses a quadrilateral as appropriate.
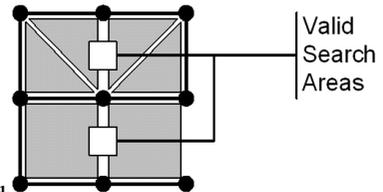
Processing Loads Outside a Search Area

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.
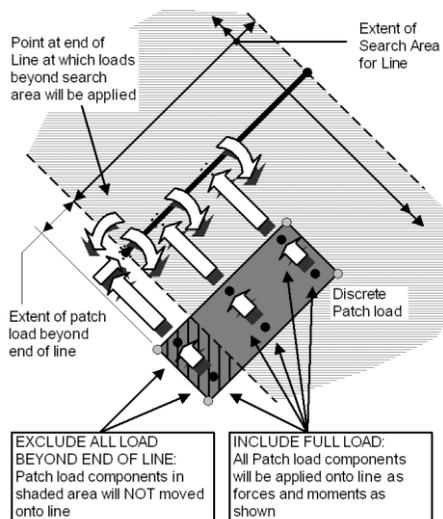
Patch Load onto Line

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.
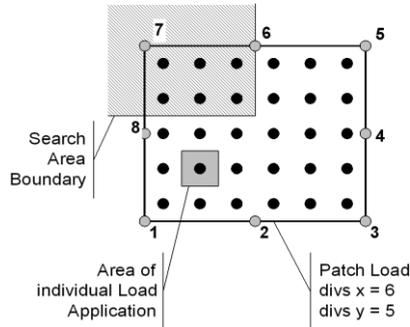
Patch Load Local Coordinates

The local coordinate set is dependent on the order in which the coordinates of the patch vertices are defined.
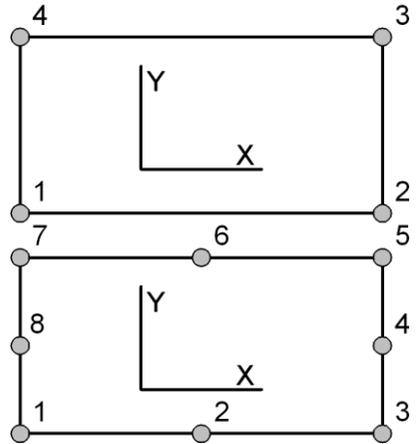
Local X Projected Load

Loads in the **local y** projected region (dark area) are lumped at nearest loading positions within the search area (light area).
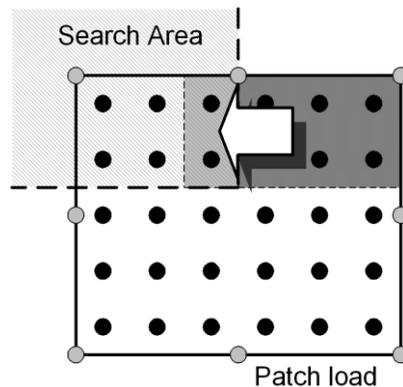

Local Y Projected Load

Loads in the **local y** projected region (dark area) are lumped at nearest loading positions within the search area (light area).
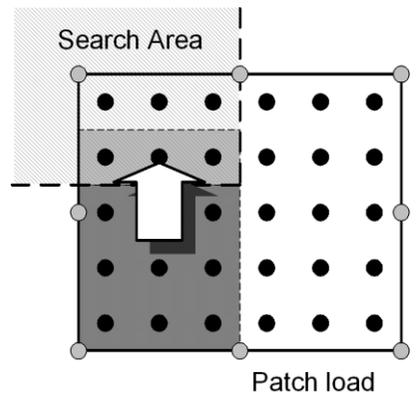
Local X and Y Projected Loads

Loads in the **local x and y** projected regions (dark area) are lumped at nearest loading positions within the search area (light area).
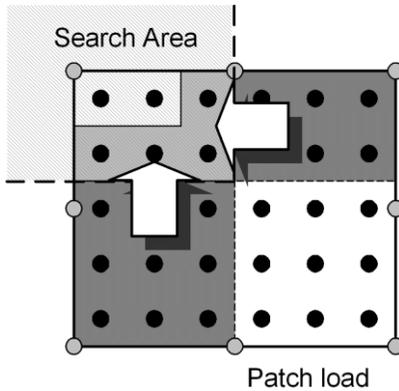
Non-Projected Load

Loads **not** in the **local x and y** projected regions (dark area) are lumped at nearest loading positions within the search area (light area).
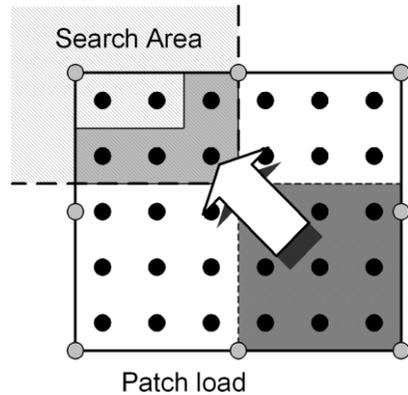
Full Local X Load

Loads in the **full local x** region of the patch (dark area) are lumped at nearest loading positions within the search area (light area).
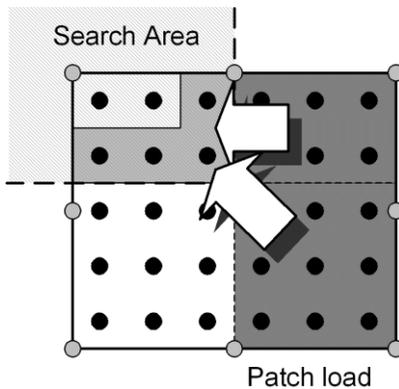
Full Local Y Load

Loads in the **full local y** region of the patch (dark area) are lumped at nearest loading positions within the search area (light area).
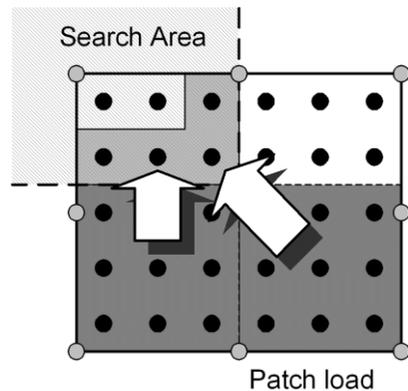
Full Load

All patch loads lying outside the search area (dark area) are lumped at nearest loading positions within the search area (light area).

Patch load
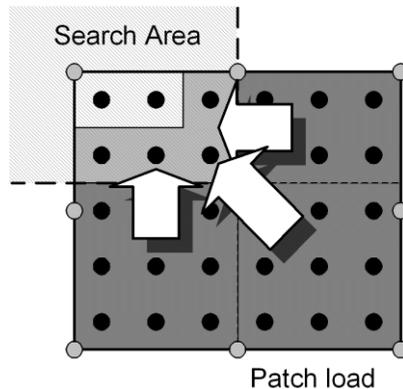
Discrete point loads over areas (grillages, slabs and 3D space frames)

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. No moment loads are created with this method.

Discrete point loads over areas (shell models)

When discrete point loads are projected over areas comprising shell elements, the offset distance from the load definition position to the load position in the search area is used to calculate a moment. A moment is also calculated when using the Bridge software product's moving load option to move loads into 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.
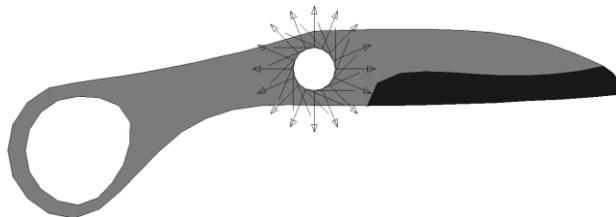
Local Coordinates

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 setting a pre-defined 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.
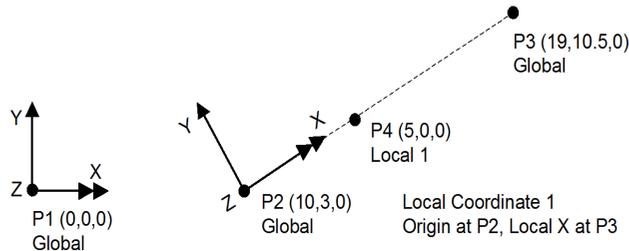
- ❑ **Transforming Nodal Freedoms** When assigned to features the effect is to transform the degrees of freedom of the underlying element nodes. This has the effect of transforming the directions of applied global load and support conditions. In the example below, global freedoms are transformed to radial 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. A surface local coordinate system is also available.

Symbol   **Local coordinate type**

❑ **Cartesian**   - Based on standard x, y and z coordinates arbitrarily oriented in space.
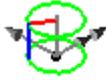
❑ **Cylindrical** - Based on the axes of a cylinder defined by a radius, angle and distance along the cylinder axis.

- 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 (t) 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.
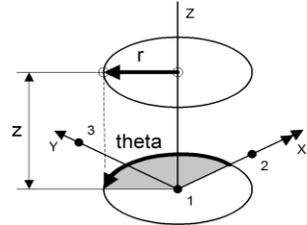
❑ **Spherical** Based on the axes of a sphere defined by a radius, tangential angle and angle around a meridian.

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

  theta (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
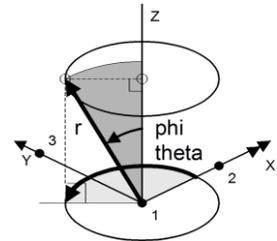
Warning. There is no equivalent spherical coordinate set in Solver, therefore freedoms cannot be transformed using this type of local coordinate system.

❑ **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.
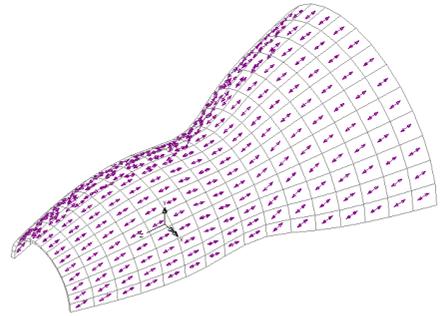
Notes

- 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.
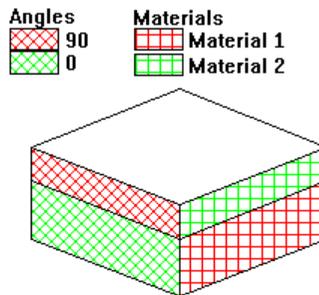
Assigning Composite Properties

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.
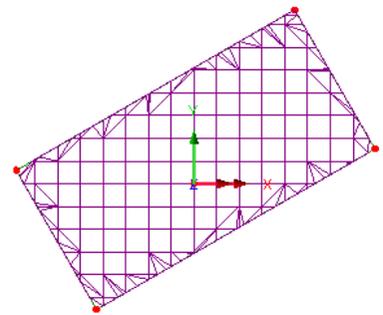
For details of how to choose a composite layer see **Setting The Active Composite Layer**.

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.
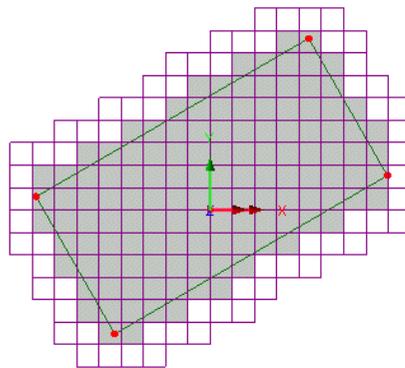
Example of draping grid being extended by one row   Example of draping grid being trimmed to a surface boundary

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**.

417

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..
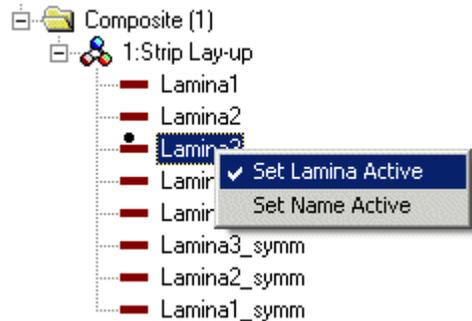
If a lamina is set active only results or orientation axes for that lamina in that composite attribute will be displayed. If the lamina name is set active, results or orientation axes will be displayed for all laminae with that name across all composite attributes.

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.
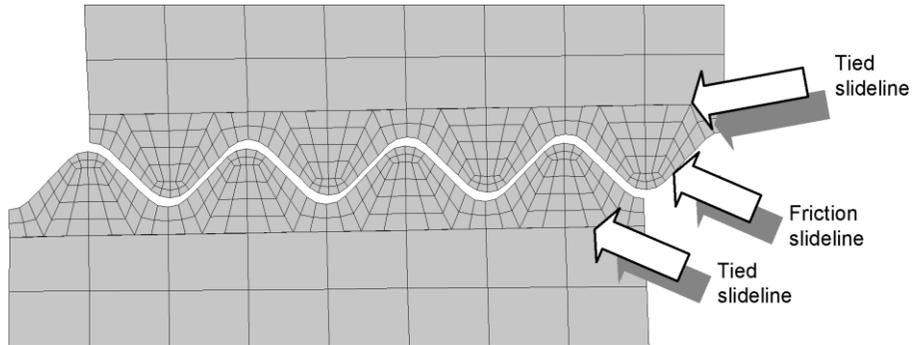
Slidelines are assigned to pairs or groups of features in a model, with one pair/group termed master and the second pair/group termed slave. An element face that lies on a slideline is called a slideline segment.

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.
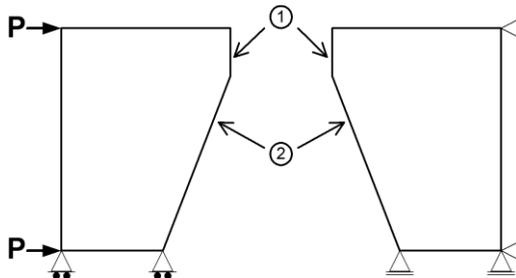
Pre-contact brings two bodies into initial contact by using interface forces that act between the slideline surfaces in order to avoid unrestrained rigid body motion. These forces act in a direction normal to each surface. One of the surfaces must be free to move as a rigid body and the direction of 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 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
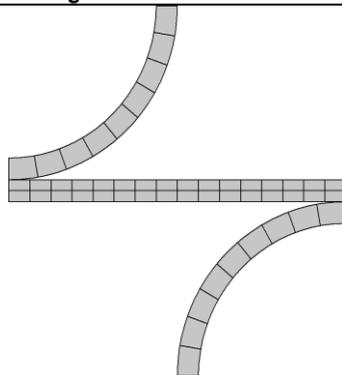
❑ The use of tied slidelines to eliminate transition meshes is recommended for areas removed from the point of interest in the structure

❑ 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.
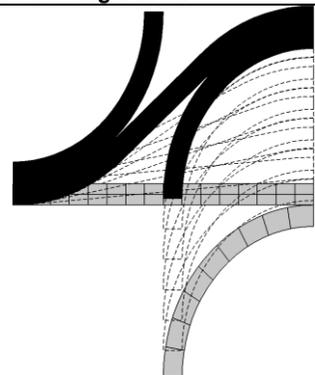
Slideline Options

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.
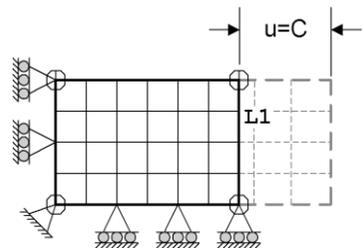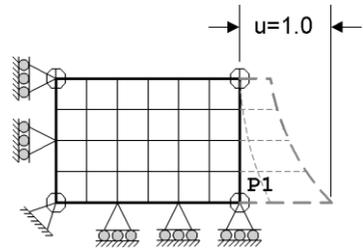
❑ **Vector Path** The nodes in the assigned features may be constrained to move along a specified vector defined by 2 Points or by 2 sets of X, Y and Z coordinates.

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.
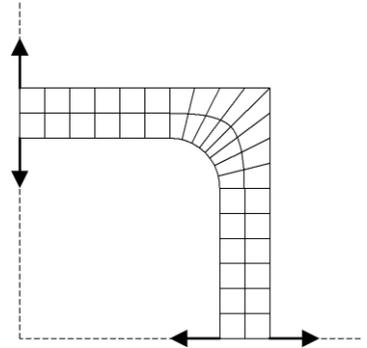
Geometric

❑ **Rigid Displacements** Rigid displacements will make the features to which they are assigned behave rigidly.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.
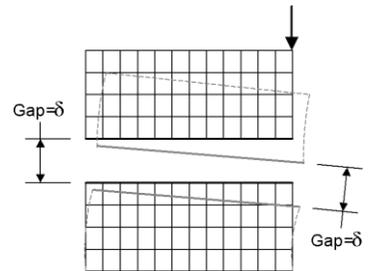
❑ **Rigid Links** Rigid links will make the features to which they are assigned behave rigidly. Each rigid link attribute can be used at one location, to create a rigid fixity between features that it is assigned to. It is similar to the Rigid Displacements constraint type, except that rotational freedoms 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.

427
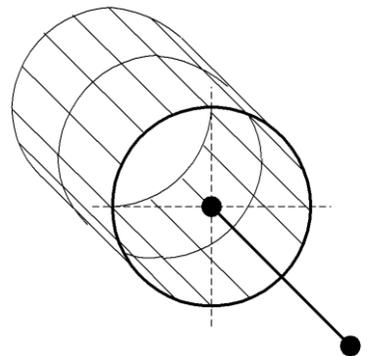
- ❑ **Planar Surface** A surface may be constrained to remain plane, the surface may translate and/or rotate but remains plane. Nodal positions may vary relative to other nodes on the surface. This type of constraint is only valid for small displacements. In this example, a planar Surface constraint 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.
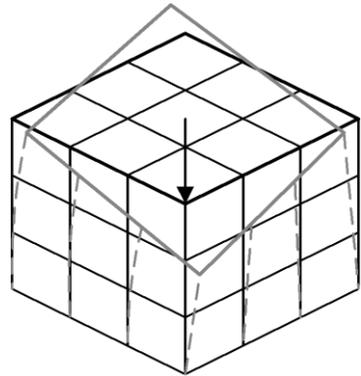
- ❑ **Straight Line** A straight line may be constrained to remain straight, the line may translate and/or rotate but will remain straight. Nodal positions may vary relative to other nodes along the line. This constraint type is only valid for small displacements. In the example shown, a straight Line 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.
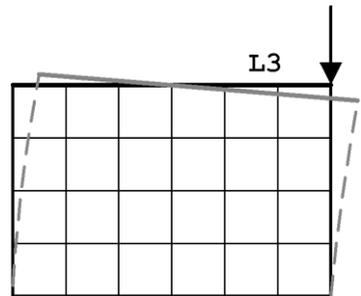
Cyclic

❑ **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.
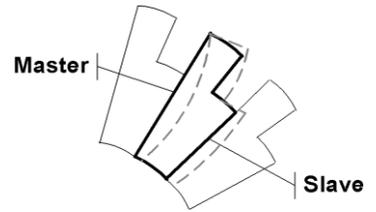
❑ **Cyclic Translation** Cyclic translational symmetry may be used to model a section from a continuous strip. The mesh on the two planes of symmetry may be different. In the example shown here, Master and Slave Surfaces define start and finish positions of repeating sections. Meshes on Master and Slave need not match.
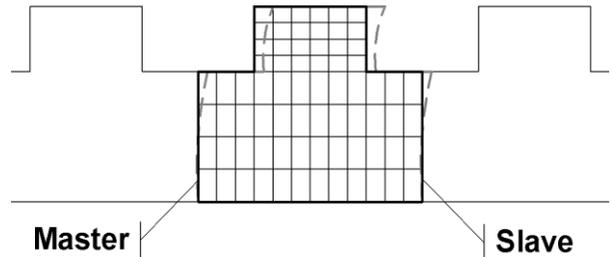
Tied Mesh

❑ **Normal Constraint** Meshes are 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. The use of rigid constraints ensures that the two sets of assigned features will both rotate with respect to their initial defined positions.
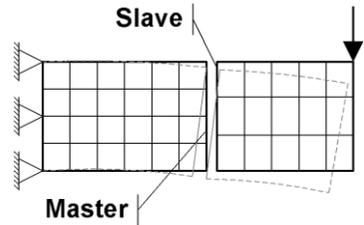
❑ **Specified Constraint** Tied meshes may be used to force two sets of assigned features to move together in a similar manner to tied slidelines. The meshes are tied along Master and Slave Lines to restrict relative movement. The mesh on the two sets of features need not match. A search direction vector is defined to limit the mesh to which it is tied. The vector defines the direction in which the constraint is applied. The use of rigid constraints ensures that the two sets of assigned features will both rotate with respect to their initial defined positions.
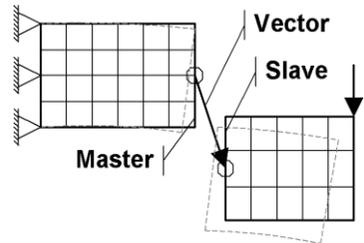
Constraint equation dependency across analyses / loadcases

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:
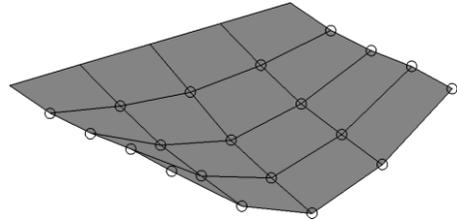
- ❑ **Guyan reduction eigenvalue analysis**
- ❑ Superelement analysis

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.
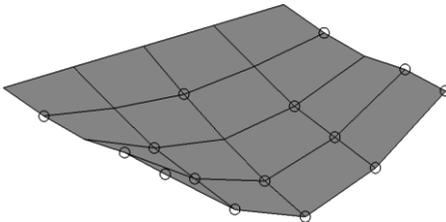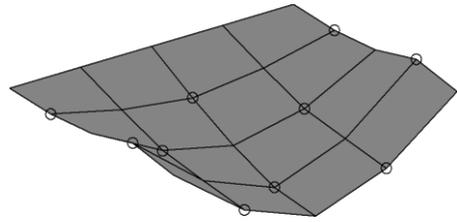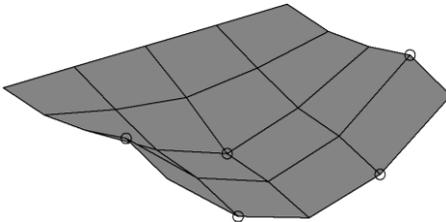
Full Subspace Iteration
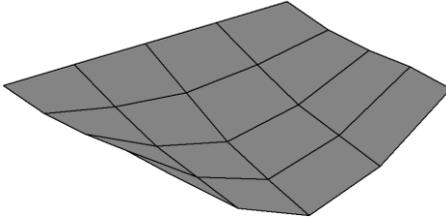
20 Masters

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. See **Isotropic/Orthotropic** for more information). If any damping attributes are specified, all damping must be defined and assigned using damping attributes; damping properties in material attributes will be ignored.

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.

Damping attributes affect the responses calculated from model shapes using the graph wizard or IMD loadcases. Note that damping attributes are not used in the calculation of responses by Solver in transient (step-by-step) dynamic analyses, in which case material damping must be used.

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.

Activation and Deactivation of Elements

The activation and deactivation of elements enables the modelling of a staged construction or demolition process, generally as part of a **nonlinear analysis**. For this, 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> Activate and Deactivate** menu item and are assigned to features (or elements in a mesh only model) and manipulated in the same way as other attributes. Deactivation of elements is also permitted in the first loadcase of a linear analysis without the need for a nonlinear licence or nonlinear control.

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 (that are still retained in the model) 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.

Stiffness reduction factor

This provides control over the amount that the stiffness matrix is reduced in magnitude when elements are deactivated.

Percentage to Redistribute

In a deconstruction situation a deactivate attribute controls the way in which internal forces in an 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.

Note that a member introduced into a structure may be a closing part e.g. a cast in-situ stitch; cables cut-to-length; parts with slotted holes to fit between two or more connections, or a prefabricated part such as a steel box girder or beam, or a precast segment.

- • 'Closing parts' have no shape or stiffness before they are introduced into the structure. In real life these parts fill the gap between and connect together two or more parts of a pre-existing structure. When they are added to a structure, end compatibility does not apply. Model features representing these members should be assigned a deactivate attribute with 'Percentage to redistribute' set to 100% and 'Stiffness reduction factor' set to 1E-15.

- Prefabricated parts have shape and stiffness before they are introduced into a structure. In real life these parts are created in isolation from the structure to which they will be added. When they are added to a structure, end compatibility applies. Model features representing these members should be assigned a deactivate attribute with 'Percentage to redistribute' set to 100% and 'Stiffness reduction factor' set to 1E-6.

In fully prefabricated structures, such as the erection of steel girders (not considering any subsequently cast concrete slab), at the end of the construction process they are as if they are "wished in place", although some members may have been through more onerous states of stress before reaching that final condition. To analyse this situation such members should not be supported until they have been introduce otherwise correct moments and deformations may not be obtained.

In most situations the specification of a 'Percentage to redistribute' value of 100% and a different 'Stiffness reduction factor' according to whether the member is a 'closing part' or a 'prefabricated part' should suffice. But in situations where additional control is required, and for lines only, inactive line control options may be of use.

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 option 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 <u>lines</u> 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 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.**
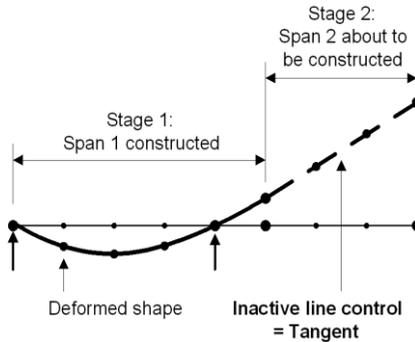
- If a deactivation attribute is inadvertently assigned to a surface any inactive line control setting will be ignored.

Inactive line control examples
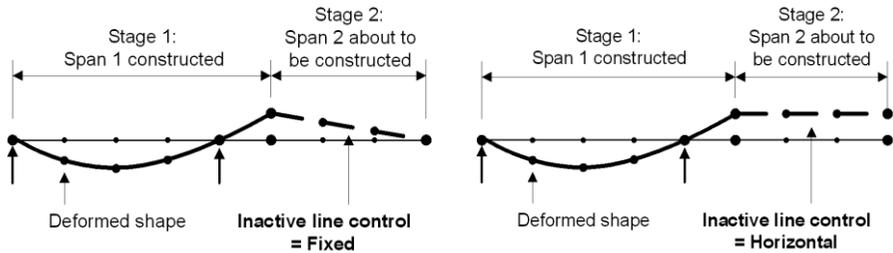
Fixed line control

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 and loadcases are not inherited between analyses, but they can be copied and pasted between analyses in the Analyses ⏰ Treeview.

Notes

- 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.

- With a forward construction method, supports should only be added when the activated members of a structure "land" on them, not before. Assigning all supports at the start of the analysis will result in incorrect deformations being achieved. When introduced into the model, the supports need to be in the right location. If this is the "as drawn" location, a 'Prescribed displacement' of zero should be used otherwise the supports will be introduced at the deformed node location. There may be occasions where the latter is required - it just depends upon the situation being modelled.

- 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
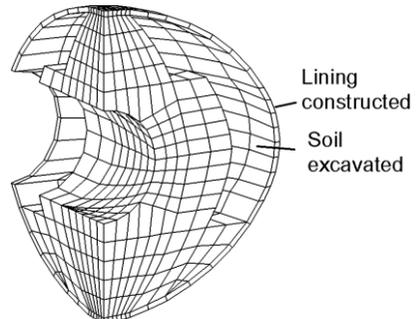
Excavation Stage 1

Top layer of soil deactivated and lining activated. Lining and soil elements duplicated in the model.
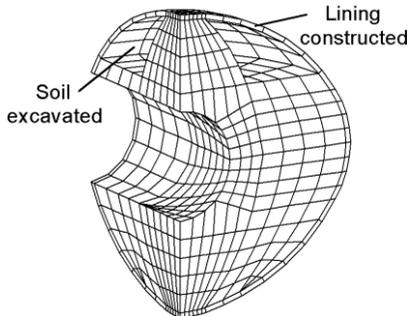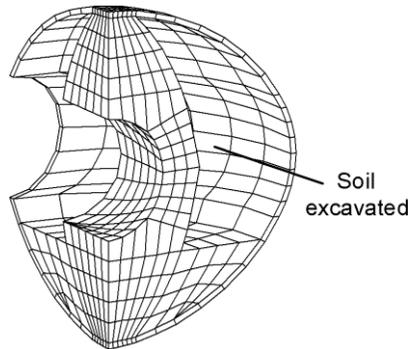
Tunnel Excavation Stage 2

Second layer deactivated as soil excavated. Surrounding lining elements activated
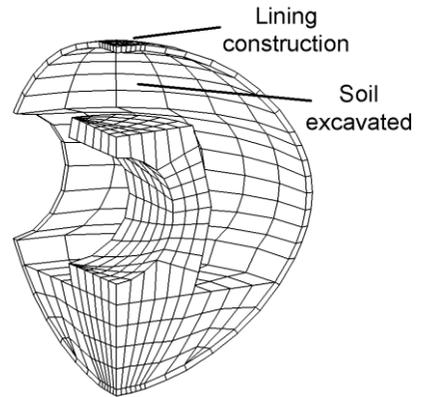
Tunnel Excavation Stage 3

Remaining second layer soil elements deactivated.

Tunnel Excavation Stage 4

Supporting soil pillar removed and top lining activated.

Lining
construction

Soil
excavated

Soil
excavated

Tunnel Excavation Stage 5

Final central soil column removed.

Soil
excavated
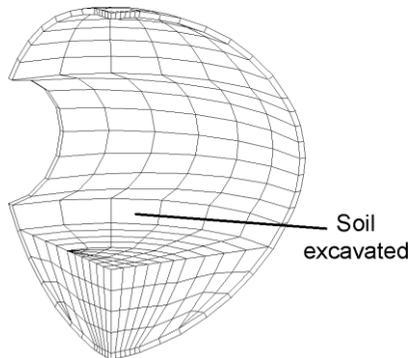
Equivalencing

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
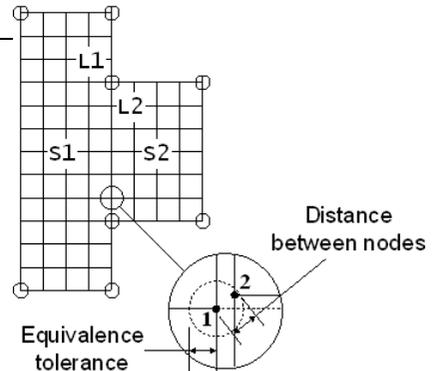
In this example, Surfaces 1 and 2 do not share a common boundary Line, therefore the nodes created on their common boundaries will not be joined and must be equivalenced. Node 2 will merge with node 1 if it lies within the equivalence tolerance.

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 **Tools> 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.
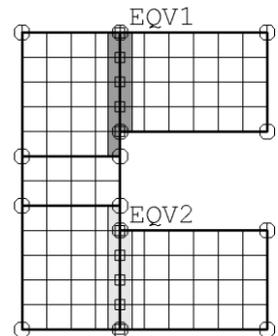
Summary

- 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.
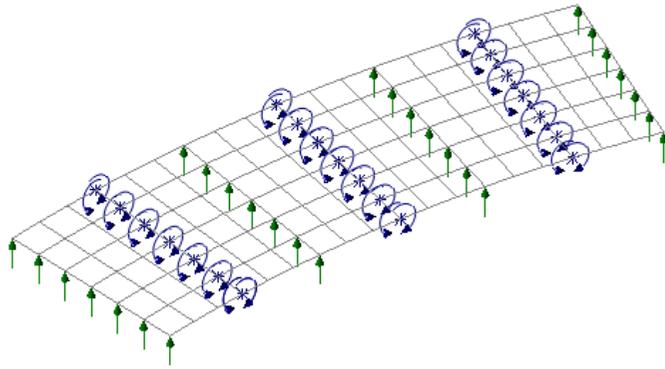
Note that Reciprocal influence attributes can be assigned to as many nodes or points on the model as necessary, but should only be assigned to those nodes or points for which they are applicable. For example, a Reciprocal influence attribute (defined for a negative displacement direction) should only be assigned to those nodes in a model for which a negative influence shape applies. For nodes where a positive influence shape exists a Reciprocal influence attribute with a positive displacement direction should be assigned. In general, when using Reciprocal influence attributes the influence shapes for nodes of interest should be viewed prior to any influence attribute assignments being made.

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 defined 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. Note that components that use absolute values are not valid since the principal of superposition cannot be applied to these components. As a result these components are not provided for selection.

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)
- ❑ **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 defined and 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 settings.

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).

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).

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.
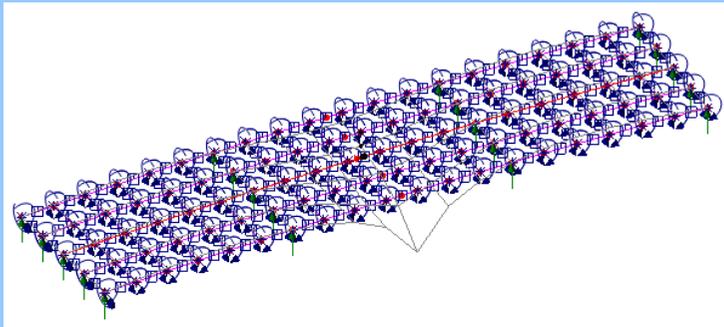
For grillages created outside of the grillage wizard a similar process would apply using repeated box selections to select all longitudinal members (and their lower order points) prior to assignment of an influence attribute to create every possible longitudinal influence, followed by repeated box selections of all transverse members (and their lower order points) prior to assignment of an influence attribute to create every possible transverse influence.
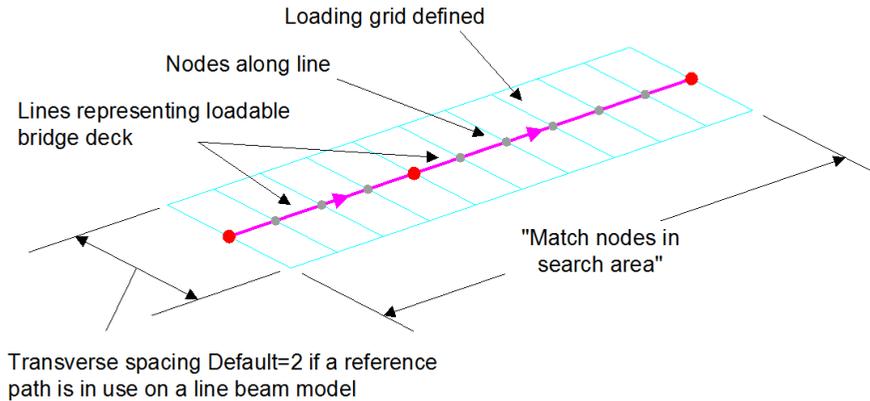
Using a Loading Grid

When carrying out certain types of Direct Method influence analysis (see Examples of Loading Grid use) it is beneficial to use a loading grid. A loading grid provides the means to define a loadable grid of points that can be used by an influence analysis as an alternative to using the loadable nodes or points inherently present in a beam, shell or plate model. For line beam analysis, where a geometric beam section represents a loadable top slab, loading grids would normally always be used. The use of a **reference path** defining the centreline of a loadable carriageway and a search area to restrict the loading to selected nodes of the model is recommended whenever a loading grid is used.

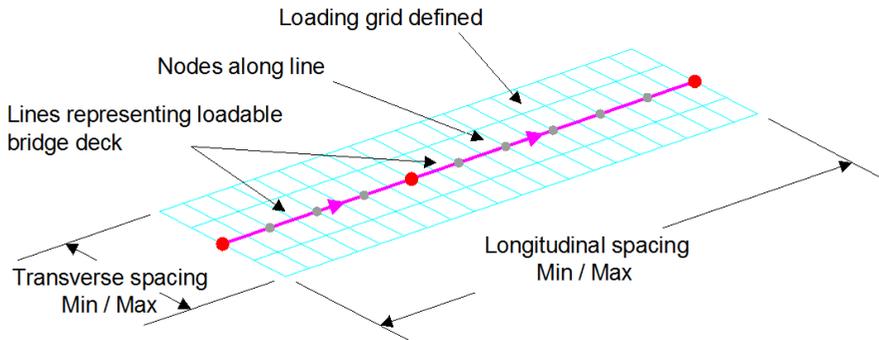The use of a loading grid will require the following to be specified:

❑ **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** - these control whether the loadable grid of points is based upon the nodes defined in a search area, or whether it should be based upon a grid of points that can be set to match the nodes in a search area (line beam models only). or be user-defined according to longitudinal and transverse values 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.

Note that manually-defined loading grids are not supported by the Rail Load Optimisation facility. Instead, for this option, 'grid' points are automatically created where the rails for each track layout intersect the elements representing the deck, and only those 'grid' points are loaded when carrying out an influence analysis.

Loading grid examples for line beam models

Loading grid defined

Nodes along line

Lines representing loadable
bridge deck

"Match nodes in
search area"

Transverse spacing Default=2 if a reference
path is in use on a line beam model

Line beam loading grid created by matching nodes in the search area.

Loading grid defined

Nodes along line

Lines representing loadable
bridge deck

Longitudinal spacing
Min / Max

Transverse spacing
Min / Max

Line beam loading grid created from either minimum or maximum number or spacing values

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:

- 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 or plate model.
- On a skewed grillage model where it is required to obtain results for longitudinal and transverse (skewed) members.
- 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).

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.
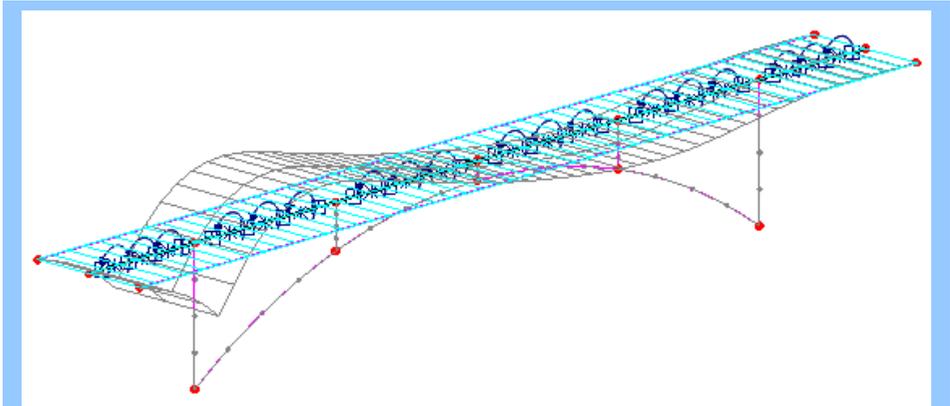
Note that to carry out vehicle load optimisation analysis, lines defining the width of the carriageway must also be defined and selected.

Naming and Visualisation of Assigned Influence Attributes (prior to Version 16)

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) depends three things: the type of influence Entity chosen; any element participation settings made when the attribute was first defined (for pre-Version 16 models only); and whether any manual (or automatic) selection of participating elements was done (pre-Version 16 models only).

The following notes generally apply to influence attributes defined and assigned to a model prior to a model prior to Version 16:

- ❑ 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. Orange arrows signify un-averaged results will be calculated. Blue arrows signify averaged results will be calculated.

Additional visualisation settings are accessed using the context menu for an influence attribute.

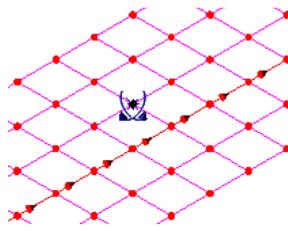# Element Visualisation Symbols (defined and assigned in Version 16 and above)

Influence attributes defined in and assigned to a model created in Version 16 or above are visualised the same on all analysis model types. No element participation symbols are visualised because all averaging of element results is done automatically according to settings made on the **Model Properties - Options (Averaging)** dialog.

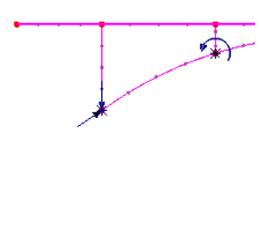| Plate/shell model	Grillage Model	2D/3D Frame Model
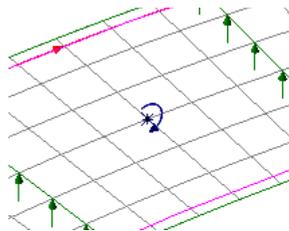
A blue arrow signifies averaged results will be calculated	Blue arrows signify averaged results will be calculated.	Blue arrows signify averaged results will be calculated.

Element Participation and Visualisation Symbols (initially defined and assigned in Version 15)

Influence attributes defined in and assigned to a model in Version 15 are visualised as they would have been when they were initially created (if the model is opened in a later version) because of the type of results averaging method that was specified <u>at the time</u> the influence attribute was generated. See the images that follow.

In addition, if influence attributes defined in a pre-version 16 model are re-assigned to nodes in a model created in later version the element participation and visualisation symbols will also be as shown on the following images. Note that even though the influence assignments have been made in version 16 or above, because the method of results averaging was defined as part of the attribute definition, settings made on the **Model Properties - Options (Averaging)** dialog will not override any averaging settings defined for these influence attributes.

Plate/shell model

Symbols used for an influence attribute defined to investigate Flexural Moment in longitudinal members with reference to a path.
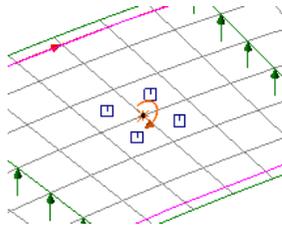
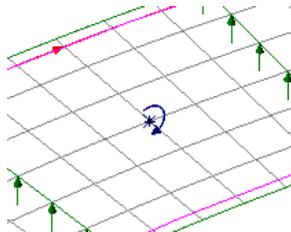| Version 15: "Separate influence for each connected element"	Version 15: "Average influence of all connected elements"	Version 15: "Selected elements only"
Elements identified by square symbols are participating at the node. An orange arrow signifies un-averaged results will be calculated.	No elements identified by square symbols signify that all connected elements to the node are participating at the node. A blue arrow signifies averaged results will be calculated	Only those selected elements identified by square symbols are participating at the node. A blue arrow signifies averaged results will be calculated.

Grillage Model

Symbols used for an influence attribute defined to investigate Flexural Moment in longitudinal members with reference to a path.

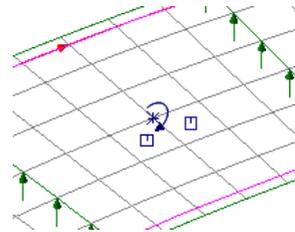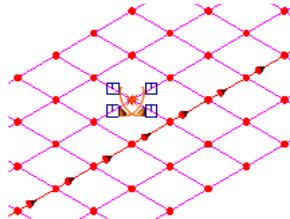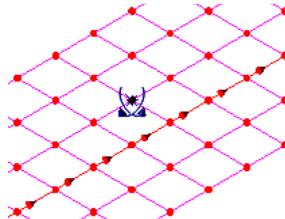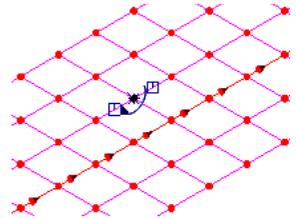| Version 15: "Separate influence for each connected element"	Version 15: "Average influence of all connected elements"	Version 15: "Automatically choose elements for averaging"

Elements identified by square symbols are participating at the node. Orange arrows signify un-averaged results will be calculated.	No elements identified by square symbols signify that all connected elements to the node are participating at the node. Blue arrows signify averaged results will be calculated. (This option is not normally applicable to grillages)	Only the longitudinal elements identified by square symbols are participating at the node. A blue arrow signifies averaged results will be calculated.

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.
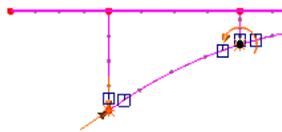
Version 15: "Separate influence for each connected element"	Version 15: "Average influence of all connected elements"	Version 15: "Selected elements only"
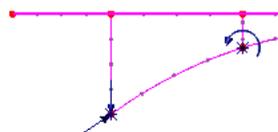
Elements identified by square symbols are participating at the node. Orange arrows signify un-averaged results will be calculated.	No elements identified by square symbols signify that all connected elements to the node are being averaged. Blue arrows signify averaged results will be calculated.	Only the elements identified by square symbols are participating at the nodes. Orange arrows signify un-averaged results will be calculated.
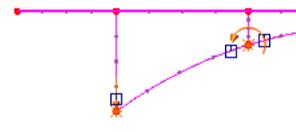
Identifying Influence Locations on the Model

The location on the model of an assigned influence attribute entry in the Attributes Treeview can be seen by using the **Select Assignments** context menu for that entry.

The location on the model of an influence analysis entry in the Analyses Treeview can be found by using the **Find** context menu for that entry. 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 still be solved with influences marked as 'not valid' and with unassigned influences present in the Analyses 🔍 Treeview.
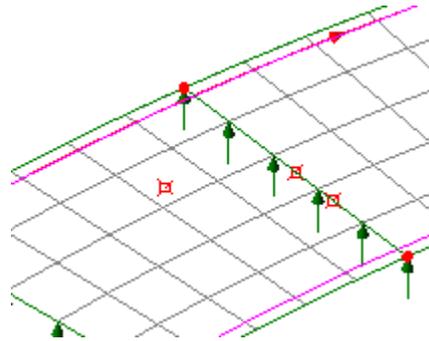
Solving an Influence Analysis

See **Influence Analysis**

Age

Age attributes are defined from the **Attributes> Age** menu item. Age attributes are used to define the age of an element at the response time in **timescale units** at which it first becomes active in an analysis. When using the concrete creep and shrinkage models (CEB-FIP, CHINESE, EUROCODE_2) the age from casting to activation can be defined, and the age after casting that shrinkage begins can also be defined. Note that it is assumed that the age that the element first becomes loaded coincides with the element age at activation.

❑ **Age at activation time** (Age at placement) The number of days/hours etc.between concrete being cast and the elements to which that concrete has been assigned being activated in the model (see activate and deactivate). For example, an element could be 40 days old but activated in the analysis on day 28. If no **activate and deactivate** attributes have been assigned then the age activation time will be at the beginning of loadcase 1.

❑ **Age at which shrinkage begins** The number of days/hours etc. between the concrete being cast and shrinkage being considered to have started. For Codes of Practice where autogenous shrinkage and drying shrinkage are handled separately, such as EN1992-1-1, this input refers to the age at which drying shrinkage begins; autogenous shrinkage is always taken as occurring from when the component is cast. For Codes of Practice where there is no such

distinction, this value is used to define the commencement age for all shrinkage.

Once defined and assigned to a feature all elements created by that feature are assigned the specified ages that apply.

Note that Modeller will always convert these ages to seconds when tabulating the Solver data file.

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.
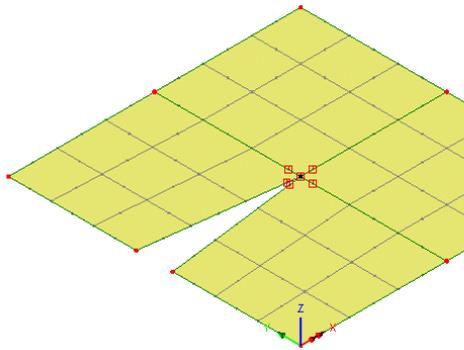
Crack tip attribute assigned to a point in a 2D model
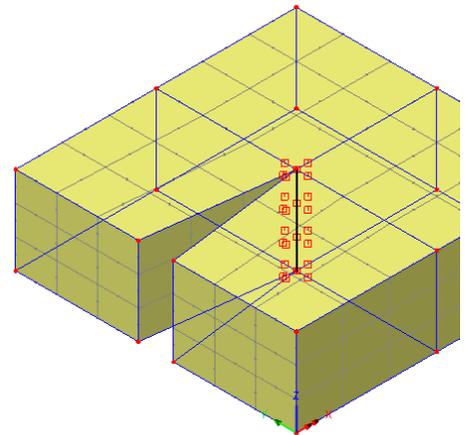
Crack tip attribute assigned to a line in a 3D model

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 **Radiation Surfaces** 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

Radiation Surface

Utilities > Heat Transfer
> Radiation Surface...

Controls heat transfer
from body to body over
large distances (where
radiation is dominant).

Thermal Surface

Attributes > Thermal
Surface...

Used to associated heat
transfer properties with
parts of the model.

Assign to
features

Thermal Gap

Utilities > Heat Transfer
> Thermal Gap...

Controls heat transfer
from body to body in
close proximity (where
conduction and
convection are dominant).

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.
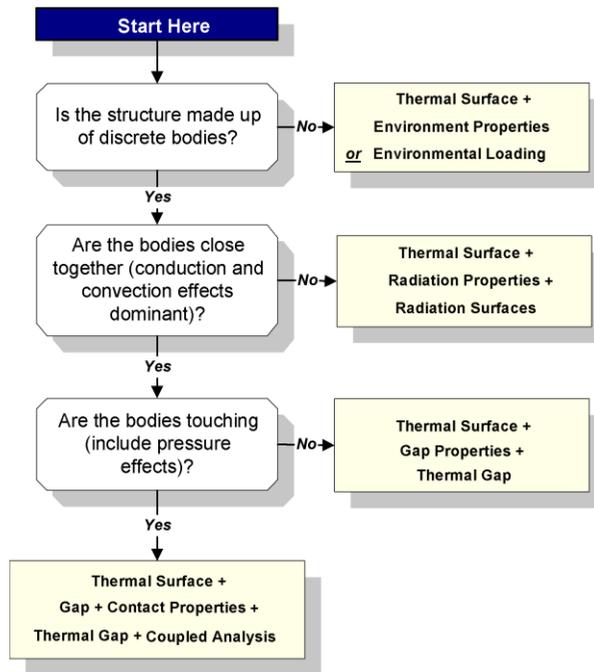
Environmental Nodes (LUSAS Solver data file only)

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 <u>on</u> 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.

Results Transformation Attributes

Results transformation attributes can be defined using the **Attributes> Results Transformation...** menu item.

If the elements in a model are orientated such that their local axes vary from one another (and note that 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 by assigning a results transformation attribute to all or selected features of a model. Multiple results transformation attributes may be defined and assigned to relevant features on a model.

Results may be transformed to the following directions:

❑ **Local axes of 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.

❑ **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**.

❑ **Reference path** Transforms nodal or element results relative to the angle or route of a **reference path**.

Once created a Results Transformation attribute is added to the Attributes 🎯 Treeview, in readiness for assignment to features of the model. Once solved, element results can be transformed by any of the above methods, or by assigned results transformation attributes., by pressing the Transform Set button on the relevant results properties dialogs.

Notes

- Different model drawing layers can each have different Results Transformation settings.

- Beam and bar elements always have their local x direction along the length of the element.

- Beam element results can only be transformed rotationally about their axis.

- 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 when viewing results contours, when it has been explicitly chosen on the contour properties dialog.

Inspection locations

An inspection location is a user-defined position of interest on a model for results extraction / viewing purposes. Inspection locations can be defined and assigned to features in a model either before or after a model has been solved, and are saved when a model is saved. Model features can have multiple inspection locations assigned to them. Inspection locations can report results on elements that belong to the assigned feature, and also for those elements that belong to topologically connected features.

Appropriate results can be viewed for any defined inspection locations by selecting Inspection Locations from drop-lists on a relevant results layer properties dialog, or for individual chosen locations. See **viewing results at inspection locations** for more details.
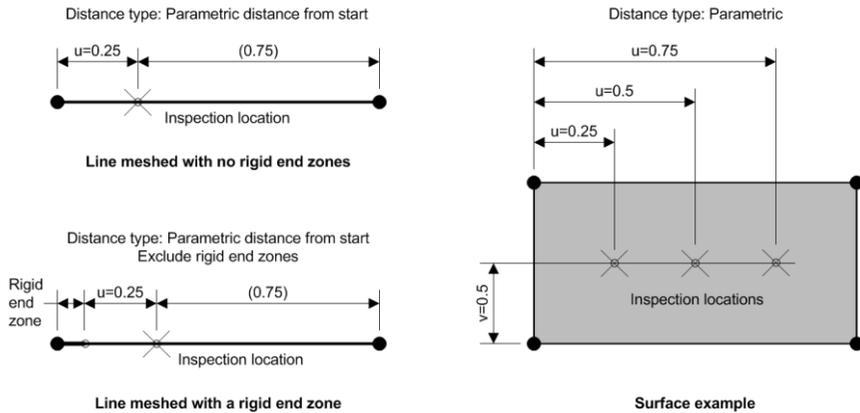
Inspection locations can be created in three ways:

❑ **By definition and assignment** - by creating an inspection attribute for a point, line, surface or volume by using the **Attributes > Inspection Locations...** menu item.

❑ **By cursor** - by defining an inspection position 'freehand' at a point, along a line, or in the plane of a surface by using the **Tools > Inspection Locations > By cursor** menu item.

❑ **By selection** of nodes or points (only) prior to using the **Tools > Inspection Locations > By selection** menu item. This facility defines a new inspection location for each mouse click, with actual distances calculated, and assigns it to the feature that was clicked on.

By definition and assignment of an inspection attribute

An inspection location attribute can be defined for assignment to a **Point**, **Line**, **Surface** or **Volume**

❑ **Point** No additional input is required

❑ **Line** For assignment to a line a distance type of **Parametric from start**, **Parametric from end**, **Actual from start**, or **Actual from end** can be stated. These options create an inspection location at a specified parametric distance between 0 (start of line) and 1 (end of line), or actual distance (in model units from the start of a line) along a line in the local coordinate system of any line to which the attribute is assigned. For lines multiple inspection locations can be defined either as one distance per grid row, or several distances can be entered together in a convenient shorthand format. For example, entering "0.5, 2@0.25, 3@0.1" is equivalent to entering "0.5, 0.75, 1.0, 1.1, 1.2, 1.3".

❑ **Surface** or **Volume** For assignment to a surface or a volume a distance type of **Parametric**, or **Actual** can be stated. These options create an inspection location at a specified parametric or actual (in-plane) distance in a simple planar triangular or quadrilateral surface, or within a volume with respect to the local coordinate system of the feature.

When a line mesh containing one or two rigid end zones has been assigned to a line it is possible to specify whether the parametric or actual distance specified to an inspection location from (say) the start of a line, starts from the start of the line itself, in which case **Exclude rigid end zones** would be underlined_unchecked, or starts from the end of the rigid zone furthest from the start of the line, in which case **Exclude rigid end zones** would be checked.

Examples of inspection location attribute defined for lines

Example of inspection location attributes defined for a surface

On completion, an inspection location attribute is automatically created in the Attributes 🎯 Treeview, comprising a user-defined name with the addition an automatically generated description to indicate for which feature the inspection location applies.

Assignment

Inspection location attributes can be assigned to one or more selected points, lines, surfaces or volumes within a model using either a drag and drop technique or the context menu options present for each attribute, that are available by right-clicking on an attribute name. Assigned inspection location attributes will move and copy with their assigned geometry, just like other assigned attributes.

Note that it is possible to assign valid inspection locations in such a way that the assignment is invalid. For example, to define a line inspection location, using an actual length of 10 units, and assign it to several lines, one of which is less than 10 units in length. Similar issues are possible with irregular surfaces, or surfaces with holes – the inspection location can be outside the boundary of the surface. Such inspection locations simply show an 'invalid' symbol and do not display any results.

Inspection locations by cursor

These can be defined by using the **Tools > Inspection Locations > By cursor** menu item. This facility is useful for defining or investigating a small number of inspection points.

An inspection location may be defined 'freehand' at a position along a line, or in the plane of a surface by use of the cursor, provided the indicated feature is within the search tolerance in use.

- For points, if the cursor position when the mouse is clicked is within search tolerance of a point, that point will be used for the search location.

- For lines, an inspection location will be created at the nearest position on the line to the cursor position.

- For surfaces, an inspection location will be created at the coordinates of the cursor position when the surface was selected.

In all cases an inspection location is automatically created and assigned to the feature found under the cursor click, and the appropriate local coordinates that are calculated are used in the Inspection location attribute's name as held in the Attributes Treeview.

Inspection locations by selection

These can be defined by using the **Tools > Inspection Locations > By selection** menu item.

Following selection of suitable points and /or nodes:

- A single inspection location attribute is created and assigned to all points in the selection (they all share the same attribute).

- An inspection location attribute is created for each and every selected node that is found. The type of the attribute, and the coordinates used within its name, reflects the type of the feature on which the node appears – e.g. for surface elements, the attribute will be of type surface, and local x,y coordinates will be automatically calculated and used to define the attribute.

Visualisation of assigned inspection locations

Once defined, inspection location attributes are added to the Attributes Treeview. Inspection locations can be deleted from this treeview singly or as part of a multiple selection.

By default, correctly assigned inspection locations are identified in the view window by this symbol: ✳. Invalid assignments or assignments requiring attention are identified with this symbol: ⊠. Note that the symbols are drawn by the Attributes layer, which has to be drawn last (bottom of the list) in Layers Treeview for the symbols to be seen in certain modelling situations.

Inspection location symbols can all be turned on and off from the Inspections locations tab of the Attributes drawing layer properties dialog. Inspection location symbols can all be turned on and off individually by using the context menu for the Inspection location entry in the Attributes ⚛ Treeview and choosing **Visualise assignments** menu item.

The feature on the model to which an inspection location attribute has been assigned can be found by using the context menu for the inspection location entry and choosing **Select assignments** menu item (to highlight the feature(s) to which it has been assigned.

Changes to mesh after assigning inspection location attributes

If a model is re-meshed or has its geometry edited affecting the mesh, assigned inspection location attributes will remain at the same location and be marked with an appropriate assignment symbol. Inspection locations assignments that are no longer valid following a change to model geometry, perhaps, will have a 'not valid' 🚫 symbol alongside their name in the Attributes ⚛ Treeview. These locations are effectively 'pending' (requiring correction or removal) and no results extraction will be possible for these locations. Note that models can still be solved with inspection locations marked as 'not valid'.
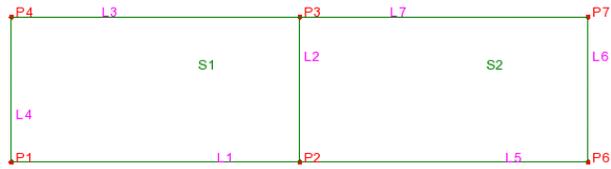
Viewing results at inspection locations

Results can be viewed for any defined inspection locations by selecting the Inspection Locations option from the relevant drop-list for a chosen results entity and component on a Values or Contours layer properties dialog, or **Print Results Wizard**. For inspection locations element averaging is always attempted. Where averaging is inappropriate, multiple values will be displayed. Averaging settings can be modified on the Options tab of the **Model Properties** dialog.

Values reported

Inspection locations can report on results for elements that belong to the assigned feature, and also those elements that belong to topologically connected features. For example at a corner junction between two surfaces, such as at P2 on the image that follows, (at least) six different inspection locations could, in practice, be defined at that location - one assigned to the point itself, one assigned to each end of the three lines meeting at that point, and one assigned the corner of each of the two adjoining surfaces. These inspection locations are all at the same coordinates in space, and can all report upon values for the same elements. These values include displacement, reactions, and other intrinsically nodal quantities at the node underlying the common point. Additionally, assuming the lines are meshed, the results of the three different beams will also be available – one for each line. Note that, in this particular example, these are not averaged, because the beams form a junction. Additionally, assuming that the surfaces are meshed, and no **averaging rules** are broken (results cannot be

averaged across different , then a single shell result will also be available – the average of contributions of the corner elements of the two surfaces.

When viewing values at inspection locations, only unique values for each nodal result will be displayed from the multiple values that may be present. For beam tstresses, values for specified fibre locations will be displayed, but these are drawn at nodal locations and not at fibre locations.
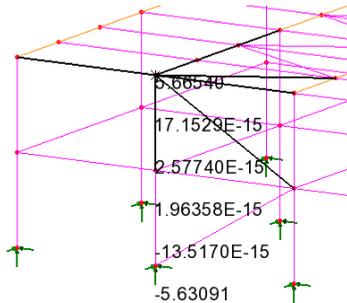
Inspection location results displayed for a single point in a model, showing a unique value for each highlighted line (representing a beam) connecting to that point.
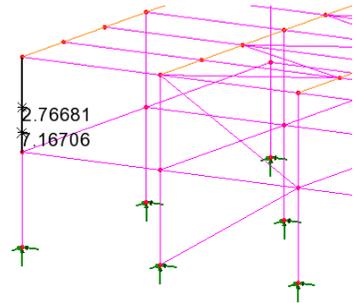
Inspection location results displayed for a line, showing a value for each inspection position defined on the highlighted line.

So, in summary, unique results are displayed at defined inspection locations for all continuous (nodal) results. Additionally, element results are displayed for elements meshed on to the assigned feature. Additionally results are displayed for elements meshed on topologically connected higher-order (parent) features. However, results are not displayed for elements meshed on topologically connected features of the same order (siblings).

Contours

When viewing contours, coloured blobs are drawn at those inspection locations that have only a single value (that is for continuous results, or where averaging has been successful). At inspection locations that resolve to multiple values (because averaging is inappropriate), no coloured blob is shown, and instead an 'invalid' symbol appears.
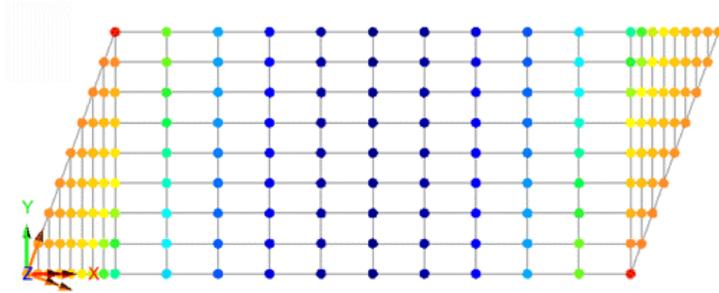
Influence

Influence locations are treated by Modeller as "additional" inspection locations, so, when examining influence results at inspection locations, results will be consistently presented at all locations to which influence has been assigned. Where the influence definition indicates the element(s) of interest, other elements are not considered in the computation of such results.

Graph wizard

When plotting graphs an inspection location assignment can be chosen from a list, rather than typing in a node number. This means that only those inspection locations that are located exactly over a node are available for graphing.

Report wizard and Print Results Wizard

When inspection location results are output to a **model report** or listed by the **Print Results Wizard** the inspection locations are referred to by both their attribute name and the global coordinate that their assignment resolves to. Where the inspection location is at a point, the point name is given instead.

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.
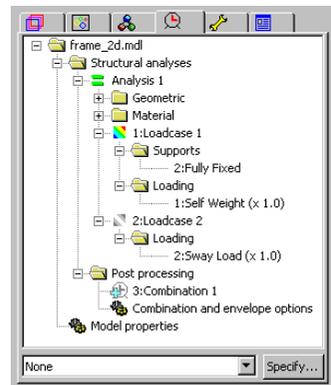
Note that nonlinear analyses produce results loadcases for each iteration used to achieve a convergent solution. Combinations and envelopes of loadcase results can be created to look at combined or maximum / minimum effects.

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. Moving of loadcases should be done with care because if a single loadcase, that was defined outside of a range of loadcases, is moved in the Analyses Treeview to a position within a range of loadcases its load factor will be retained in addition to any load factor already specified for the range of loadcases in which it now sits, and both load factors will be applied during an analysis.

Loading variation with time

For analyses in which the load varies with time (or increment number in the case of nonlinear analysis) **load curves** may be used. When using load curves all loads must be assigned to a load curve instead of a loadcase, but all other loadcase dependent attributes (support, slidelines etc.) and analysis control are assigned to loadcases in the usual way. 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 latter 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 **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.

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.

Solving loadcases

Loadcases are solved in the order in which they are listed within each analysis entry in the Analyses ⬛ Treeview.

Setting and Changing the Active Loadcase

Loadcase dependent attributes and results for a loadcase can only be visualised by setting that loadcase active. Each view window can have a different active loadcase specified.

The active loadcase is set within the Analyses ⌖ Treeview by selecting **Set Active** from the context menu for that loadcase. The active loadcase can also be changed by dragging and dropping a loadcase name into a view window.

A loadcase icon changes from grey to coloured when 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 Results for loadcases

Results can be viewed for 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**

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. For example, to evaluate the dynamic response of a pipe to an increase of pressure over a given period, the load curve factor would consist of the variation of pressure with time. Or, in a transient thermal analysis, the environmental temperature loading on specific surfaces (say) of a dam may change with time.

Creating Load Curve Entries

New load curve entries can be added to an existing analysis entry in the Analyses ⌖ Treeview in the following ways:

- By creating a load curve from the **Analyses> Load Curve**menu item.

- By right-clicking on an 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 for Structural and Thermal analysis

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. Specifying a **Termination time** cuts-off the variation-generated load curve loading at that time. Specifying a **Sampling increment** samples along the time axis, and uses those values and the corresponding load curve factors to define the load curve output written to the Solver data file.

For each of the above options:

- ❑ **Activation time** Specify at what time along the defined load curve the corresponding value of loading for that time will be applied.
- ❑ **Scaling factor** This scales the loading that is assigned using the defined load curve by factor 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.

Load curve parameters

Load curve parameters are illustrated in the following image. Note that in a transient or time based analysis the x-axis represents time. In a nonlinear analysis, the x-axis represents load increment. In a fourier analysis, the x-axis represents angle. Note that, for a standard curve, specifying a mean amplitude shifts the defined load curve in the y-axis direction.

User-defined load curve   Standard curve   Variation-based
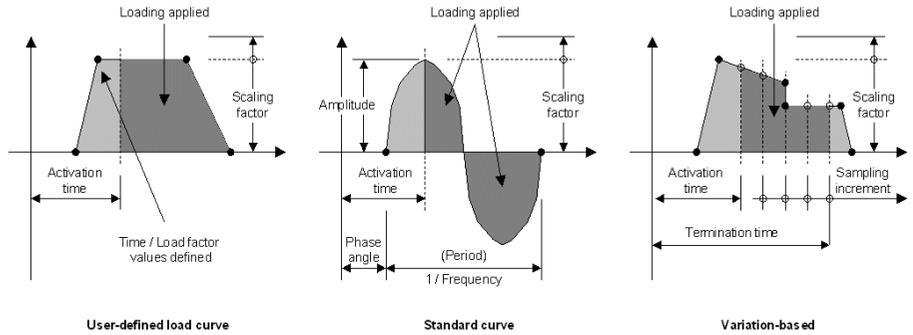
Defining Load Curves for Hygro-thermal analysis

For a structural analysis, the value of the load curve at the end of the current load step is used to determine the magnitude of the load. For a transient hygro-thermal analysis this is not suitable and an energy equivalent load curve with a constant value during each time step should be used instead so that any step reduction would not decrease the accuracy of the equivalent heat. Similar considerations are valid for a load curve applied to environmental temperature, relative humidity and water vapour mass transfer coefficients.

Experimental curve   LUSAS energy equivalent load curve
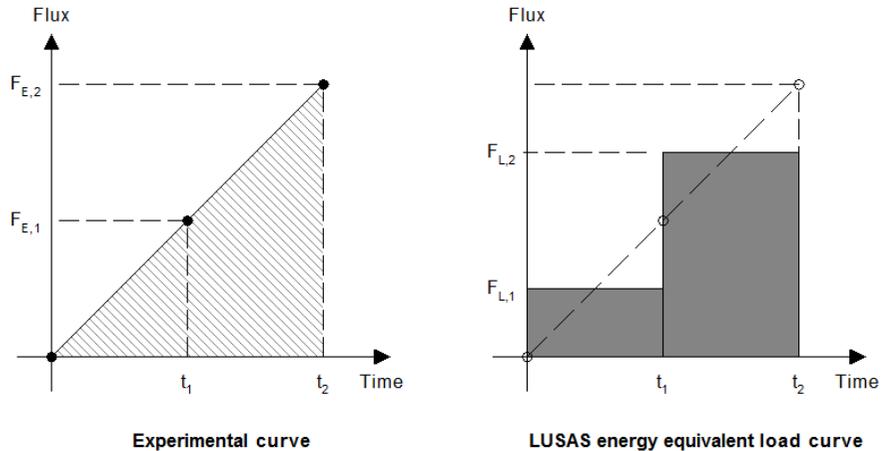
For time steps t_1 and t_2 the heat provided by the experimental curve is equal to the area under the curve for the time interval considered. LUSAS assumes that the load has a constant magnitude during the time step, so the load curve that should be used in LUSAS needs to provide the same heat energy during the time step:

$$\tfrac{1}{2} F_{E,1} t_1 = F_{L,1} t_1 \qquad \text{hence} \qquad F_{L,1} = \tfrac{1}{2} F_{E,1}$$

$$\tfrac{1}{2}\left(F_{E,1} + F_{E,2}\right)\left(t_2 - t_1\right) = F_{L,2}\left(t_2 - t_1\right) \qquad \text{hence} \qquad F_{L,2} = \tfrac{1}{2}\left(F_{E,1} + F_{E,2}\right)$$

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.

Assigning Loading to a Loadcurve

After defining a load curve for an analysis, a load is assigned to a load curve instead of to a loadcase. All other loadcase dependent attributes (support, slidelines etc.) and analysis control are assigned to a loadcase. 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 start from t=n the load curve is assumed to be zero when t<n.

For selected hygro-thermal load types, if more both components of the specified load are non-zero, each component should be applied in a separate load case. See **Hygro-Thermal loads** for more details.

General Load Curve 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 **Analyses> 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 provide a particular functionality, such as to produce a graph or model report for example. Utilities generally create entries in the Utilities Treeview.

Utilities are added to the Utilities Treeview. in creation order, but can be re-ordered by using the **Sort by name** context menu item accessed from the Utilities tab.

Details of the features and use of many of the utilities listed below are provided in other relevant chapters of the manual. Transformation utilities, for example, are described in Chapter 3: Geometry in the Moving, Copying and Sweeping Geometry section. The remainder are described in this chapter.

The complete list of LUSAS utilities (accessed from the **Utilities** menu) is shown below.

- ❑ **Transformation** - used in the process of moving, sweeping and copying of geometry.
- ❑ **Heat transfer** - specification of thermal gap properties and radiation surfaces.
- ❑ **Variation**- ways of varying attributes over features.
- ❑ **Reinforcement** - defining **line reinforcement** and **section reinforcement** details for line beams
- ❑ **Reference Path** - defines a route through the model that provides a concept of distance for positioning multiple varying sections or applying vehicle load patterns.
- ❑ **Response Spectrum** - defines either a user-defined or a codified response spectrum for use with an IMD loadcase.

- ❑ **Background Grids** - grade the mesh pattern locally when irregular surface meshing.
- ❑ **Objects to drape** - for use with modelling laminated composites.
- ❑ **Design Factors** - assess the reserve strength capacity of a component or structure.
- ❑ **Graph Wizard** - plot results on x,y graphs.
- ❑ **Graph Through 2D** - slice a model and plot a graph based upon the intersection of the elements sliced.
- ❑ **Slice Resultants Beams/Shells** (Analysis type 3D only) - creates beam equivalent forces and moments along a beam from a slice section.
- ❑ **Print Results Wizard** - printing of selected results to a tabular listing format or a file, or for inclusion in a model report.
- ❑ **User Defined Results** - create results components from user-defined expressions.
- ❑ **Prestress** - define **tendon properties** and **tendon profiles** for **prestress loading**. This option is only present for LUSAS Bridge and LUSAS Civil & Structural software only.
- ❑ **Layout Grid** - used to define a regular or specified set of construction grid lines to aid with defining geometric model features.
- ❑ **Notes** - Plain text notes relating to a model can be written and saved for inclusion in a report.
- ❑ **Reporting** - view a **one-click report** summary, or build a new **model report** containing chapters of model and results data.

The Heat transfer and Objects to drape menu items are only listed if an appropriate license is in use.

Copying utilities between models

Utilities can be transferred between models. See **Transferring Data Between Models.**

Utilities symbols explained

See **Treeview Panels and Icons** for details of all symbols used in the Utilities 🔧 Treeview.

Variations

Variations allow parameters in attributes to be varied over the assigned geometry by defining the manner in which the parameter will vary. They are defined using the **Utilities > Variations** menu item, and are saved 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.

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.

Load curves can be used to describe the variation of 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.

Variation types

- ❑ **General Field** variations are 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 including Points (if global coordinates are used). Variations on volumes are limited to field variations.

- ❑ **Line** variations take two forms: interpolation variations and function variations. The former are 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. Line variations can be applied to any Lines including lines forming the boundaries of Surfaces.

- ❑ **Surface Function** variations are expressed as symbolic functions in terms of the parametric coordinates of a feature. The distances to be specified are the local parametric u and v coordinates.

- ❑ **Surface Boundary** defines values by specifying variations around the Surface boundary Lines.

- ❑ **Surface Grid** defines a grid of values in Surface local x and y directions.

- ❑ **Profile** defines variations that allow modelling of the variation of (typically) material properties at any location in a model. Normally used with soil modelling.
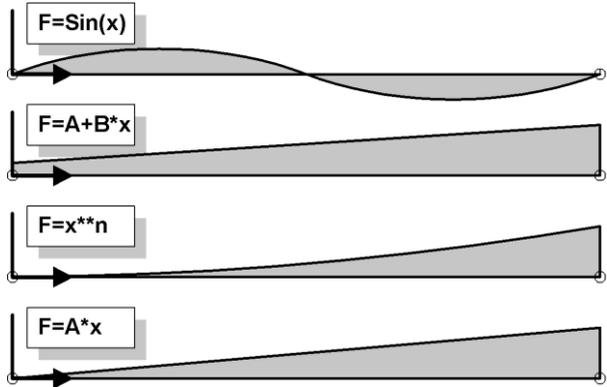
General Field Variations

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.

F=Sin(x)

F=A+B*x

F=x**n

F=A*x

For example, a field expression in Cartesian coordinates would typically be:

```
-9.81*y
```
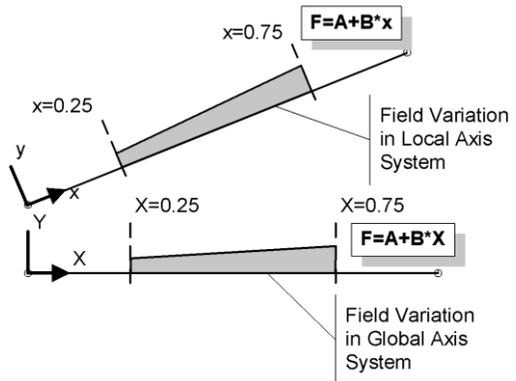
and in cylindrical coordinates:

```
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

$+$ $-$ $*$ $/$ $\wedge$

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)
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.

This example shows a variation in terms of the global Z axis ordinate with minimum and maximum cut-offs at specified Z ordinate values.
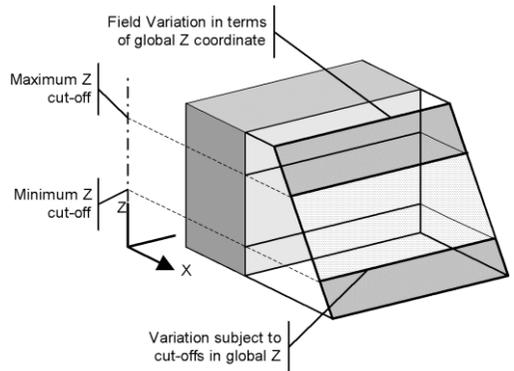
Case Study. Applying Hydrostatic Loading

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.
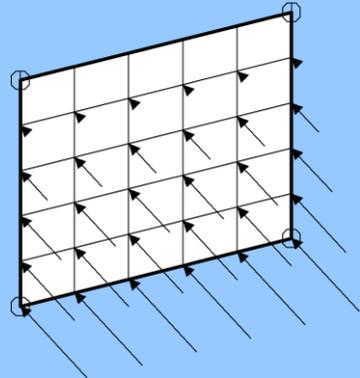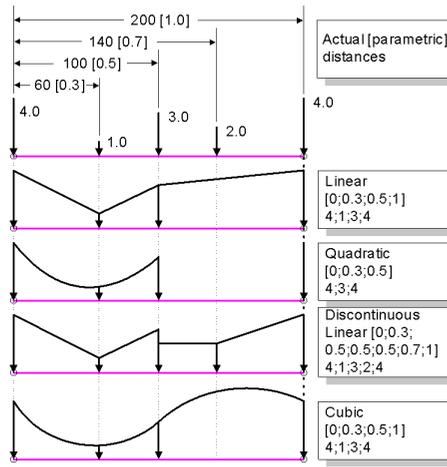
Line Variations

Line Interpolation options

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.
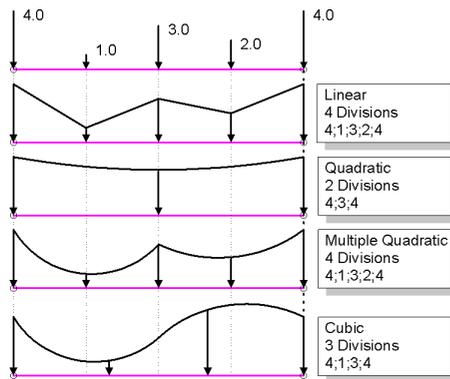
❑ **By Unequal Distances** defines values at specified distances along a Line. The distances can be entered as actual or parametric values. The actual value used will be interpolated at the appropriate distance between these values using the interpolation method specified.

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.
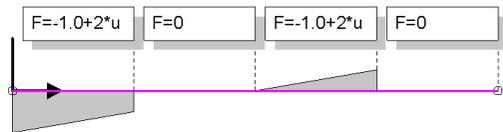

▪ **By Unequal Distances [in u]** defines a series of parametric or actual distances, and a set of functions. The distance specified is the starting position for the function
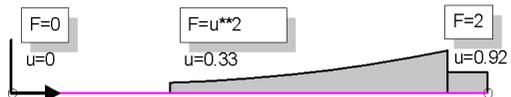
associated with it. Each distance must have an associated function specified. To enter a maximum cut-off position, associate a zero function with it. In this example, a parametric distance of 0 is associated with the value 0.0, a parametric distance of 0.33 is associated with u**2 and a parametric distance of 0.92 is associated with the value 2.0

Surface 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.


Variation Max(4,10*u)
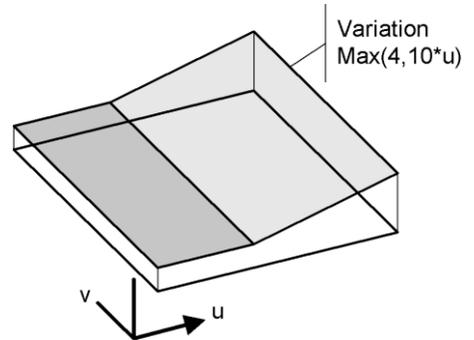
The example shown here defines a variation using the function **max(4,10*u)** in terms of the local Surface x direction parametric distance. The **max** function takes two arguments and returns the maximum of both arguments. In this case, 4 is the maximum value until u exceeds 0.4.

Surface Boundary Variations

These define 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. Surface boundary variations are affected by individual Line directions, so care should be taken to ensure line direction are appropriate. See example that follows.

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.
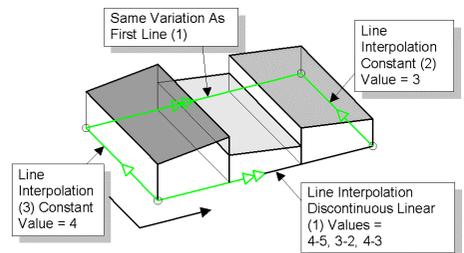
The second and fourth Lines in the Surface definition use constant interpolation variations. The variation sense is denoted by double and single arrows shown on

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 Grid Variations

These define a grid of values in Surface local x and y directions. For interpolation by grid the interpolation order may be constant, linear, quadratic or smoothed (cubic). 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.
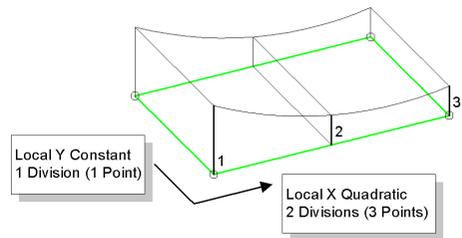
- **Smoothed (Cubic) vs. Linear Surface grid variation**

  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.
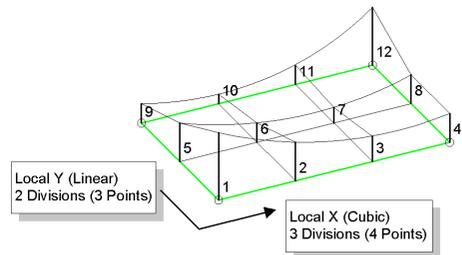
Profile Variations

Profile variations allows modelling of the variation of (primarily, but not exclusively) material properties at any location in a model. A single profile variation is defined by distance against value, however, any number of profiles in-plane or out-of-plane may be grouped to form a profile set variation. When used to define particular material properties in a supported material model, material properties for elements in any region of a model can be calculated by LUSAS Modeller by interpolating between the distance and values defined for each element between each profile variation location. A typical use is modelling the variation of soil properties with depth, as obtained by borehole data.

A profile variation is defined by entering value against distance data that describes the profile.
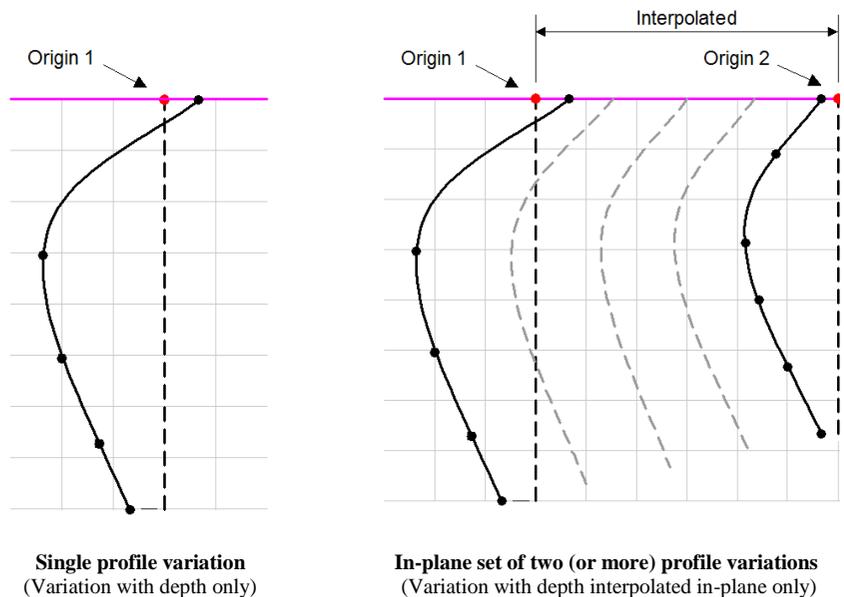
- For a 2D analysis category any number of profiles can be specified together with the x,y coordinates for each profile origin, the z coordinate is not shown and defaults to zero.

- For a 3D analysis category x,y, and z coordinates must be specified for each profile origin. The same profile variation may be used any number of times in the profile set definition but no profiles can have the same origin.

Once referenced by an assigned material, profile variations may be visualised. Contouring of materials using profile set variations is done on an element node basis. Materials not using profile set variations are contoured on an element basis.

Note that profile variations cannot be used with temperature dependent materials.

Profile variation types and calculations made

When used to define particular material properties in a supported material model, material properties for elements in any region of a model are calculated by LUSAS Modeller by interpolating between the distance and values defined for each element between each profile variation location. The following images show how properties are calculated with respect to the profile variation types shown.

Single profile variation
(Variation with depth only)

In-plane set of two (or more) profile variations
(Variation with depth interpolated in-plane only)
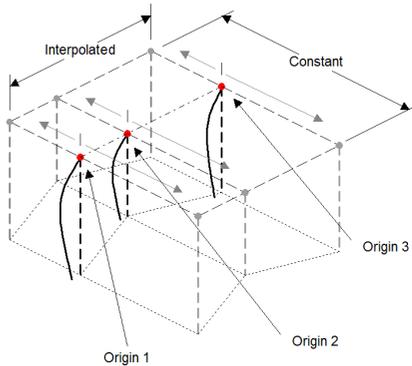
In-plane set of profile variations in a 3D model
(Variation with depth is only interpolated in the plane of
the profile set) Additional profiles are automatically
generated out-of-plane to edge of model. (Variation with
depth is constant in this out-of-plane direction.)
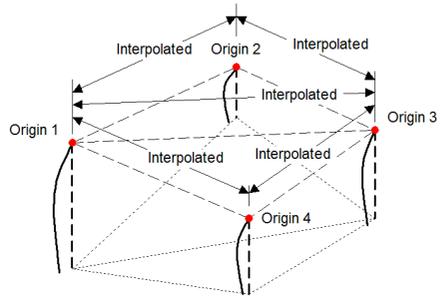
Out-of-plane set of profile variations in a 3D model
(Variation with depth is interpolated in 3 dimensions)

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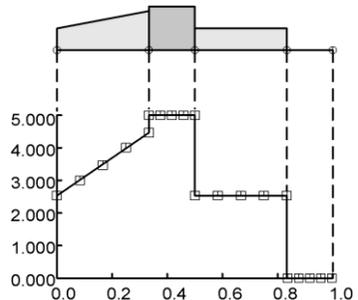 points at which to sample the variation can be specified. A factor may be applied to the variation values before the ordinates are calculated. This example demonstrates the graphical visualisation of a discontinuous Line interpolation variation.

Variation Notes

❑ If a variation is not specified, the parameters within an attribute will be constant over the geometry to which the attribute is assigned.

❑ Global coordinates can be used for any LUSAS feature/object (points, lines, nodes etc). Local coordinates can be used for all LUSAS features/objects except points.

❑ 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.

❑ 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).

Section reinforcement

Section reinforcement defines the bar arrangements to be provided in a member's cross-section. Section reinforcement is a utility object stored as an entry in the Utilities Treeview and cannot be assigned to lines in the model directly. Section reinforcement utilities are referenced by **line reinforcement** utilities (that define the length over which each section reinforcement utility acts along a line representing a reinforced concrete member.)

The section reinforcement dialog can be accessed from a reinforcement line dialog (by selecting the 'New' option in the reinforcement section droplist) or by using the **Utilities > Reinforcement > Section** menu item. Section reinforcement can be defined for any standard or arbitrary shape, with or without voids. Tapering members are also supported.

Section reinforcement is available for use with selected Bridge and Civil software products only, and for licences that support **RC frame design**. A model **analysis category** of 3D must also be in use.
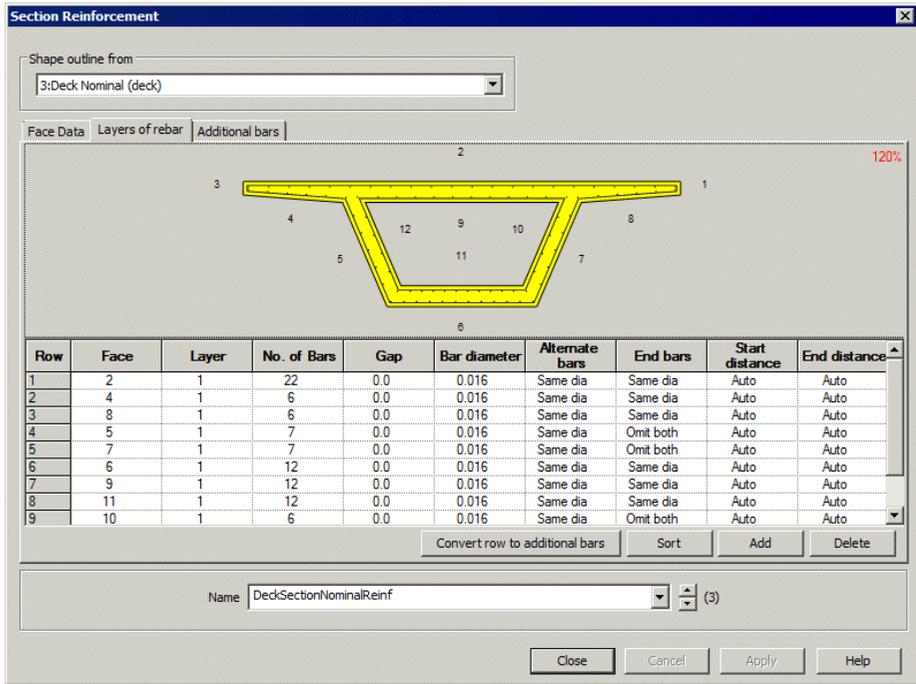
Defining section reinforcement

The section reinforcement dialog comprises three tabbed pages that are used to define the reinforcement arrangements for a concrete member's cross-section. This requires stating the concrete faces to be reinforced, the cover and allowance for links, and the layers and positions of reinforcing bars.

❑ **Shape outline from** A predefined geometric line attribute can be selected to assist in the definition of section reinforcement. The coordinates of the lines defining the section shape of the geometric line attribute are used to automatically determine and number the faces of the beam cross-section. Faces are always numbered anti-clockwise on the visualised section. Layers of reinforcement are defined with reference to each numbered face. Additional bars are defined with respect to the beam axis.

Notes

- In specifying a shape outline, the chosen **geometric line attribute** is not explicitly connected to the reinforcement section, it merely provides a visual reference for the face numbering in use and reinforcement arrangement being defined for that section type. This means that a section reinforcement arrangement for one section shape may also be applied to other geometric attributes of a similar shape. For example, section reinforcement could be

defined for a beam that is 400mm x 400mm and also be used to reinforce other sections of a similar shape, such as 400mm x 600mm, or 300mm x 600mm, by including the appropriate line reinforcement utility in unique geometric line attributes. This also makes the system effective / efficient for tapering sections.

- Deleting a geometric line attribute from the Attributes Treeview that is referenced by the section reinforcement utility will not make the section utility invalid. However, no visualisation of the section type on the dialog is possible until a geometric line attribute is selected again from the 'Shape outline from' droplist.

- In all cases covers, distances, bar diameters, and spacings are to be stated in model units.

Visualising section reinforcement

A section formed from the shape outline selected and the reinforcement as it is defined is shown on the dialog. Section reinforcement can be inspected by zooming and panning the visualised section. 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.

Reinforcement terms used

The following image explains the main terms used on the section reinforcement dialog:
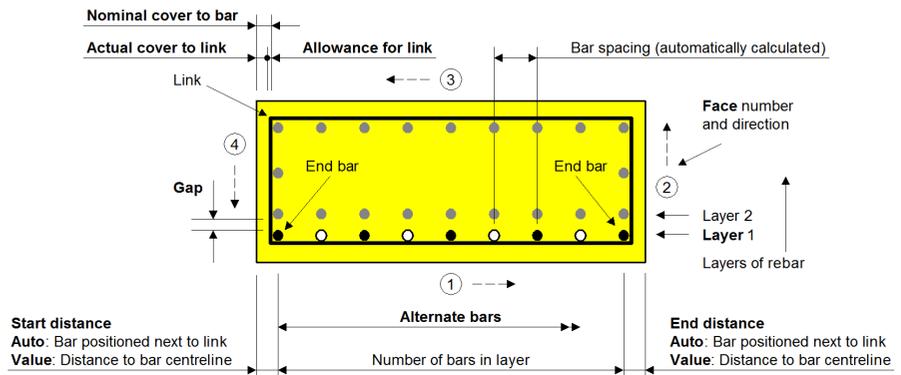
Reinforcement terms used

Face data
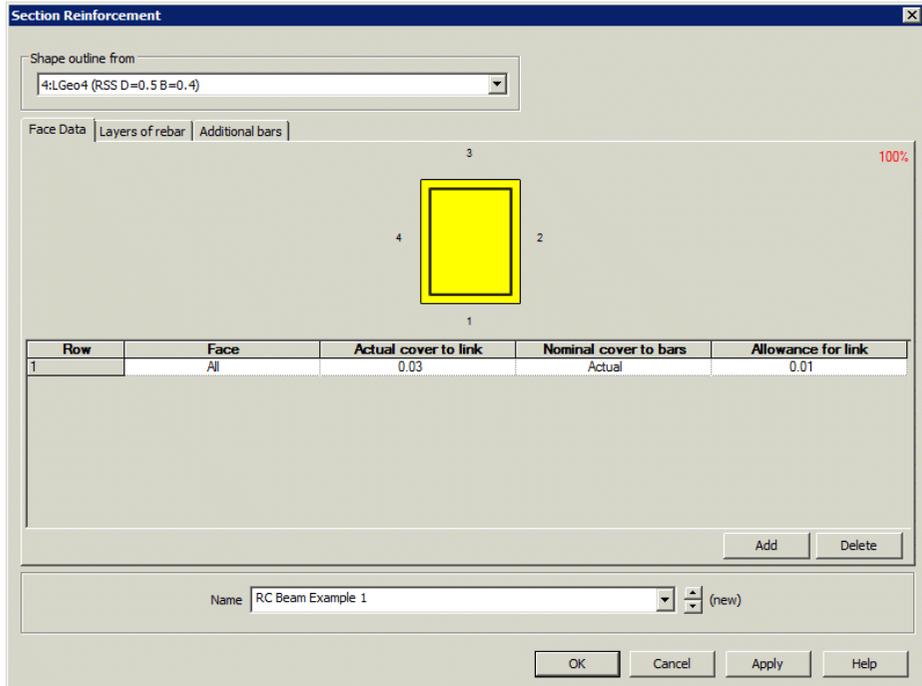
Defining face, cover and link data for a member

For all, or individual faces of the section, the actual and nominal covers, and allowance for links must be stated.

- ❑ **Face** When a shape outline has been specified, the default entry for Face is 'All', meaning that all faces will be assigned the settings made on the row. However, additional rows may be added and by entering a face number different covers on different faces can be stated. Cover and link allowance settings on those referenced faces will override the settings made for 'All' on those faces.

- ❑ **Actual cover to link** Define the distance from the face to the outside fibre of the link.

- ❑ **Nominal cover to bars** If 'Actual' is entered, the nominal cover will be calculated automatically from 'Actual cover to link' + 'Allowance for link'. Alternatively, a user-defined value may be entered. The value entered is used in

- ❑ **Allowance for link** Define the bar diameter of the link for positioning of longitudinal reinforcement. Shear and torsion checks are not included.

The leg(s) of a link in a selected row is drawn in blue in the section visualisation on the dialog, otherwise all legs of a links are drawn in black. Layers of reinforcement bars are not drawn in this page.

Table related buttons

❑ **Add** and **Delete** buttons add or remove rows from the table.

Layers of rebar

Defining main section reinforcement for four faces of a member

Overview

Layers of reinforcement are defined by entering a row of data for a numbered face. Faces forming the perimeter of a solid section are numbered anti-clockwise. Faces forming the perimeter of voids within a section are numbered clockwise. References to 'Start' and 'End' with regard to bar diameters and distances stated in the grid relate to the start and end of a face. For example face 2 starts at the end of face 1, and ends at the start of face 3.Note that bar diameters and distances must be defined in model units.

Bars are spaced equally in a layer with respect to a specified face. Where end bars in layers in different faces clash (as in the preceding example), the end bars from selected faces (the side faces in the preceding example) may be omitted by making the appropriate selection on the dialog.

Multiple rows of data in the table can be used to position bars in a specified face, enabling the definition of more dense or sparse reinforcement within a layer for a face, as shown on the following image.
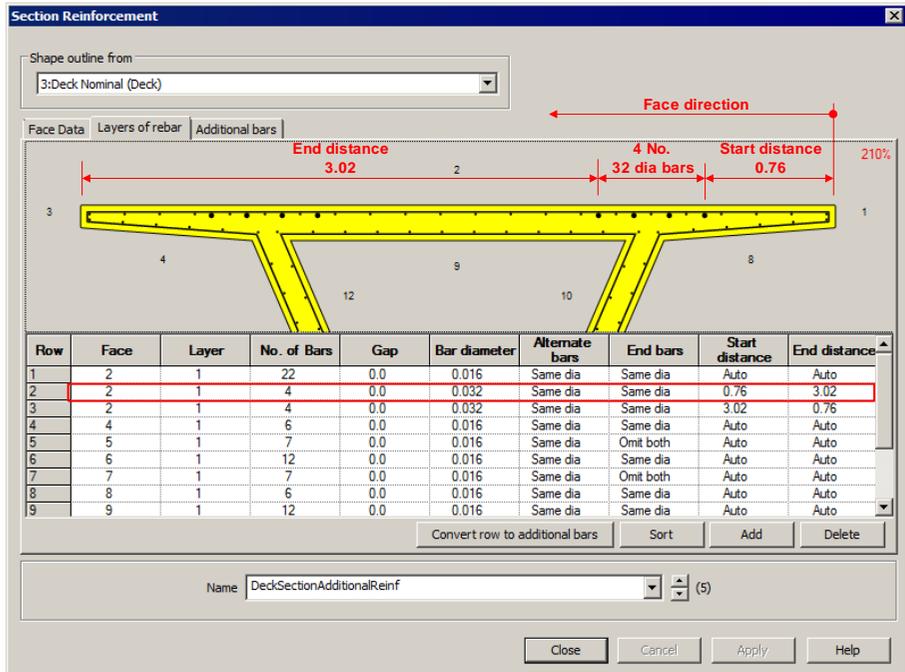
Defining additional reinforcement in a specified region of a face for a box section

Layers of rebar terms

See **Reinforcement terms used** for further explanation of the following options.

- ❑ **Face** The number of the face for which the layer reinforcement is being defined.
- ❑ **Layer** The layer number for the reinforcement. Layer 1 is the layer nearest the face. Multiple layers can be defined for a face with layers separated by a Gap (see below).
- ❑ **No. of Bars** The number of bars to be included in this layer / row.
- ❑ **Gap** The distance between the outermost fibre of this layer of rebar and the innermost fibre of the previous layer on the same face
- ❑ **Bar diameter** The diameter of bars in this layer / row of data entered in model units.
- ❑ **Alternate bars** The diameter of every alternate bar in this layer / row. If the option **Same dia** is selected all bars are the size defined by 'Bar diameter'. A user-defined bar diameter may be input instead.
- ❑ **End bars** Defines the diameter of the outermost (end) bars in this layer / row. If the option **Same dia** is selected the end bars are the size defined by 'Bar diameter'. A user-defined bar diameter may be input instead. Defining bars in a

layer for one face typically results in the corner bars at each end of a set or run of bars coinciding with the corner bars in a layer in another face. In such cases select **Omit both** from the droplist to remove the unwanted bars at both ends of one layer / row (as shown in for Faces 2 and 4 in the preceding image), or **Omit start** to remove one bar at the start of a run, or **Omit end** to remove one bar at the end of run of bars for the layer / row, as appropriate. The previously calculated and visualised spacing of the other bars in the face being edited will not change.

❑ **Start distance** Defines the location of the outermost (starting) bar in this layer / row. If 'Auto' is selected the starting bar of the layer / row will be placed automatically to sit tight to the adjacent leg(s) of the link. If a value is entered by over-typing in this cell, for orthogonal faces the centreline of the bar will be placed at the stated distance from the adjacent face. In non-orthogonal faces the centreline of the bar will be placed at the stated distance from the intersection of the two adjacent faces.

❑ **End distance** Defines the location of the outermost (ending) bar in a layer. If 'Auto' is selected the ending bar of the layer will be placed automatically to sit tight to the adjacent leg(s) of the link. If a value is entered by over-typing in this cell, for orthogonal faces the centreline of the bar will be placed at the stated distance from the adjacent face. In non-orthogonal faces the centreline of the bar will be placed at the stated distance from the intersection of the two adjacent faces.

The leg of a link in a selected row is drawn in blue in the section visualisation on the dialog, otherwise all legs of a links are drawn in black. Any additional bars are drawn in grey. Bars that clash with other bars, or that are positioned outside the section are drawn in red.

Note that whilst bars may be omitted at one end, the remaining bar at the other end cannot currently be specified with a different diameter and will inherit the bar diameter from the Bar diameter column.

Table related buttons

❑ **Convert row to additional bars** Deletes the currently selected row in the grid, inserting new rows in the grid of the 'Additional bars' page for each bar containing the appropriate data.

❑ **Sort** Sorts the grid rows by face and then layer

❑ **Add** and **Delete** buttons add or remove rows from the table.

Additional bars
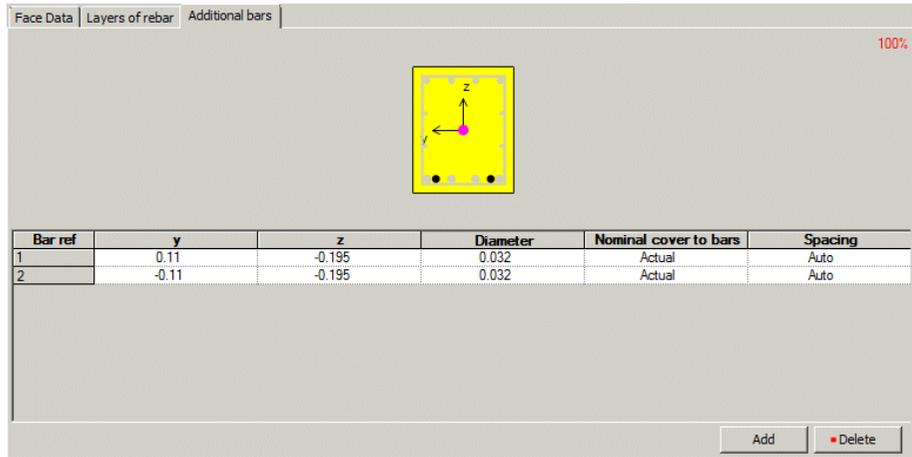
Defining additional bars in a face/layer

Additional bars may be added to a reinforcement arrangement. They are defined by a coordinate defining the centre of the bar with respect to the beam axis. Unlike bars defined on the 'Layers of rebar' tab, additional bars are not defined with respect to a specified face.

The local axes of the section are shown at its centroid. The magenta filled circle 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.

- ❑ **y** Defines the y location of the bar relative to the axis shown in the visualisation of the section shape.
- ❑ **z** Defines the z location of the bar relative to the axis shown in the visualisation of the section shape.
- ❑ **Diameter** The diameter of the additional bar.
- ❑ **Nominal cover to bars** If 'Actual' is entered, the nominal cover will be calculated automatically from 'Actual cover to link' + 'Allowance for link' (as stated on the 'Face Data' page.) Alternatively, a user-defined value may be entered.
- ❑ **Spacing** Selecting 'Auto' will cause bar spacings for crack width calculation checks to be based on the nearest bar. A user-defined value may optionally be entered, and if done, is used for the crack width calculation for that bar location only.

For a selected row, a bar is drawn in blue in the section visualisation on the dialog, otherwise all other bars are drawn in black. Bars that clash with other bars, or that are placed outside the section, are drawn in red. Other reinforcement (defined on Faces data and Layers of rebar tabs) is drawn in grey.

Table related buttons

- ❑ **Add** and **Delete** buttons add or remove rows from the table.

Bar positioning, spacing and crack width calculation

By default the positioning of end bars in a layer assumes a zero internal radius for a link, with the bars being fitted tight to the link. For many purposes, the effect of a difference in actual to modelled end bar position will be negligible in comparison to other assumptions inherent in the analysis and design process,. However, manual positioning of end bars can be done by omitting the end bars on the 'Layers of rebar' tab, and using the 'Additional bars' tab to manually place end bars in a desired location.

Bar spacing used for determination of crack widths is calculated by considering each bar in the section. The presence of any bundled bars is identified initially, with an appropriate equivalent diameter and location based upon the centre of area of the bundle being calculated. Then, for each bar in the section, the spacing used for crack width calculation is arrived at by considering the distance to the closest bar measured parallel to the concrete face, discounting bars in a 45 degree cone 'behind' the bar in question. Two nearest faces are considered in order to evaluate values for corner bars reasonably. If more control over the spacing assumed for the determination of crack widths is required, the 'Additional bars' tab allows user input of the value, bar by bar. Individual layers of rebar may be converted to 'Additional bars'.

Copying reinforcement details between models

Section reinforcement details can be transferred between models. See **Transferring Data Between Models.**

Line reinforcement

Line reinforcement is referenced by **geometric line beam** attributes (that define the concrete section size/shape and properties for a member). Line reinforcement is used to specify how individual **section reinforcement** arrangements apply over a length of a line or lines representing a concrete member. Line reinforcement is a utility object stored as an entry in the Utilities 🔧 Treeview and cannot be assigned to lines in the model directly.

The line reinforcement dialog can be accessed from a geometric line attribute dialog (by selecting the 'New' option in the reinforcement panel), or by using the **Utilities > Reinforcement > Line** menu item.

Line reinforcement is available for use with selected Bridge and Civil software products only, and for licences that support **RC frame design**. A model **analysis category** of 3D must also be in use.
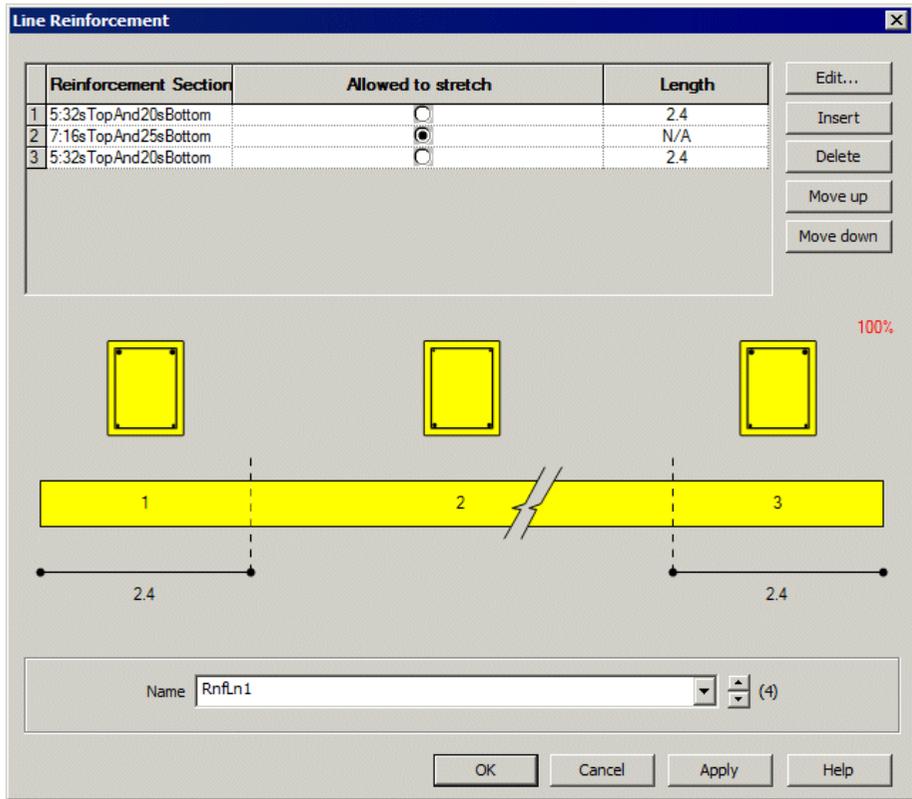
Defining line reinforcement

In the line reinforcement table the reinforcement sections and the length over which each reinforcement section applies are stated.

- ❑ **Reinforcement section** Clicking on the launch dialog button ![icon] in the this cell lists any previously define **section reinforcement** utilities for selection. A **New** button allows new **section reinforcement** arrangements to be defined.
- ❑ **Allowed to stretch**/ **Length** When more than one reinforcement section is included, the length over which each section applies must be stated in the **Length** column. The remaining section will be defined as **Allowed to stretch** to signify that this reinforcement section arrangement will apply over the remaining length of line to which it is assigned.

As each row is added or edited, a schematic diagram beneath the table will update to show the revised arrangement. Any number of section reinforcement arrangements may be included to form a line reinforcement utility.

Table related buttons

- ❑ **Edit**, **Insert** and **Delete** buttons provide the means to edit reinforcement sections, create new rows above a selected row, and to delete rows from the table.
- ❑ **Move up** and **Move down** buttons allows re-ordering of a selected row within the table.

Usage

Once defined, line reinforcement utilities are added to the Utilities 🔧 Treeview. They cannot be directly assigned to lines in a model. They are assigned to the model as a function of assigning the **geometric line beam** attribute in which they are referenced. When assigning to single or multiple selected lines the following options are provided:

- ❑ **Consider selection as a single assignment** The line reinforcement details will start and end at the start and end of the line or set of lines selected. For a set of lines the line direction of each line must be the same.
- ❑ **Consider each selected item individually** The line reinforcement details will start at the start of each selected line, and end at the end of each selected line. as dictated by the line directions.

Examples of line reinforcement assignments
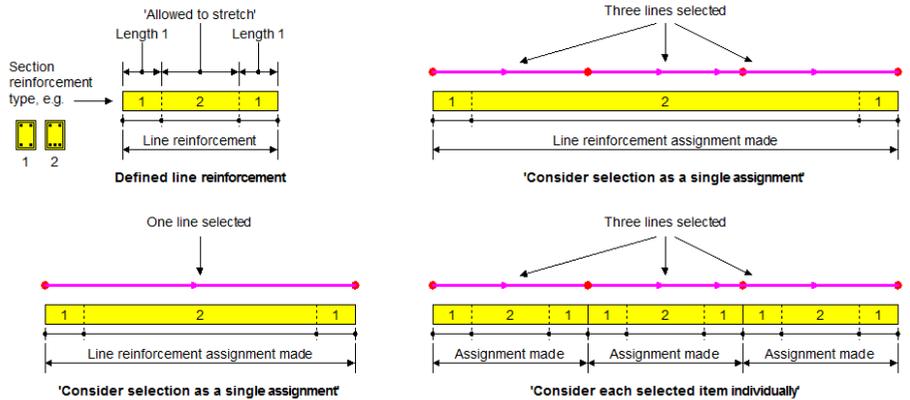
Examples of assigning geometric line attributes that reference line reinforcement

Note that it also possible to select separate sets of continuous lines (for example three sets of lines with three lines in each set) and assign a line reinforcement attribute to all at the same time using the 'Consider as a single assignment' option.

Example of 'Consider selection as a single assignment'

Three section reinforcement utilities are defined, assembled and assigned to a set of lines representing a bridge deck.
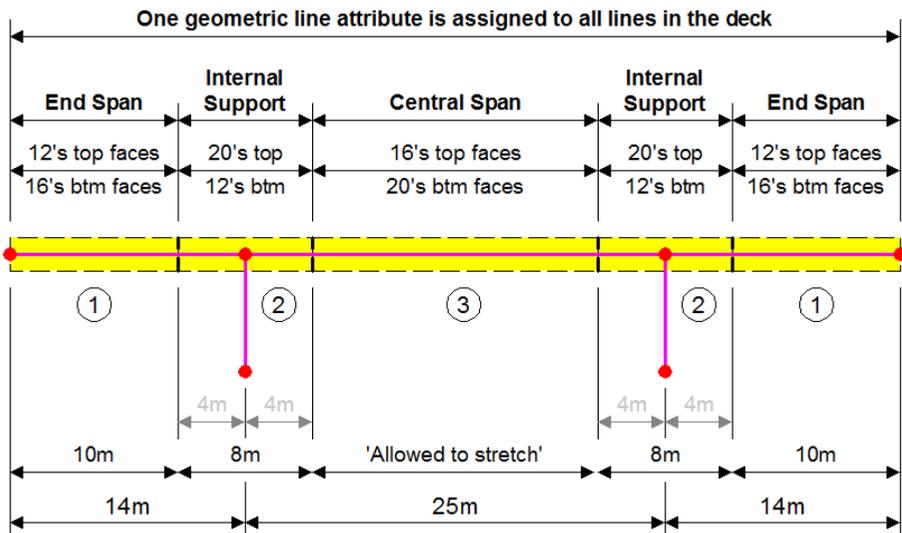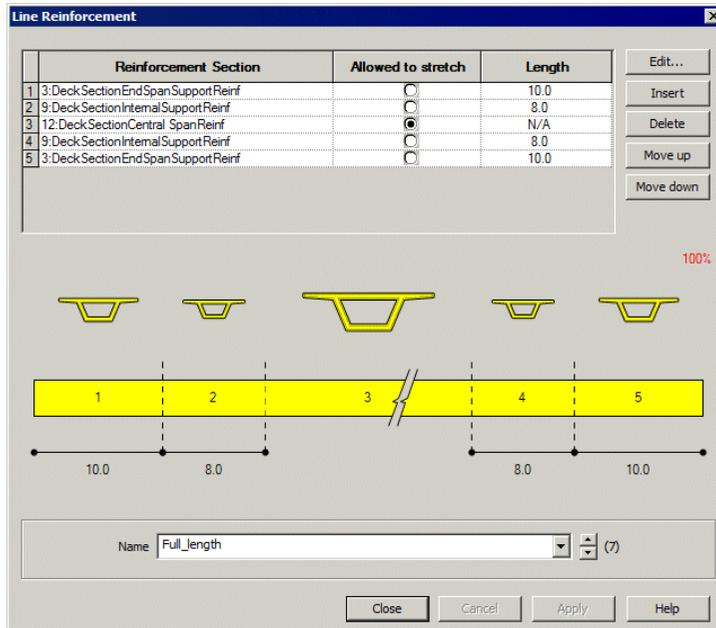
Copying between models

Line reinforcement details can be transferred between models. See **Transferring Data Between Models.**

Response Spectrum Types (for IMD loadcase use)

❑ **User-defined** response spectrum data
❑ **Design code** response spectrum data

Response Spectrum - User Defined for IMD loadcases

Response spectrum data for use with IMD loadcases is defined by selecting the general **Utilities > Response Spectra** menu item.

Response Spectrum datasets are referenced from IMD loadcases when calculating spectral response.

❑ **Frequency / Period** versus **Displacement / Velocity / Acceleration / Gravity**
The value of the frequency used in spectral calculations is interpolated from the frequency-amplitude or period-amplitude values defined. The type of values entered, i.e. displacement, velocity, acceleration or acceleration/gravity should match the type of support motion which is used to excite the structure.

❑ **Spectral curve damping** Defines the damping inherent in the response spectrum. A default value is 0% is initially set.

Notes

- More copious Response Spectrum tables are more easily defined using a command file, or by copying and pasting the data from a text file or spreadsheet. Command files can be used to store a library of frequently used Response Spectrum datasets.

- Defined Response Spectrum datasets are stored in, and may be edited from, the Utilities Treeview.

Response Spectrum - Code Based (For IMD loadcase use)

Design code response spectrum data for use with IMD loadcases is defined by selecting the general **Utilities > Response Spectra** menu item. The following codes of practice are currently supported:

❑ **ASCE 7-10 (2010)**
❑ **ATC-40 (1996)**

❑ **China GB 50011-2010**
❑ **EN1998-1:2004 Design (Horizontal)**
❑ **EN1998-1:2004 Elastic (Horizontal)**
❑ **FEMA 356 (2000)**

Note: When defining a design code-based response spectrum for an IMD loadcase the following options (not documented in the above links) are additionally available on the dialog shown.

Curve definition

❑ **Incremental period** Enter the increment in period of vibration, in seconds, used in the creation of the response spectrum curve (Default = 0.2s). The incremental period cannot be greater than the maximum period.

❑ **Maximum period** Enter the maximum period of vibration, in seconds, used in the creation of the response spectrum curve (Default = 6.0s).

Show graph

Displays a graph based upon the defined design code data.

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 Attributes** 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.
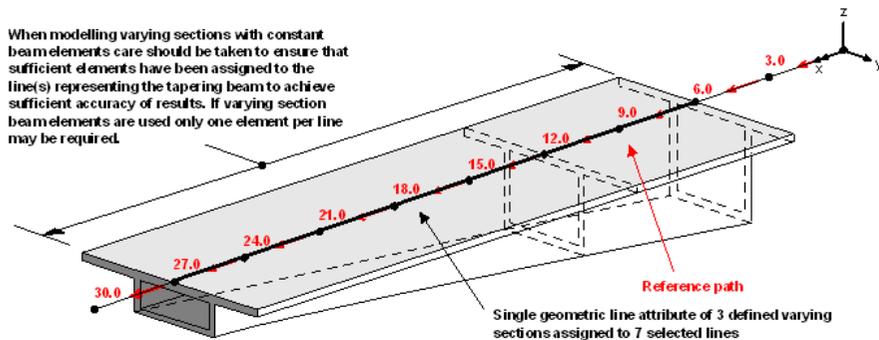
Reference path showing multiple line beams assigned a single multiple varying section (for clarity beam lines have been visualised at top of section)

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.
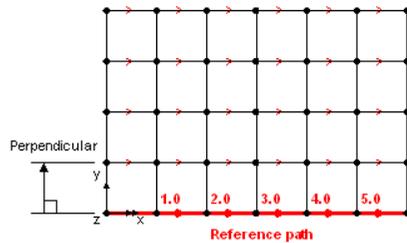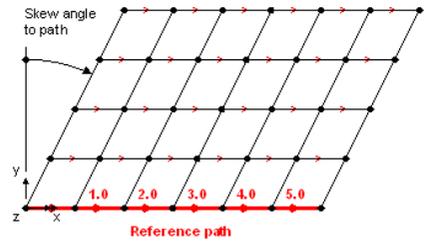
Perpendicular   **Skew angle to path**
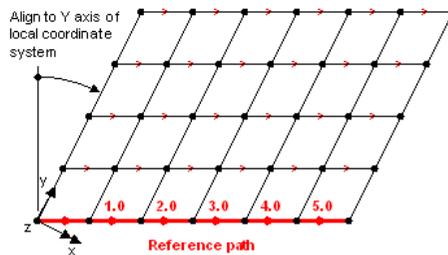
Local coordinate

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 [icon] 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.
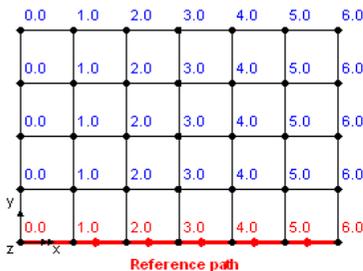
Perpendicular (orthogonal) grid/grillage   Skewed grid/grillage
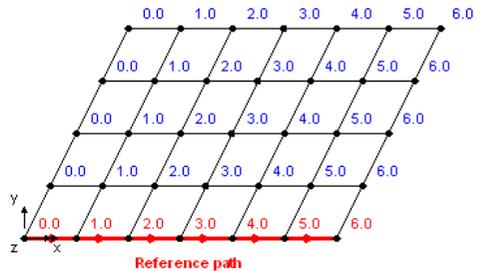
Examples of valid reference path points visualised on the model

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, whilst 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.

One-click reports

The one-click report facility creates a one-page modelling and results summary for each selected feature showing a graphical representation of primary load effects on that feature or object for the active loadcase. In its simplest form a one-click report is created by first selecting a line or surface in the model (or element) and then selecting the context menu item **One-click report...** for that feature. Multiple features (but only of the same type) can also be selected and be reported upon, and in this situation a multi-page report summary would be created instead.

One-click reports can be printed off individually or be added to as individual chapters to **main reports**. Each time a main report is generated the contents of an included one-click report are updated to reflect the current state of the model and any corresponding results.

The one-click report facility can also be accessed from the **Utilities > Reporting > One-click report** menu item, or from the One-click report ⊞ toolbar button.

Report contents

A one-click report contains a multi-celled table filled with modelling data, and unaveraged results plots and diagrams that differ according to the element type that is assigned to the line or surface feature, or element that was selected. The layout of a one-click report is pre-defined by a supplied template. A separate template is provided

for each element type that is supported by the one-click reporting facility. By default no page margins are initially applied, but they can be added by using the Edit page margins button.

Example reports
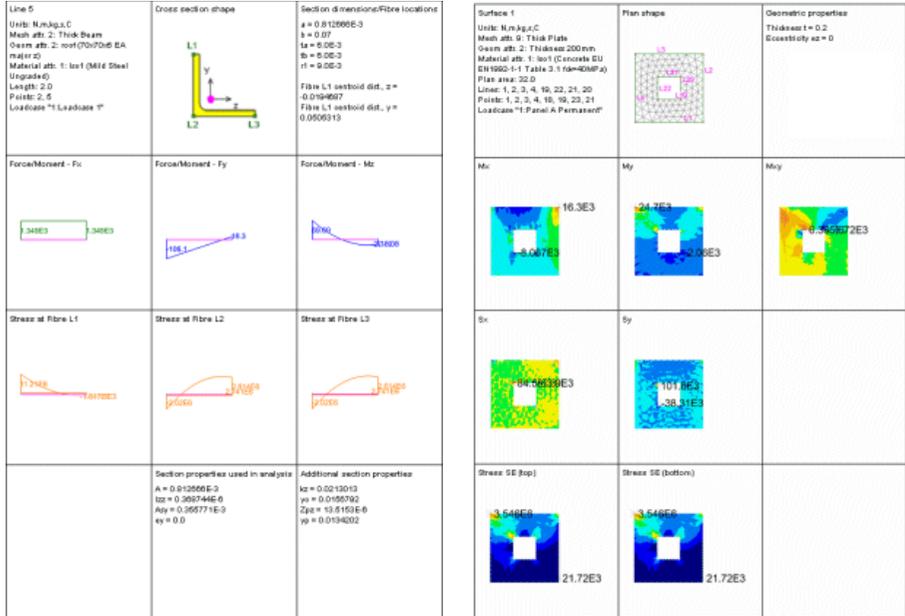
One-click report for a line beam member
(2D thick beam template)

One-click report for a surface
(2D thick plate template)

Typical report contents for a beam element

A 2D thick beam element the template plots the following model and results data and images on a single one-click report page:

Feature type and ID	Cross-sectional view of shape with respect to its assigned line showing any defined fibre locations	Section dimensions and fibre locations
Modelling units in use		
Mesh attribute ID and name		
Mesh type name		
Geometry attribute ID and name		
Material attribute ID and name		
Length of feature		
Points (or nodes) defining feature		

Loadcase ID and name		
Diagram of Force / Moment results of Fx along the feature	**Diagram of Force / Moment results of Fy along the feature**	**Diagram of Force / Moment results of Mx along the feature**
Diagram of Stress results at fibre location 1 (if defined)	**Diagram of Stress results at fibre location 2 (if defined)**	**Diagram of Stress results at fibre location 3 (if defined)**
Diagram of Stress results at fibre location 4 (if defined)	**Section properties of feature as used in analysis**	**Additional section properties**

Typical report contents for a plate element

A 2D thick plate element the template plots the following model and results data and images on a single one-click report page:

Feature type and ID	**Plan view of shape of selected surface or element showing surface axes (for surface) and mesh**	**Geometric properties**
Modelling units in use		
Mesh attribute ID and name		
Mesh type name		
Geometry attribute ID and name		
Material attribute ID and name		
Plan area of feature		
Lines and Points (or nodes) defining feature		
Loadcase ID and name		
Contour plot of Force / Moment results of MX for the selected feature	**Contour plot of Force / Moment results of MY for the selected feature**	**Contour plot of Force / Moment results of MXY for the selected feature**

Contour plot of Force / Moment results of Stress Sx for the selected feature	Contour plot of Force / Moment results of Stress Sy for the selected feature	(empty cell)
Contour plot of Stress results of Stress SE (top) for the selected feature	Contour plot of Stress results of Stress SE (bottom) for the selected feature	(empty cell)

Printing / previewing

With a One-click report displayed and in the One-click report window:

- Press the **Print** button to print the report page(s).

- Use the **Print preview** button to view how the page will print. Zoom and unzoom options are provided. Note that the page orientation is pre-set to be Portrait and is not governed by any settings made on the Page Setup dialog that would be accessed via File > Page setup.

- Use the **Edit page margins** button to set left, right, top and bottom margins to the report cells. If margins are applied, use the Print preview button to see a true representation of what will be printed.

Adding to main reports

With a One-click report displayed and in the One-click report window:

- Press the **Add to report** button. Select the pages to be inserted, identify the position within a report where the page is to be added, and click the OK button.

Each time a main report is generated the contents of an included one-click report are updated to reflect the current state of the model and any corresponding results.

One-click report templates

One-click report templates are supplied for the following element group types:

- Bars (2D and 3D)

- Beams (2D, 3D, Thick, Thin, Cross-section and Grillage

- Plates and shells (Thick and Thin)

- 2D continuum (Plane stress, plane strain, axisymmetric) and 3D membrane

- Axisymmetric shell

- Axisymmetric membranes

Customisation

For further details regarding the location and potential user customisation of one-click report templates, please contact LUSAS technical support.

Tools

Tools differ from model **utilities** in that they do not generally create entries in the Utilities 🦴 Treeview.

Details of the features and use of many of the tools listed below are provided in other relevant chapters of the manual. The remainder are described in this chapter.

The complete list of LUSAS tools (accessed from the **Tools** menu) is shown below. Clicking on a hyperlink will take you to the appropriate location.

- ❑ **Mesh** - mesh node measurement, and controlling of automatic remeshing.
- ❑ **Annotation** - adding of text, line, bitmap, window summary and border annotation to the view window.
- ❑ **Inspection Locations** - define points of interest on the model for results extraction / viewing purposes,either by cursor or by pre-selection.
- ❑ **Set Fourier Angle** - specify the angle around the circumference at which Fourier results are required
- ❑ Set Harmonic component -
- ❑ **Animation Wizard** - animate the mode shapes or load history of a structure
- ❑ **Section Through 3D** (Analysis type 3D only) - slice a solid model and plot results on the section defined
- ❑ **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** - renumbers Nodes and Elements, and Point, Line, Surface and Volume geometry features for all of a model.
- ❑ **Renumber Selection** - renumbers 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 Calculator** - calculates cross-sectional geometric properties for a range of sections

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 **Tools > 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 **Tools > 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**.

Vertical axis

Vertical axis settings on the Model Properties dialog can be accessed from the **Tools > 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.

- Determines the direction in which construction table values are reported.

Note that setting the vertical axis on the Model Properties dialog will supercede any vertical axis setting defined on the Direction definition dialog.

Library Management

The location of the section and material libraries is defined from the **Tools> 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 **Tools> 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 **Tools> Library Management> Section Librray> Delete Properties** menu item.

Renumbering

By using **Tools > Renumber All...** or **Tools > 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 **Tools> 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 **Tools> Renumber All**
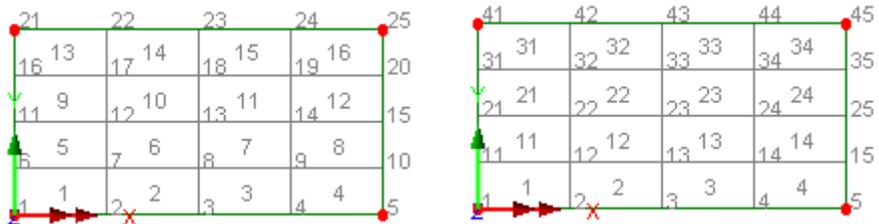
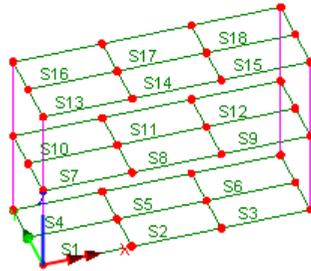Before: All nodes and elements numbered using global   After: All nodes and elements renumbered using

global directions.

Start value=1, Positive Z-axis, increment 1, Positive Y-axis, increment 1, Positive X axis, increment 1 (Default numbering method)

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 **Tools> 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)
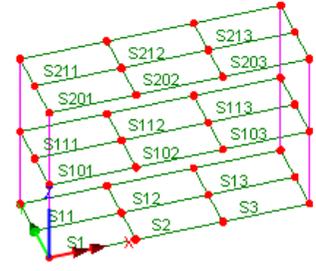
After: All surfaces renumbered using global directions.

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
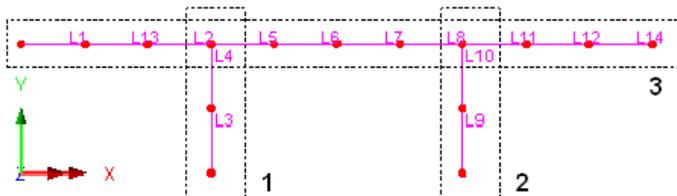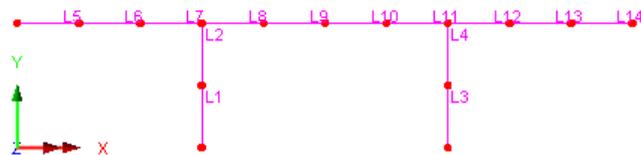
Lines are renumbered in a 2D line model using **Tools> Renumber Selection**.

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 **Tools> Renumber All**.
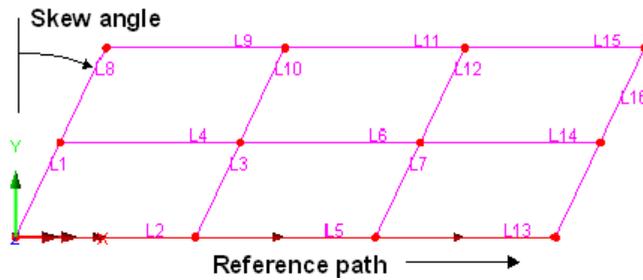
Before: All lines numbered as drawn. Reference path defined along lines L2, L5 and L13
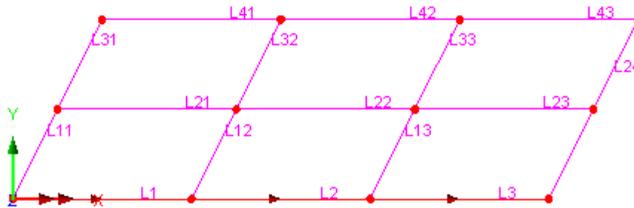
After: All lines renumbered along and transverse to reference path.

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 **Tools> Renumber All**.
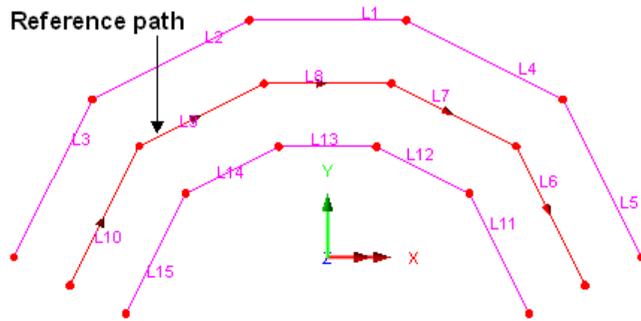
Before: All lines numbered as drawn. Reference path defined along lines L10, L5, L8, L7 and L6
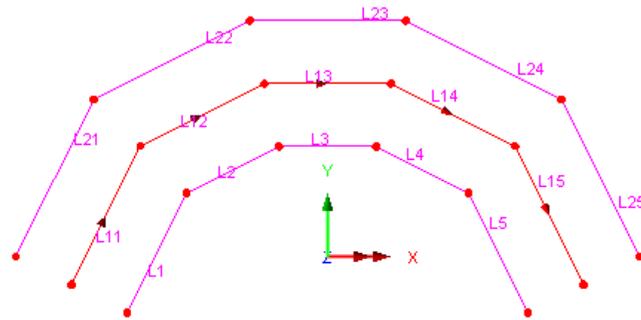
After: All lines renumbered along and transverse to reference path.

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.

❑ **Infill/encased sections** for a selection of commonly used section shapes.

❑ **Plate sections** for a range of riveted or welded sections, with some sections having an option to incorporate library sections.

Standard Section Property Calculator

Standard section property calculators are accessed from the **Tools> Section Property Calculator** menu item. The following sections are supported:

- ❑ **Rectangular** - solid, hollow, equal and unequal flange/web thickness, stiffened box.
- ❑ **Circular** - solid, hollow, elliptical hollow, stiffened pipe.
- ❑ **I section** - equal and unequal flanges, lipped, haunch, cross I, combined T.
- ❑ **T section** - of standard shape.
- ❑ **L section** - single, single with external radius, double (back-to-back).
- ❑ **C section** - lipped, unlipped, with external radius, with unequal external radius, double (back-to-back), double (front-to-front), top hat
- ❑ **Z section** - lipped right-angle, lipped inclined, unlipped
- ❑ **More sections** - Hexagonal (both solid and hollow), Octagonal solid, Trapezoidal hollow.
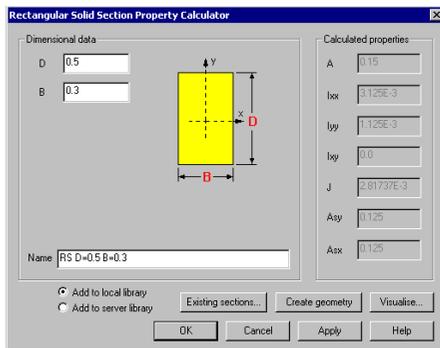
Typical standard section property calculator dialog
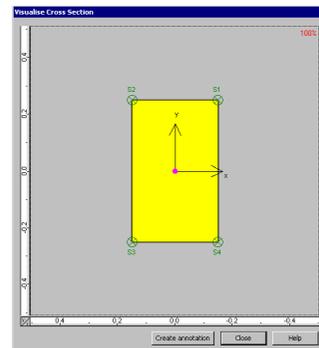
Section visualisation showing fibre locations

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 (a user-defined location that can be shared with other users of the same computer or network) 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.

Back-to-back or front-to-front sections

Note that in the case of back-to-back sections (for L-sections and C-sections) or face-to-face sections (for C-sections only) the pair of sections are treated as separate sections and not as rigidly connected sections. As such, setting the distance between the sections 's' to zero, or changing it to be a positive number does not have any effect on the calculated torsion constant J. The two back-to-back or front-to-front sections are assumed to twist independently about their own shear centres and hence the torsional constant calculated will be twice the torsional constant of a single C-section/ L-section. More accurate calculation of the torsion constant J can be achieved using the Arbitrary Section Property Calculator, and this can also be used to calculate J for the case where the back-to-back / front-to-front section is to be considered as rigidly connected.

Using standard sections

To use the computed section properties in a model the section must be saved to a local or server library. Once saved, 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. On completion, pressing OK creates an attribute, which can be subsequently **assigned** to the required Line(s) in the model.

Arbitrary Section Property Calculator

The arbitrary section property calculator is accessed from the **Tools> Section Property Calculator> Arbitrary Section** menu item. It computes section properties (area, moments of inertia and torsion constant etc.) for any open or closed section of similar (homogenous) material, or for a compound section made of two or more materials, and calculates extreme fibre positions for use when plotting stresses on fleshed beams. Note that the Arbitrary section calculator cannot be used to calculate properties of a cross section comprised of two or more shapes that are not connected.

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 (as used in results viewing) 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 cross-sections can be created initially using points and lines, ultimately they must contain

surfaces that define a single continuous shape. For solid sections or solid sections with holes this will need to be done by manually creating the surfaces required.

For **thin walled box sections** surfaces can be extruded automatically from lines drawn to represent the centrelines of the plated members.

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 cross-sections with voids.


```
A    = 0.499939
Ixx  = 0.0409351
Iyy  = 0.0259411
Ixy  = -3.59487E-15
J    = 0.0122665
Asy  = 0.190254
Asx  = 0.353654
kx   = 0.286147
ky   = 0.227791
xo   = -3.75468E-6
yo   = 0.0957573
Cw   = 1.85182E-3
xp   = -21.8081E-15
yp   = 5.39388E-3
Zpx  = 0.132739
Zpy  = 0.0926569
Zpt  = 0.0
Theta= 0.0
Betax= -0.152609
Betay= 21.9485E-6
Ixr  = 1.60145E-3
Iyr= 1.77395E-12
Irr= 0.0129323
Iwr= 62.9907E-12
xc   = 21.8081E-15
yc   = -0.204953
```

Centroid
Shear Centre
Plastic Neutral Axis
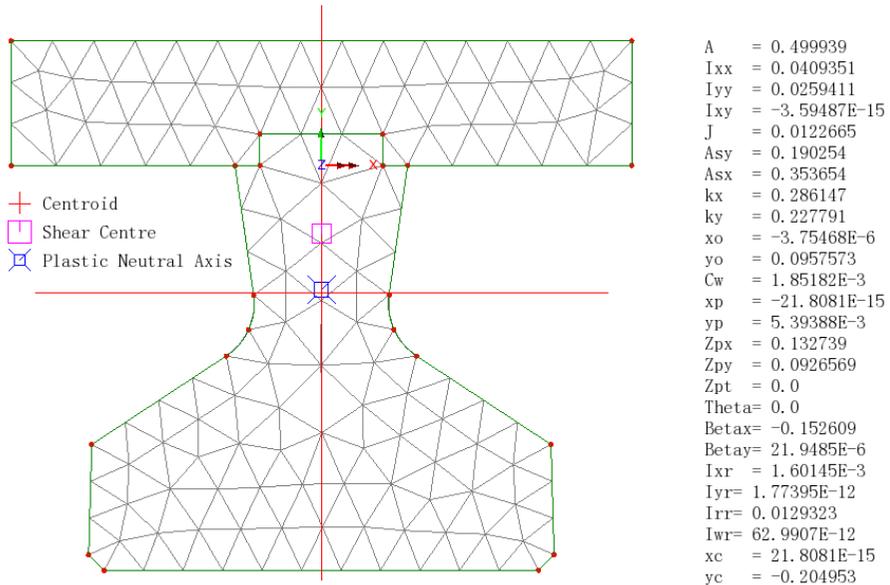
Section properties calculated

Standard properties	Additional properties for design checking	Plastic properties
A - Cross-sectional area	kx - Radius of gyration about x axis	Ap - Plastic cross-sectional area (= elastic area)
Ixx - Second moment of area about x axis	ky - Radius of gyration about x axis	Zpy - Plastic section modulus about y axis
Iyy- Second moment of area about y axis	kz - Radius of gyration about x axis	Zpz - Plastic section modulus about z axis
Izz - Second moment of area about z axis	Cw - Warping torsional constant	Zpt - Plastic torsional section modulus
Ixy, Iyz, Izx - Product moments of area	xo - Shear centre, distance from centroid along x axis	Yp - Plastic neutral axis, distance from centroid along y/z axis
J - Torsional constant	yo - Shear centre, distance from centroid along y axis	Zp - Plastic neutral axis, distance from centroid along y/z axis
Asx - Effective shear in x direction	zo - Shear centre, distance from centroid along z axis	
Asy - Effective shear in y direction		
Asz - Effective shear in z direction		

Plastic geometric properties are required for beams when using the stress resultant material model (model 29)

Thin walled box sections

For thin walled box sections, only the points and lines that define the centrelines of the plated members, and the geometric thickness of each line needs to be initially defined. The thickness of each line is specified by assigning a **geometric line ASPC Thickness** attribute. LUSAS will automatically extrude the lines to create surfaces in order to compute the section properties. Note that a mesh does not need to be assigned to the lines defining the section if automatic meshing will be used when calculating section properties. This functionality is only available for models defined with an **analysis category** of 2D Inplane. Surfaces of a width equal to half of the assigned thickness are automatically extruded either side of each line during the section property calculation process to determine the correct section properties, unless a line is also attached to a surface in which case the extruded surface is generated away from the existing surface and does not overlap with that surface. The extruded surfaces are automatically meshed QFD8 elements; one to each side of the centreline and a number of elements along the length to achieve a sensible aspect ratio. These surfaces can be optionally retained after the section property values have been calculated. To enable the model to be returned to the original line model a new group named "Extruded" is created to allow the geometry that was used to defined the surfaces to be easily identified and deleted.
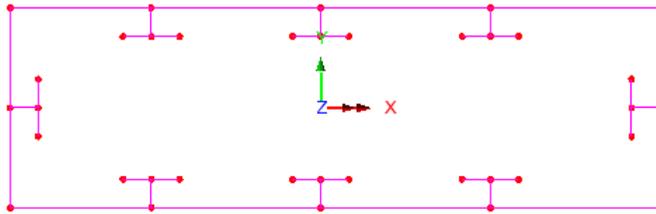
Note that a cross-section representing the shape of a single (homogenous) material does not need to have a material assigned to it for section property calculation to take place, but if a material is assigned it must have a unit Thermal conductivity defined.

A thin walled box section constructed of two or more materials can be accommodated using the modular ratio approach. For this different materials should be assigned to the lines as appropriate. If specified, the material properties on the line will be transferred to the extruded surfaces.

The section properties calculated for thin walled box sections modelled using the 'extruded surfaces' method very closely match those of cross-sections defined using solely surfaces, but very slightly over-estimate the values due to the localised overlapping of extruded surfaces at junctions.

Compound sections

Compound sections can be defined from existing library sections that are positioned relative to each other - see **Compound Sections (from library items)**, or by being drawn in the view window. Compound sections constructed of two or more materials can be accommodated using the modular ratio approach. For this an isotropic thermal 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 Young's Modulus of the material the surface represents and E_{base} is the Young's Modulus of the material to be used in the structural analysis. When defining a modular ratio, using thermal material properties of less than 1% of the structural material properties is recommended. See **Case study: Arbitrary Section Property Calculation of a Compound Section** for a basic procedure.

A thin box section constructed of two or more materials can also be accommodated using the modular ratio approach. For this, different material properties should be assigned to the relevant lines. If specified, the material properties on the line will be transferred to the extruded surfaces.

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** allows LUSAS to mesh the cross-section based upon the Max element/line setting.
- ❑ **Max element/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.
- ❑ **Retain extruded surfaces** For use with cross-sections defined by points and lines only, when lines have been assigned with a **geometric line ASPC Thickness** attribute. This option retains the surfaces that are generated as part of the section property calculation.
- ❑ **Use Solver** uses the Solver-based Arbitrary Section Property Calculator provided in Version 15.2 instead of the default Modeller-based calculator. The Solver calculator does not run in-memory, and is provided as an alternative option in case memory limits are reached.
- ❑ **Name** is 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 the maximum 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 have been 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.

Note that the section properties created by the arbitrary section property calculator follow beam theory with the x-axis of a beam sitting along the beam's length, and with the y-axis to the left and the z-axis vertical.

Worked example

See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Arbitrary Section Property Calculation and Use', for a typical use of this facility.

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.

Infill / Encased Section Calculator

The **Infill / Encased Section Calculator** is available for Bridge and Civil & Structural software products only. See *Application Manual (Bridge, Civil & Structural)* for details.

Plate Section Calculator

The **Plate Section 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 steel beam and concrete slab is as follows:

1. Create a new model for a thermal analysis (required so that a thermal material attribute can be defined)

2. Draw the section and run the Arbitrary Section Property Calculator (ASPC) from the **Tools > Section Property Calculator> Arbitrary Section** menu item to calculate homogeneous section properties, saving the section to a user library with the name 'homogeneous section'. This will set the model up and assign a unit thermal conductivity to all surfaces.
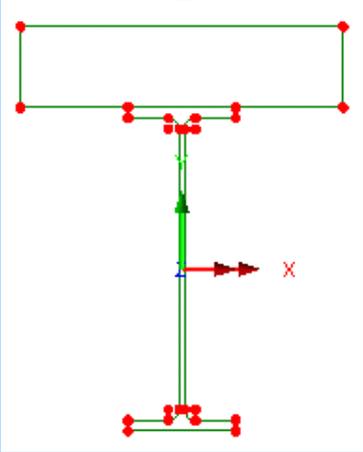
3. Use the **Attributes > Material > Isotropic** menu item to define a new material, select the Thermal tab and set the Thermal Conductivity, K, to be the modular ratio E2/E1 where E2 is the Young's Modulus of the material <u>to be represented</u>, and E1 is the Young's Modulus of the material <u>to be used in the structural analysis</u>. The calculated section properties will take into account the modular ratios.

4. Note that if the structural analysis is to be based upon steel material properties then the modular ratio assigned to the surfaces representing the concrete would be $E_{concrete}/E_{steel}$.

5. Assign the newly defined material property to the surface representing the concrete slab.

6. Re-run the ASPC to calculate the compound section properties, saving the section to a user library with the name 'compound section'.

Note that compound sections can also be defined from library items using the **Attributes > Geometric > Compound Section** menu item. See **Compound Sections** for details.

Chapter 7 : Analyses

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

❑ **Structural Analysis** - used to carry out all kinds of linear static, eigenvalue, fourier, nonlinear, or transient dynamic analysis.

❑ **Thermal Analysis** - used to assess conduction, convection or radiation effects.

❑ **Hygro-Thermal Analysis** - used to model the evolution of temperature, water saturation and degree of hydration.

❑ **Direct Method Influence Analysis** - used to create an influence line or surface by way of a unit load placed at each node or grid location on the loadable area of a structure.

❑ **Vehicle Load Optimisation** - used to identify critical vehicle loading patterns on bridges and apply these loading patterns to LUSAS analysis models.

❑ **Railway Load Optimisation** - used to identify critical train loading patterns on bridges and apply these loading patterns to LUSAS analysis models

❑ **Linear 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, Rail 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

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 or Hygro-thermal analysis entry or vice-versa.

- ❑ **Coupled Thermal /Structural Analysis** - performs a thermal and structural analysis simultaneously, or one after the other, by transferring data between them via an additional data transfer file. Use to model used to assess conduction, convection or radiation effects and resulting stresses due to the thermal effects
- ❑ **Coupled Hygro-thermal/Structural Analysis** performs a hygro-thermal and structural analysis simultaneously, or one after the other, by transferring data between them via an additional data transfer file. Use to model the thermo-mechanical behaviour of concrete at early age and the prediction of cracks due to shrinkage.

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 with time in nonlinear, transient and Fourier analyses.
- ❑ **Nonlinear Cable Tuning Loadcase** for use with nonlinear analysis or staged construction modelling, where a nonlinear cable tuning loadcase could be created to follow each loadcase representing a construction stage, or follow any chosen loadcase or series of loadcases.

These facilities can also be accessed using the context menu of an analysis entry.

Dynamic response analysis

- ❑ **IMDPlus** - the investigation of various dynamic responses using the results from an eigenvalue analysis

Other entries

- ❑ **Envelope** - create an envelope of maximum and minimum effects.
- ❑ **Basic Combination** - combine results from different loadcases applying load factors to each.
- ❑ **Smart Combination** - combine results from different loadcases taking into account beneficial and adverse load factors to get max and min effects.
- ❑ **Design Combination** - used to assign LUSAS loadcases, envelopes or basic combinations to predefined load types for a supported design code.
- ❑ **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)

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.
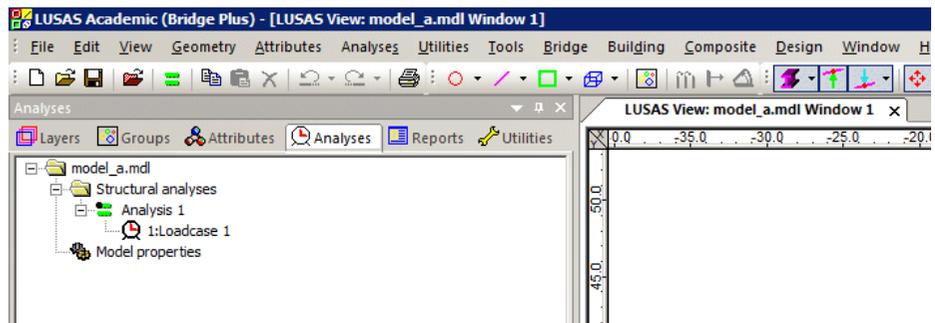
By default, a base analysis ▬ entry is automatically created in the Analyses 🔍 Treeview by LUSAS Modeller when a new model is defined.

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, and when done all loadcases and loading assigned to a model is copied with the analysis. 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 are held within folders according to analysis type. Folder names are Structural Analyses, Thermal Analyses, Hygro Analyses.

Main folder

Context menus for the main model folder ⬜ provide the means to view model properties and define new analysis types such as Structural, Thermal, Hygro-Thermal, Direct Method Influence, Railway Load Optimisation, Vehicle Load Optimisation and Linear Cable Tuning.

Analysis folders

Context menus for the analysis folders 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. The option to 'Close All Results Files' does exactly that. The option to 'Sort by Name' lists loadcase and loadcurve names alphabetically 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, output options, time management capabilities, the means to add new loadcases, the adding of gravity as a property of an analysis, and various Solver related options.

Loadcase entries

Loadcases 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.

Loadcase names are added to the Analyses treeview in creation order, but can be re-ordered by using the **Sort by Loadcase Name** context menu item accessed from the Analyses tab.

Copying and pasting analysis, loadcase and other attribute data

- Analysis entries can be copied and pasted within the Analyses ⬜ Treeview.

- Copying an analysis entry in the Analyses ⬜ Treeview will copy all Support and Loading data for the selected analysis 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.

- Loadcases may be moved within an analysis with respect to other loadcases, and if done so, the numeric identifiers will be updated.

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, delete all post-processing entries, or create loadcases from selected combinations.

The 🐾 Combination and envelope results options object provides optional and advanced settings.

Analyses symbols explained

See **Treeview Panels and Icons** for details of all symbols used in the Analyses 🕒 Treeview.

Analyses Treeview folder control

Geometric, Material, Support, and Loading folders in the Analyses 🕒 Treeview can be expanded or contracted by use of the options list and the Specify button located at the bottom of the treeview panel. Making a selection of one of these folder names opens that folder in the treeview. Selecting more than one folder name enables a Custom entry. Selecting All or None opens and closes all folders. The Results Only option display only results loadcases.
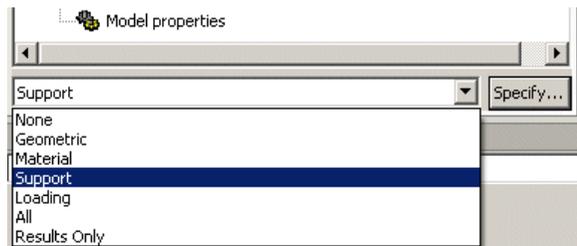
Creating a new analysis

New analysis entries can be added to the Analyses 🕒 Treeview by using the **Analysis** menu item, or by using the context menu for an analysis entry in the Analyses 🕒 Treeview. The analysis types available for selection will depend upon the existing analysis entry in the Analyses 🕒 Treeview.

General settings

❑ **Set this analysis as the base structural analysis** By default this is not checked (selected). As a result a new analysis will inherit most of the attribute assignments, options and settings etc. of the base analysis.

❑ **Inherit from base analysis** Deselecting the **All** option provides the means to individually select which attribute assignments, options and settings should be inherited from a base analysis and, by their non-selection, which should not.

A name for the new analysis is also required.

Notes

- For a summary of how attributes assigned in a base analysis are inherited in other dependent analyses see **Assignment behaviour of each attribute type** for more details.

- The check box for Material also controls whether Composite, Age and Damping attributes are inherited or not inherited from a base analysis.

Specifying an initial state for an analysis

Specifying an initial state for a new analysis can be set-up with respect to the state of the model at any point in another relevant analysis, or by selecting the **Edit** context menu item for an analysis in the Analyses treeview.

Deformation

A deformed mesh of a chosen loadcase from one analysis can be used as the starting point for the 'undeformed' mesh in a new analysis. All the coordinates of all nodes in the mesh are adjusted based upon a specified deformed mesh factor.

❑ **Start with undeformed mesh** uses the undeformed mesh of the model. This is the default.

❑ **Start with deformed mesh from** uses the deformed mesh from a chosen Analysis and Loadcase.

Increment / timestep / eigenvalue

Where a loadcase consists of multiple loadcase increments:

❑ **Last (also use this for linear analysis)** uses the last increment, time step or eigenvalue as the starting point.

❑ **Specified** In the case of a nonlinear, dynamic or eigenvalue analysis a particular increment, timestep or eigenvalue can be specified for which the deformed mesh data will be used in this analysis.

❑ **Scale factor** adjusts all the coordinates of all nodes in the mesh by scaling them by a factor. A scale factor of 0.015 will scale a 1 metre maximum deflection down to an initial deformation of 15mm.

Restart options (for a new analysis)

When an incremental analysis has failed for some reason and a re-solve is required from that point, or if a solution is to be deliberately continued by branching from a specified stage reached in a previous solution, two restart options are provided for use when a new analysis is created in the Analyses treeview:

❑ **Continue from failed analysis (restart)** For this, the left-hand droplist is filled with all the analyses that contain at least one nonlinear / transient control, and for which a restart file already exists on disk. Once an analysis is chosen, the right-hand droplist is filled with the increments/time-steps found in that restart file. For a chosen increment, the new analysis created in the Analyses treeview is given an automatic description identifying it as being one that continues from a restart. Any modification to the original analysis (change of geometry, supports, material, loading, etc) will require that original analysis to be solved again, and users should be aware that the new solution may contain differently named / numbered increments, potentially making any continuation from a specific increment invalid, requiring the continuation analysis to be re-stated.

❑ **Continue from specified loadcase (restart)** For this, the left-hand droplist is filled with all analyses that contain at least one nonlinear / transient control, and for which a restart file already exists on disk. Once an analysis is chosen, the right hand droplist is filled with the loadcases from that analysis. For as chosen loadcase, the new analysis created in the Analyses treeview is given an automatic description identifying it as being one that continues from a restart. Any modification to the original analysis (change of geometry, supports, material, loading, etc) will not require that original analysis to be solved again. This is because by choosing an existing loadcase (rather than an increment), LUSAS Modeller can automatically detect which increment this corresponds to, each time the model is solved.

A restart file must have been written to disk to continue a solution with a new analysis as described above. This is done by setting the '**Save a restart at the end of this control**' option within the nonlinear / transient control dialog.

Some practical uses of continuing a solution from a specified loadcase include:

- Evaluating four alternative ways of completing the last 40 stages in a 120 step construction sequence. The first 80 steps – identical for all sequences in this example – can be solved only once, and the four alternative 'endings' stored one after the other in the Analyses treeview.

- Considering three alternative ways of completing the last 3 seconds of a transient dynamic analysis or a structure, which share the first 9 seconds.

- Nonlinear buckling of a partially complete structure – from a particular stage – with six independent wind directions, while leaving the whole staged construction analysis intact from end to end.

- Carrying out a trial considering several alternative ways of obtaining convergence to the end of an analysis (for example, with different loading scenarios, convergence criteria, options etc.) without having to solve all the stages from the beginning each time.

Advanced Output Options for an Analysis

Advanced output options can be specified as part of initially defining an analysis and can be also accessed from the **Advanced** button on the Export Solver datafile dialog. These options determine what is optionally output to the **Solver output file** or to a **History file** for an analysis, each time a solve is carried out.

Following results calculation for the whole model, the results written to a results file output can be stated to be for all elements in a model (full model), or for only a subset of elements (those part of a **group**) within a model. Choosing the latter produces smaller results files more quickly than would be created for a whole model.

Results file scope

- ❑ **Full model** calculates and output results to a results file for all elements in a model.
- ❑ **<Group>** (if groups are present) Choosing a group calculates results for the whole model but only outputs results to a results file for those elements within the chosen group. The choice of Group also restricts the output options (nodal reactions and similar).
- ❑ **New...** provides the means to create and specify a new group based on selected items, or visible items, or just a new empty group to be referenced and added to later..

Solver output file - Element output

- ❑ 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 **Solver output file** at **Gauss** and/or **Nodal** positions.
- ❑ **History** when checked outputs the named variables, and selected node and element results from LUSAS Solver to a **History file.**
- ❑ **Scope** This has the same options as the results file scope but here the scope is defined for element output to the solver output file and history file.

Solver output file - Node output

- ❑ Node **Displacements** and **Reactions** can be written to the **Solver output file**, and also be written to a **History** file if required.

❑ **Scope** This has the same options as the results file scope but here the scope is defined for node output to the Solver output file and history file.

Solver restart file

❑ **Generate** (if selected) configures the **LUSAS Solver data file** to create a **Solver restart file** (.rst), which enables a nonlinear, dynamic, transient or viscous problem to be restarted after it has been chosen to stop an analysis during a solve.

Base Analysis

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. Other analyses present in the Analyses ⬜ Treeview that inherit assignments, options and settings from the base analysis are identified by a cyan solve ⬜ icon.

- The base analysis should generally be a linear analysis.

- 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 and normally suggested, that the base analysis is set to be a simple linear static analysis.

- For **influence analysis**, the influence results (and therefore the Vehicle Load Optimisation results) are based on the properties of the base analysis.

- Where multiple analyses may have been defined within the model, only the geometric properties defined for the base analysis are used in the calculations carried out to derive the lengths of any **rigid zones** present.

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, Results Transformation, Inspection Locations, Wood-Armer, Steel Frame Design, RC Slab Design	Assignments apply to all analyses
Damping, Geometric, Material (Composites, Age, Damping) Radiation Surface, Section Property Modifier, Thermal Surface, User Attributes	Assigned per-analysis. Optionally inherited from a base analysis. Appear in their own folder in the Analyses Treeview. Note that the Geometric entry also controls whether Section Property Modifier attributes are inherited or not. 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, Constraints	Optionally inherited from the first loadcase of a base analysis. Supports and constraints can be assigned to any loadcase (in a nonlinear model) but note that only supports and constraints from the first loadcase of a base analysis can be inherited between analyses. Supports and constraints in other loadcases are not inherited.
Loading, Activate, Deactivate	Not inherited between analyses.
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 Influence 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, thermal or hygro-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. The base analysis should generally be a linear analysis. 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 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.

Analysis Types

LUSAS may be used to numerically model a wide range of engineering problems. The following section briefly explains the analysis types available.

Structural analysis

❑ **Linear Analysis** is the most common analysis carried out by engineers and unless specified otherwise, By default, LUSAS will perform a linear elastic, static structural 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.

General settings

An analysis can be optionally set as a **base analysis** analysis, or to optionally inherit a number of attribute assignments, options and settings etc. of the base analysis.

Specifying an initial state 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 structural 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. See **Specifying an Initial State for an Analysis**.

Restarting an analysis

Where an incremental analysis has failed for some reason and a re-solve is required from that point, or if a solution is to be deliberately continued by branching from a specified stage reached in a previous solution, restart options are provided for use when a new analysis is created in the Analyses treeview. See the heading 'Restart Options' in **Specifying an Initial State for an Analysis**.

Advanced options

Advanced output options can be specified as part of initially defining a Structural Analysis (or Thermal or Hygro-Thermal Analysis) and can be also accessed from the **Advanced** button on the Export Solver datafile dialog. These options determine what is optionally output to the **Solver output file** or to a **History file** for an analysis, each time a solve is carried out. Following results calculation for the whole model, the results written to a results file output can be stated to be for all elements in a model (full model), or for only a subset of elements (those part of a defined **group**) within a model. Choosing the latter produces smaller results files more quickly than would be created for a whole model.

A **Solver restart file** (.rst) can be created which enables a failed or terminated nonlinear, dynamic, transient or viscous analyses to be restarted from the last saved or chosen increment or loadcase.

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 **P-Delta Analysis**, **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.

❑ **Coupled Analysis** performs a **thermal/structural analysis** or a **hygro-thermal/structural analysis** either one after the other, or simultaneously, with a continuous transfer of information between the two analyses.

Other analyses

Other generally available or software product-restricted analyses (that do not require analysis controls to be specified) include:

❑ **Direct Method Influence Analysis** - used to create an influence line or surface by way of a unit load placed at each node or grid location on the loadable area of a structure.

❑ **Vehicle Load Optimisation** - used to identify critical vehicle loading patterns on bridges and apply these loading patterns to LUSAS analysis models.

❑ **Railway Load Optimisation** - used to identify critical train loading patterns on bridges and apply these loading patterns to LUSAS analysis models

❑ **Linear 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.

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.

Nonlinear Analysis

Nonlinear analysis controls are defined as properties of a loadcase. To do so, in the Analyses 📖 Treeview access the context menu for a loadcase and from the **Controls** menu item select the **Nonlinear and Transient** option to define the required controls. The **Nonlinear and Transient** object 🐢 created stores and provides the means to modify the control settings. Nonlinear analysis options can be specified separately for each loadcase and each analysis present. See **Nonlinear solution procedures** for more information.

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 when progressing a model 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 image) 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 image) the linear solution would fail to consider the **progressive eccentricity** of the vertical load on the bending moment diagram.
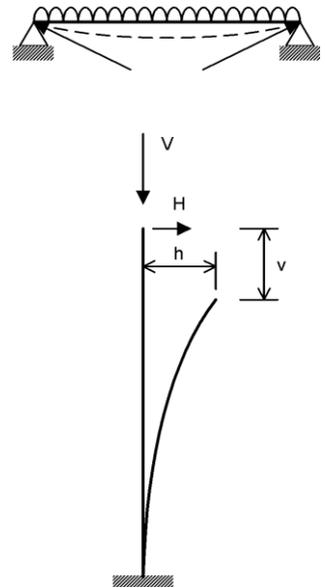
In both these cases depending on how large the deflections were, serious errors could be introduced if the effects of nonlinear geometry were neglected.

In LUSAS geometric nonlinearity can be accounted for using these formulations / analysis methods:

- ❑ Total Lagrangian
- ❑ Updated Lagrangian
- ❑ Eulerian
- ❑ Co-rotational
- ❑ P-Delta

Nonlinear analysis control options for these methods are accessed via the Nonlinear analysis options control object 🐛 in the Analyses 🔍 Treeview.

Total / Updated Lagrangian, Eulerian and Co-Rotational formulations

These 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.

For most elements these formulations are valid for small strains. For some elements the Updated Lagrangian and Co-rotational formulations are 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 a particular formulation 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*.

P-Delta analysis method

P-Delta analyses are approximate geometrically nonlinear analyses, and are typically used to take account of the interaction between vertical and horizontal (sway) loading on tall, slender structures. The use of P-Delta analysis is limited to a certain type of problem, that is, where axial (or membrane) stiffening/softening is important within a small displacement analysis. It is therefore not as generally applicable as a full geometric nonlinear analysis.

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. Note that it is possible to use a simplified linear lift-off support via support attributes also.
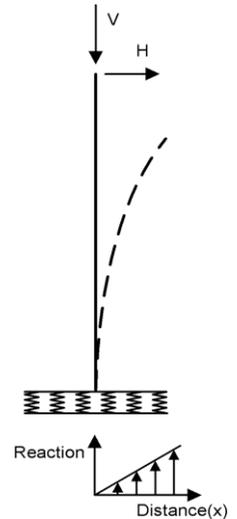
Materially Nonlinear Analysis

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 sudden 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.
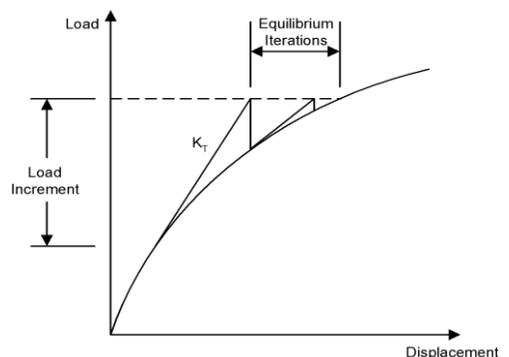
The iterative corrections are referred to some form of **convergence** criteria which indicates to what extent an equilibrium state has been achieved. Such a solution procedure is therefore commonly referred to as an **incremental**-**iterative** (or **predictor**-**corrector**) method shown in the figure 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. See **Specifying Manual or Automatic nonlinear loadcase controls** later in this topic for more information.

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.

Specifying Manual or Automatic nonlinear loadcase controls

The following flow chart can be used to decide if Manual or Automatic controls need to be defined for a nonlinear loadcase. Use total loads in each nonlinear step.
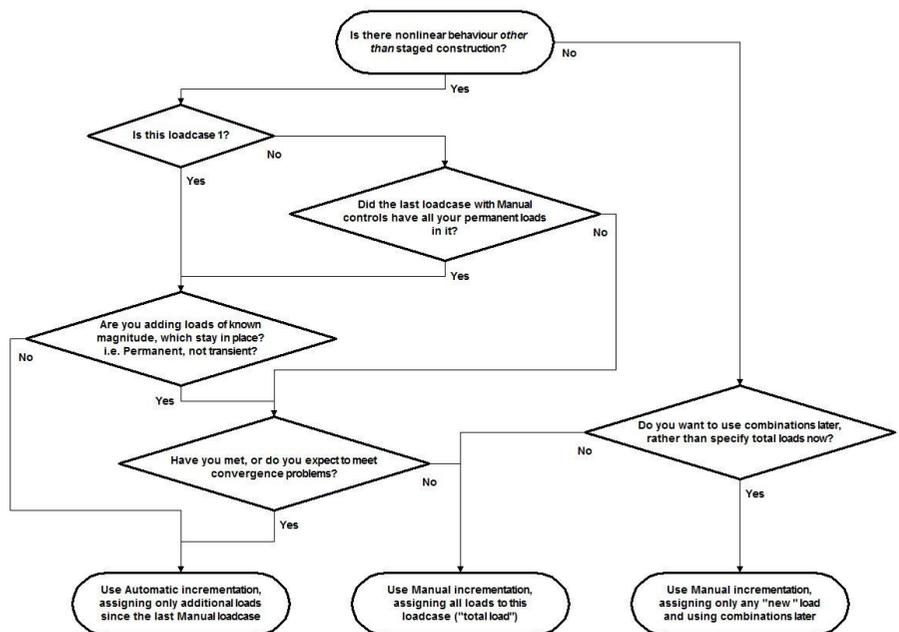
Flow chart to determine whether to use Manual or Automatic nonlinear loadcase control

(This chart is not applicable to time domain problems or those using load curves, where 'Manual' incrementation should generally be used.)

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 ![icon] Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object ![icon] 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 ![icon] Treeview access the loadcase context menu and choose the **Controls** menu item. The Nonlinear and Transient object ![icon] 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:

$$\Delta t \ \geq \ \frac{\gamma_\omega}{6\mathrm{E}k}(\Delta h)^2$$

where 'delta h' is the minimum distance between mesh nodes, which can be determined by selecting the two nodes and picking the Tools> Mesh> Distance between Nodes menu entry.

Additionally, for a consolidation analysis, both the elastic and two-phase soil properties need to be defined.

Worked example

- See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > ' 2D consolidation under a strip footing'.

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 by selecting the **Displacement reset** option on the Nonlinear and Transient control settings dialog for a loadcase, which can be accessed from the Nonlinear and Transient analysis control object 🐝 in the Analyses 🕒 Treeview.

Note that when performing a displacement reset a combination of stresses and strains may be saved depending both on the individual element formulation as well as the particular material model to allow the correct calculation of the forces in the element when the displacements are set to zero. In the case of joints the displacements prior to the displacement reset are saved and are added to the displacements measured from the reset shape to allow the correct calculation of the nodal stiffness and hence force.

Eigenvalue Analysis

Eigenvalue analysis controls are defined as properties of a loadcase. To do so, in the Analyses 🕒 Treeview access the context menu for a loadcase and from the **Controls** menu item select the **Eigenvalue** option to define the required controls. The **Eigenvalue** object 🐝 created stores and provides the means to modify the control settings.

Nonlinear analysis options can be specified separately for each analysis present. In a linear analysis eigenvalue control should be specified as part of the first loadcase.

Applications

- ❑ **Frequency analysis** extracts the natural modes of vibration of a structure.
- ❑ **Frequency (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.

It is possible to convert any assigned loading that acts on the model to mass. By default the mass will be applied on all directions. This can be changed by changing the parameter LDTOMA in the Model Properties : Solver System Variables tab. The following options are available:

- 1 = assume mass acts in all directions (default)

- 2 = assume mass acts in lateral directions only (the "vertical" direction is taken to coincide with the largest load component)

- 3 = assume mass acts in the load direction only

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:

$$\frac{\lambda_i^k - \lambda_i^{k-1}}{\lambda_i^k} \leq \text{rtol}$$

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:

$$\lambda = \omega^2 \qquad\qquad \lambda = (2\pi f)^2 \qquad\qquad f = \frac{\sqrt{\lambda}}{2\pi}$$

- 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. It is possible to define loads that remain constant and those that can vary for the computation of a load factor to cause buckling. 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 factor, 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.

When carrying out a linear buckling analysis, two load cases can be defined. The first contains any loading that is not to be factored in the buckling analysis (for example, the dead load), while the second contains the load that is to be factored during the buckling analysis (for example, the live load).

Eigenvalue buckling of stressed structures

Linear buckling analysis can be carried out after a static nonlinear analysis, but after the linear buckling analysis the analysis must terminate. Care needs to be taken when using the eigenvalue buckling facility at the end of a nonlinear incremental analysis. In a linear buckling analysis linearisation of the stress stiffness matrix leads to a linear relationship between load and load factor, but for the geometric nonlinear analysis (GNL) this is not the case if the live load specified is too large. As long as the live load specified is reasonably small, the buckling load computed should be consistent. Note that for some types of flexible models carrying out a buckling analysis at the end of a GNL analysis may not always yield useful results, and may lead to large numbers of local modes.

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, but close to it. 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. That is, the buckling load factor is greater than one, but close to it.

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 (for example by 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. Note that the mode shapes are 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
- ❑ Frequency or period vs acceleration/gravity

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 context menu for a loadcase and from the **Controls** menu item select the **Nonlinear and Transient** option to define the required controls. The Nonlinear and Transient object created stores and 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 $-\Delta t/2$, $\Delta t/2$

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, slidelines may be used for modelling impact in dynamic analysis. These permit 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 (or field) analysis can be defined initially when **creating a new model** by specifying a Thermal analysis type. It can also be created by adding a thermal analysis to an existing structural base analysis.

Two types of thermal (field) analysis may be performed:

- ❑ **Steady State** thermal (field) analysis
- ❑ **Transient** thermal (field) analysis

It is also possible to carry out a **thermo-mechanically coupled** analysis .

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 shown in the following table.

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

Carrying out a thermal (field) analysis

The modelling and solving of a thermal (field) analysis follows an identical process to that of modelling and solving a general structural analysis.

- Discretise the model using a series of field elements.

- Specify **thermal material properties**.

- Specify **thermal loads**

- Specify thermal link elements (assigning **thermal link material** and **thermal link geometric properties**) or **thermal surfaces** to define how heat is conducted, convected or radiated across gaps and spaces between different domains.

- Solve to determine the values of the field variable at each nodal point.

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 from the **Controls** menu item select the **Nonlinear and Transient** option to define the required controls. The **Nonlinear and Transient object** 🎯 created stores and 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.

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 λ_{max} 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 latent heat (of phase change) is introduced.

Latent heat (of phase change), 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 latent heat (of phase change) 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.

Notes relating to oscillations in transient field analyses

- The default value of the `beta` constant for a linear transient analysis provides the best accuracy but is susceptible to oscillation. Changing to `beta`=1, gives a more stable algorithm which eliminates the oscillations but potentially at the expense of accuracy. See Theory Manual for more details.

- The use of lumped specific heat (LUSAS option 105) in preference for consistent specific heat is essential, particularly in nonlinear analyses.

- Mesh refinement also has a significant effect in reducing spurious oscillatory response.

- The application of a step load to a structure will cause a thermal shock which will excite transient oscillations. Where possible, the use of the **Load Curve** facility to ramp the loading in a realistic manner would be recommended to reduce these initial transient effects.

- Any warnings (particularly those relating to pivots) in the LUSAS output file should be investigated closely for modelling errors.

Coupled Analysis

A coupled analysis can be defined initially when **creating a new model** by specifying a Coupled Thermal/ Structural analysis type or a **Coupled Hygro-thermal/Structural** analysis type.

A coupled analysis can also be created by adding a general thermal or hygro-thermal analysis to an existing structural base analysis, or adding a structural analysis to an existing thermal or hygro-thermal 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-coupled 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.

Coupled Hygro-thermal/Structural Analysis

A coupled hygro-thermal/structural analysis can be defined initially when **creating a new model** by specifying a Coupled Hygro-thermal/Structural analysis type. It can also be created by adding a hygro-thermal analysis to an existing structural base analysis, or adding a structural analysis to an existing hygro-thermal base analysis.

A coupled hygro-thermal/structural analysis models the time (hydration) dependent behaviour of concrete from time of casting to old age. It can:

- Take into account the time dependent environmental conditions when the concrete cures, such as daily cycles of temperature and humidity, to the desired degree of detail.

- Model the effect the formwork (and environment) has on the heat and moisture transfer to and from the concrete

- Predict the internal heat generated as the concrete cures so that any excessive temperature gradients that might cause the concrete to crack can be identified.

- Compute the amount of shrinkage in the concrete as it cures via the migration of moisture and the degree of saturation.

- Take into account the creep strains that occur during cure as the concrete is stressed.

- Model the evolution of properties E, fc, and ft.

- Predict failure when the concrete is fully (or partially) cured, and if a suitable mesh is used, obtain reasonable estimates of crack widths. See notes below for element

It assumes that the pore pressure remains constant (i.e. the rate of pore water evaporation is low in comparison to the rate of water vapour migration in the porous network). As a result it can only be used when concrete temperatures are under 100 degrees Celsius.

Carrying out a Coupled Hygro-thermal/Structural analysis

In LUSAS the procedure to carry out a coupled hygro-thermal/structural analysis is similar to running a coupled thermal/structural analysis. A hygro-thermal analysis is always nonlinear and transient, run over the time span of interest. The hygro-thermal analysis is run first and creates a data transfer file that is used by the subsequent structural analysis.

For a hygro-thermal analysis

❑ Define **hygro-thermal material properties**, including cement type and concrete mix proportions to represent the concrete (default values are provided). During the analysis these are used in the computation of heat generated as the concrete cures.

❑ Define **time dependent boundary conditions** to represent the environmental conditions (temperature and humidity) and the formwork material (convection heat transfer coefficient)

The temperature, degree of water saturation and degree of hydration are computed in the hygro-thermal analysis at each time step, and these are written to a data transfer file that is read by the subsequent structural analysis.

For a structural analysis

All material properties must be defined with respect to a concrete age of 28 days. At earlier times, reduced structural properties are computed to account for the fact that the concrete has not fully cured.

When using the transient, smoothed, multi-crack nonlinear concrete model 105 The structural analysis reads the temperature, degree of water saturation and degree of hydration data at each time step and uses the temperature to compute thermal strains, the degree of saturation to compute shrinkage, and the degree of hydration to compute the mechanical properties of the concrete. The creep strain in the concrete is also computed, using either a User defined creep coefficient or that defined by the CEB-FIP (1990) or EN1992-1-1:2004 design codes.

During the step-by-step structural analysis the effects of thermal, shrinkage and creep strains, along with time (hydration) dependent structural material properties can be combined with any further structural loading that may be applied. At each time step the transient smoothed multi-crack nonlinear concrete model 105 is used to predict if cracking will occur.

If cracking is not of interest, but creep and shrinkage are to be considered, use concrete model 86, or a very high tensile strength or a linear material.

Notes

- 2D plane, 2D axisymmetric solid, and 3D solid hygro-thermal elements are available. These can have either 'hygro' material properties or isotropic thermal properties. In the latter case, to model non concrete material (e.g. steel), the hygro 'stiffness' is computed with a permeability of about six orders of magnitude less that the concrete permeability. There are no thermal or hygro-thermal shell elements.

- Coupled thermal/structural bar elements can be used. Steel shell elements can be used in the structural analysis but these will only read in temperatures at nodes that are shared with the concrete (any unshared nodes receive zero temperatures). Note that such bar and shell elements occupy space already occupied by the concrete. This would introduce an error in the effective values of the material properties. To avoid this, the material properties of the embedded bars and shells can be recalculated, e.g. density of steel reinforcement equals steel density minus concrete density.

- When the purpose of the analysis is to check if cracking would occur and compute the crack widths mesh refinement using a denser mesh, and time step convergence studies using a sufficient number of load steps should be done. Start with small load steps as the initial orientation of cracks tends to propagate, as in real life. Linear order elements are recommended for crack width results. An element length of greater than or equal to the reinforcement

cover is suitable for linear elements. An element length of greater than or equal to twice the reinforcement cover is suitable for quadratic elements.

Benefits of a Coupled Hygro-Thermal/Structural analysis over a Coupled Thermal/Structural analysis

The benefits of running a coupled hygro-thermal/structural analysis over a coupled thermal/structural analysis in which heat of hydration, codified creep and shrinkage, and a suitable nonlinear concrete model are combined. are:

- Shrinkage is computed more accurately in a coupled hygro-thermal/structural analysis. Codified creep and shrinkage models are aimed at beam and shell like structures, and require the input of a cross-section area/perimeter ratio, and the shrinkage for any given member would be constant throughout that member. This is clearly inappropriate for a stocky (dam like) structure.

- The coupling effect of the degree of saturation (or capillary pressure) and temperature is not accounted for in a thermal analysis.

- The environmental boundary conditions used in a thermal analysis are less accurate, for instance no account is taken of the effect of humidity

- The evolution of the mechanical properties used in the coupled hjygro-thermal/structural analysis is a function of the degree of hydration of the concrete, which will vary throughout a member. The codified creep and shrinkage mechanical properties are a function of time only; they will be constant throughout a member and are therefore likely to be less accurate.

- The rate at which the concrete cures affects the long term strength of the concrete (a faster cure rate produces higher temperature in the early stages and leads to a lower long term strength). A coupled hygro-thermal/structural analysis will take this "cross-over effect" into account whereas the use of design code equations will not pick it up.

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 followng 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.
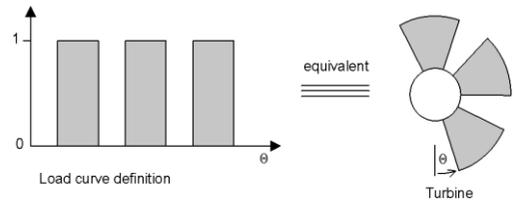
Thermal Problems

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 allows the visualisation of the variation of a load effect such as reaction, shear, moment, or deflection at a specific location in a structure due to a unit concentrated load moving over the structure. This makes it possible to identify regions within a "loadable area" where application of this unit load leads to effects which might be regarded as "adverse" or "beneficial" at the location of interest which may be close to, or remote from, the loadable area. This is useful to help determine the most onerous patterns of transient (live) loads in structural analysis and design, and is used by the LUSAS Vehicle Load Optimisation facility and the LUSAS Rail Load Optimisation facility for that purpose. Additionally, for some design codes, traffic or train 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.

With influence analysis, the influence results (and therefore the Vehicle Load Optimisation and Rail Load Optimisation results) are based on the properties of the **base analysis**. Because of this it is a requirement that the base analysis is a simple linear static analysis.

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. The **Reciprocal Method** provides options to solve influences for grillage, slab and deck models, for shear, moment, reaction and

displacement influence types. Solving a Reciprocal influence will take an increasing amount of time as more influence locations are solved.

❑ **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. Unlike the **Reciprocal Method** (where influence types are limited to Shear, Moment, Reaction and Displacement) the list of available results entity / components available for influence calculation is only limited by the elements used within the model. Solving an influence analysis using the Direct Method will take the same time regardless of the number of influence locations solved, and the solution time will depend upon the grid or mesh density used.

❑ A **Direct Method Influence Analysis for General use** can calculate influences for shell/plate/grillage models; for 3D beam models including torsional effects; calculate influence for user-defined results components; calculate influence for derived results, as a post-processing facility; and can calculate influence for many locations and quantities with a single analysis.

❑ A **Direct Method Influence Analysis for Rail use** can calculate influences for shell or plate models; calculate influence for user-defined results components; calculate influence for derived results, as a post-processing facility; and can calculate influence for many locations and quantities with a single analysis.

In summary, the reciprocal theorem method provides less functionality than the Direct Method, but it is much faster when only a limited number of influences are required at strategic locations. It is also 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. The Direct Method provides more general functionality, and must be used if either 3D line beam modelling or Rail Load Optimisation analysis is to be carried out.

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** and **Rail Load Optimisation** analysis requires influence attributes to be assigned to a model in order to generate influence surfaces and calculate the most adverse vehicle and train 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.

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 <u>any</u> 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. Influence Analysis for the Reciprocal Method and for the Direct Method influence attribute assignments can be solved in isolation from any other analyses. Influences for the Reciprocal Method can also be solved selectively by choosing the **Influences to Solve** context menu for an influence entry in the Analyses ◔ Treeview.

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, if desired, 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. For Rail Load Optimisation uses, the influence shape for a particular track layout can only be displayed if that track layout is set to be visible.

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

- 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.

Reciprocal Method

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.

Direct Method

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, or permitted). 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. Having a grid density finer than the actual mesh does not improve the accuracy of the DMI results and may miss the peak values.

In order to accurately decide if the influence profile for an adverse area meets any codified cusping rule, a Direct Method Influence analysis must be defined as follows:

- For shell/plate/grillage models the influence node (the location where cusping occurs) and the DMI grid node must have the same coordinates for so that the supported nodes are loaded with the unit load to correctly identify the start/end of adverse areas and to combine them to get the most onerous effect. For these models the most accurate results are obtained when the Direct Method Influence Analysis setting 'Nodes in search area' option is chosen.

- For beam models the influence node (the location where cusping occurs) and the DMI grid node need to have the same longitudinal position. For these models the most accurate results are obtained when the Grid Settings option 'Match nodes in search area' is chosen.

Direct Method Influence Analysis - General

A Direct Method Influence Analysis for general use requires **Direct Method influence attributes** to be defined and assigned to a line beam, frame, grillage, or shell/plate 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.

Materials, sections, supports and other attributes for a DMI analysis are taken from the **base analysis**. Because of this, it is a requirement of a DMI analysis that the base analysis is a simple linear static analysis.

Load locations

❑ **Search area** - A **search area** can be used to limit the features over which the prescribed load is applied so that the effect of the load on other features in the model may be prevented. 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.

- For shell/plate/grillage models, it is recommended that the DMI analysis should be defined with the 'Nodes in the search area' selected.

- For line beam models where the geometric beam section represents a loadable top slab, it is recommended that the Grid option be used instead,

specifying Grid settings to automatically create grid points to match the longitudinal positions of the nodes in the search area.

❑ **Grid** - predominantly used with line beam models where the geometric beam section represents a loadable top slab. For this, a defined 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. See the worked example 'Vehicle Load Optimisation of a Box Beam Bridge', which provides more details on the use of a loading grid.

❑ **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 loadable grid to be generated (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. See **Direct Method Influence Attributes** for more details.

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.

Results file scope

Following results calculation for the whole model, the results that are written to a results output file can be restricted to be those only for any assigned influences (loadcases), or for those assigned influences (loadcases) and a subset of elements (those part of a group) within a model. Restricting the results output produces smaller results files more quickly than would be created for a whole model, but note that if subsequent influence assignments are made to the model a Direct Method Influence analysis would have to be run again because results for those influence locations would not be present in the previously calculated results file.

❑ **Assignments only** calculates and writes results to a results file for only the assigned influences (loadcases) in the model.

❑ **Assignments and additional group** calculates and writes results to a results file for only the assigned influences (loadcases) in the model, and for those elements within the chosen group , and automatically includes the elements needed to provide results for assigned influences.

Direct Method Influence Analysis - Rail

A Direct Method Influence Analysis for use with the Rail Load Optimisation facility requires **Direct Method influence attributes** to be defined and assigned to a shell or plate model. Note that line beam models are not currently supported by the RLO facility. The locations at which a concentrated 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.

Materials, sections, supports and other attributes for a DMI analysis are taken from the **base analysis**. Because of this, it is a requirement of a Rail DMI analysis that the base analysis is a simple linear static analysis.

Load locations

❑ **Track layout** - Previously defined **track layouts** can be selected for inclusion in the influence analysis.

❑ **Search area** - A **search area** can be used to limit the features over which the concentrated load is applied so that the effect of the load on other features in the model may be prevented. If a search area is not specified the default loading area is **Whole model**.

Advanced settings

The magnitude of the loading applied for an influence analysis can be modified. Note that for a Rail DMI analysis there is no requirement to specify a loading direction.

Notes

• Unlike **Vehicle Load Optimisation** (where the traffic loads can be placed anywhere between the selected kerbs defining a width of carriageway) **Rail Load Optimisation** loading can only follow the track definitions defined for each **track layout**. Instead of the grid of virtual points that is used by the Vehicle Load Optimisation facility to represent the loadable region of the slab

that is to be loaded with a unit point load as part of an influence analysis, for a Rail Direct Method Influence analysis, nodes are created automatically where each rail intersects with an element in the search area. In this way, the number of locations where a unit load can be applied is minimised, and ensures that the influence shapes are drawn following the alignment of each rail track.

- Modifying a rail track layout that is used in a Rail DMI analysis will invalidate the results of that analysis resulting in an invalid results symbol (a red circle with a line through it) appearing alongside any influence results loadcases in the Analyses treeview, requiring a re-solve to be done.

Vehicle Load Optimisation

Vehicle load optimisation analysis is available in Bridge software products only. It is accessed from the **Analyses > Vehicle Load Optimisation** menu item.

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.

See **Vehicle Load Optimisation** in *Application Manual (Bridge, Civil & Structural)* for more details.

Rail Load Optimisation

Rail load optimisation analysis is available in Bridge software products only. It is accessed from the **Analyses > Rail Load Optimisation** menu item.

Rail load optimisation is used to identify the most onerous rail loading patterns for specified track layouts 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 rail bridge structures.

See **Rail Load Optimisation** in *Application Manual (Bridge, Civil & Structural)* for more details.

Cable Tuning Analysis

Linear and nonlinear cable tuning analysis is available in selected Bridge and Civil & Structural software products only.

- Linear cable tuning analysis can be set-up by use of the **Analyses > Linear Cable Tuning Analysis** menu item.

- Nonlinear cable tuning loadcases can be set-up by use of the **Analyses > Nonlinear Cable Tuning Loadcase** menu item.

See **Cable Tuning Analysis** in the *Application Manual (Bridge, Civil & Structural)* for more details.

LUSAS Solver Types and Options

Choosing a Solver

The Solver type and associated options can be specified by selecting the **File > Model Properties** menu item, selecting the Solution tab of the Model Properties dialog, and choosing the Solver Options button. The Model Properties dialog can also be accessed from the Model Properties control object in the Analyses Treeview. The main solver types are:

- **Frontal (direct) solvers** are more robust than iterative solvers and are applicable to all types of analysis. Frontal solvers should always be used for very **ill-conditioned** problems since the time taken to obtain a solution is independent of the problem conditioning.
- **Parallel solvers** high-performance, robust, memory efficient solvers for solving large sparse symmetric and non-symmetric linear systems of equations (only) on shared memory multiprocessors
- **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 (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 are only applicable for a single loadcase

Solvers supported

The current solver options supported are:

- ❑ **Default** The default solver is the standard Frontal solver. If the Fast Solvers software option has been licensed the Fast Multi Frontal solver is used by default instead.
- ❑ **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.
- ❑ **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 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.

❑ **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.

Fast Parallel Solver settings

The Fast Parallel Solvers can be configured using the following parameters:

- ❑ **In Memory Solution** When selected produces a solution within memory. When not selected solves out-of-memory.
- ❑ **Message Level** options are None, which provides no output; or Statistics, which prints data to the Solver window.
- ❑ **Pivot Tolerance** (Parallel Direct Solver Only) Default value is 1e-12
- ❑ **Pivoting Method** (Parallel Direct Solver Only) options are Diagonal pivoting (the default) or Bunch and Kaufman pivoting.
- ❑ **Maximum number of iterative refinement steps** (Parallel Direct Solver Only) The default is 10.
- ❑ **Stopping criteria tolerance** (Parallel Direct Iterative Solver Only) The stopping criteria in Krylow-Subspace iteration. The default is 1e-6.

If a model with constraint equations fails to solve using a Fast Parallel Direct Solver, modifying some of the Fast parallel Solver parameters will enable the analysis to complete successfully, If necessary, contact LUSAS Technical Support for assistance.

Iterative (Conjugate Gradient) solver settings

The Iterative (Conjugate Gradient) solver can be configured using the following parameters:

- ❑ **Drop tolerance** This is 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** This is 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 additional check box options are also available:

❑ **Output iterative solution information**

❑ **Extra preconditioning with a hierarchical basis**

❑ **Keep global matrix assembly in-core**

❑ **Minimise memory usage** forces the Fast Solver to use the minimum amount of memory possible.

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*

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 platform selection

If 32 and 64 bit versions of LUSAS Solver have been installed, you can specify which version of LUSAS Solver should be used with your current instance of LUSAS Modeller:

- ❑ **Follow Modeller** - use the same version of LUSAS Solver as being used by LUSAS Modeller
- ❑ **32 Bit** - use the 32 bit version of Solver
- ❑ **64 bit** - use the 64 bit version of Solver

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 coordinate systems between the finite element model and any engineering drawing(s) 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. For nonlinear analyses the loadcase of the last solved increment will be set active by default. For eigenvalue analysis the first eigenvalue will be set active.

Using a 32 bit or 64 bit Solver

The default behaviour when the Solve Now button is selected within 32 bit Modeller, is to run the 32 bit Solver. See the *Installation Guide* for details of how to run the 64 bit Solver when using the 32 bit Modeller.

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 the 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

- Link 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 and Processing 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, and setting of fibre location and a composite lamina. It provide a basic overview of results components, results entities and results display capabilities.
- ❑ **Results Transformation** provides options for transforming results to a consistent or alternative coordinate system

Results Processing

Results 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 can be manipulated to create results at particular model locations for a particular results viewing use.

Further Calculations

- **Combinations and envelopes**
- **Design combinations**
- Fatigue analysis of a linear model
- **IMDPlus analysis**
- **Modal analysis**
- **Design factors**
- **Target values**
- **Steel frame design**
- **RC frame design**

- **Wood - Armer reinforcement**
- Beam stress resultant from beam and shells
- **RC Slab Design**
- **Crack width calculation to EN1992-1-1**
- Expansion of Fourier analysis results
- **Composite failure criteria**
- **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**
- **Inspection locations**
- **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
- Reporting on individual features (one-click reporting)
- **Generating a Main Report**
- **Generating a Design 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. For each view window, the last used loadcase is set active by default.

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.
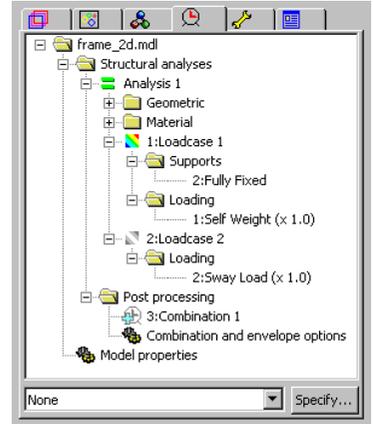
The Analyses 🕒 Treeview (right) shows a typical listing obtained when a results file is loaded for a model.

Opening a results file without a model being open

A results file can be opened in isolation from a model by using the **File> Open** menu item. By changing the Files of type drop-down list, results files (.mys) can be selected.

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 were saved with the model file.
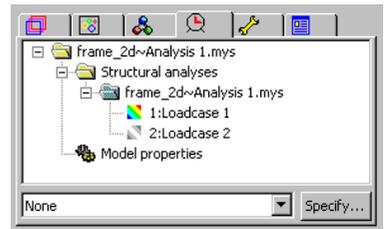
When a results file is opened in isolation from the model from which it was created the results data and stored mesh data is loaded from the .mys file. This method is used when access to model information, such as feature definitions or group names, is not required or is not available for results-processing.

The Analyses 🕒 Treeview (right) shows the listing obtained for the same analysis when only the result file is loaded .

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 file selected was the first results file 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. To set a loadcase active either:

- Right-click on the loadcase in the Analyses 🔍 Treeview, and choose **Set Active** from the context menu, or:
- Drag and drop a loadcase from the Analyses 🔍 Treeview into a view window to set that loadcase active in that view.
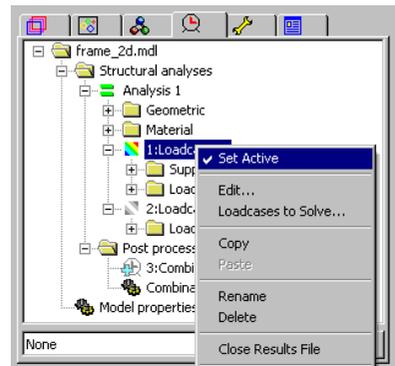
In this way a single window is used to plot results from a single loadcase, and multiple windows can be used to compare results from different loadcases.

Setting the Active Fibre Location

For plotting values and /or diagram results of stress (this step is not required for forces and moments) 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 be visualised by double-clicking on the Geometric Line name and selecting the Visualise button on the dialog presented.
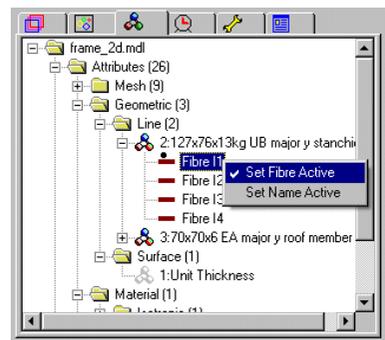
Choosing the Set Fibre Active option shows results for that fibre in just that attribute. Setting the Name Active shows results for all similarly named fibres in other attributes throughout the model.

Setting the Active Composite Lamina

With composites analysis, in order to plot results on composite 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 lamina information can be seen and visualised by double-clicking on the Laminate stack name and selecting the Visualise button on the dialog presented.
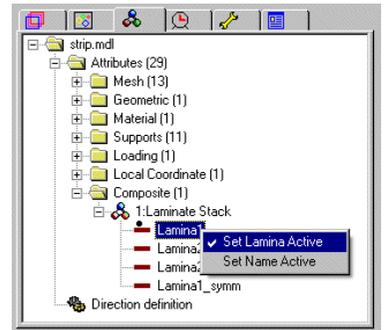
Results Entities (Types)

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 the Results Tables of the *Element Reference Manual*.

To view results a chosen results entity must be specified. Only results 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		**Thermal Analysis**
❑ **Displacement**	❑ **Plastic Strain**	❑ **Potentials**
❑ **Force/Moment**	❑ **Creep Strain**	❑ **Fluxes**
❑ **Stress**	❑ **Rubber Stretch**	❑ **Gradients**
❑ **Strain**	❑ **Strain energy**	❑ **Thermal Surface**
❑ **Loading**	❑ **Plastic work**	
❑ **Reaction**	❑ **Slideline**	
❑ **Reaction Stress**	❑ **Named variables**	
❑ **Residual**	❑ **State variables**	
❑ **Velocity**	❑ **Slab Design (RC)**	
❑ **Acceleration**		

Results Components

For each results entity a results component (Fx, My etc) will also need to be chosen. See the Results Tables in the Element Reference Manual for component definitions. User-defined results components will appear within the entity within which they were placed..

Results Display options

By default, results are averaged respecting discontinuities formed by geometry orientation, geometric assignments and material assignments. Default averaging settings can be changed on the Model Properties - Options (Averaging Options) dialog. Unaveraged nodal results are displayed on an element-by-element basis, and will show inter-element discontinuities. This is useful for checking possible mesh discretisation errors and for displaying results across geometry, material or other discontinuities (if averaging for these situations has been otherwise allowed).

- Results visualisation is performed using results **layers** in the Layers Treeview. **Viewing results** requires the relevant layer to be present for the active current window. The properties of the results layers may then be set to display the required entity and component in a specified style.

- **Inspection locations** can report on results for elements that belong to the assigned feature, and also for those elements that belong to topologically connected features.

- Feature maximum, Feature minimum, and Feature extreme values can be reported.

- When **section slices** have been created through a solid model, results can be displayed on those slices

- When an **envelope or a smart combination** is set active, options to view maximum, minimum or extreme results are available.

- Sometimes it may be necessary to transform the results, for example if the elements in a model are orientated such that their local axes vary from one another. Numerous options are available. See **Results Transformation** for details.

Notes

- Stresses are stress resultants for beams, plates and shells

- For plates and shells top, middle and bottom stress and strain are available.

- When applicable, 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 many context menus, including those for groups, attributes, and selection details.

- See **Results Transformation** for general notes regarding the orientation of results.

Results Averaging

Finite element results are calculated for each element individually. With a sufficiently fine mesh, the results of adjacent elements are usually sufficiently similar that the eye sees no visible difference at the boundary between adjacent elements. But when a mesh is coarse, the results of adjacent elements can be quite different. Where an engineer is satisfied that such differences can be safely ignored, it can sometimes be desirable to show a smoothed result, averaging the contributions from adjacent elements at their common nodes. However, there are times when averaging is not appropriate. For example, it is generally considered invalid to average across a boundary between elements that have different materials, or different thicknesses, or at a junction where elements lie in different axes or planes.

LUSAS employs rules to spot situations where averaging of results is not appropriate. For each pair of elements that is considered, if all the averaging rules "pass" then averaging of results across those elements is permitted. If any averaging rule "fails" then a discontinuity is said to exist between those elements, and averaging will not be permitted. The user may choose to switch rules on or off for finer control.

Averaging rules

The following rules exist, where each rule considers the common boundary between exactly two elements.

- Any elements that are meshed in the same feature are assumed to be suitable for averaging. This is a fundamental rule and as such does not appear in the user interface. The presence of this implicit rule means that all the rules below can assume that the elements are from different features.

- The **Feature** rule is a very conservative strategy. It assumes that averaging should only be permitted where elements reside in the same feature (line, surface or volume). This rule is not on by default. Switching this rule on makes all the other rules below irrelevant.

- The **Different assignments** rule dictates that a discontinuity exists wherever the two parent features have any difference in their attribute assignments, such as material assignment, age, and loading to name a few. For example, if element 1 belongs to surface 1 and element 2 belongs to surface 2 and a load is applied to element 1 in loadcase 1, and that same load is applied to surface 2 in loadcase 2, the assignments are therefore different, and a discontinuity is assumed to exist between element 1 and element 2 in all loadcases.

- The **Common LOF assignments** rule dictates that a discontinuity exists wherever there is any attribute assignment to the feature (the lower order feature) that forms the common boundary between the two features being considered. For example, if both elements are surface elements, then they each belong to a surface. Since they belong to different surfaces (see the first rule), and they are adjacent, then there is a common line separating them. If that line

has any attribute assignment whatsoever, then averaging is not permitted between those two elements. Note there are three specific exceptions: NULL mesh assignments, "assign to all" assignments, and "Default" assignments. Note also that no check is made to see if the assignment(s) apply in the active loadcase.

- The **Visibility** rule is not applied (turned 'on') by default. It is provided for those users who wish to emulate the old V15.2 behaviour that averaging was carried out on all visible elements. When this rule is on, and one element is visible and the other is invisible, a discontinuity is assumed to exist. Thus, averaging will not include invisible elements. Making elements visible and invisible will change the displayed results.

- The **Junction** rule dictates that a discontinuity exists wherever three or more features meet at the same common feature. For example three lines meeting at a point, or three surfaces meeting at a line.

- The **Angle exceeds** rule dictates that a discontinuity exists wherever element axes differ by an angle greater than the threshold input.

Results averaging options are accessed via the Model Properties dialog or the Model Properties control object 🐷 in the Analyses 🕒 Treeview. Averaging options are specified to be common for all analyses.

Results Transformation

Viewing Results in a Consistent Direction

⚠️ **When viewing results in LUSAS 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.

Transforming results

❑ Results displayed on contour, values or vector layers in a particular view window can be transformed using the Transform **Set...** 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 **Set...** 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 of 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.

❑ **Assigned results transformation attribute** Transforms element results to the local element material directions, or to directions specified by any assigned **results transformation attributes**.

❑ **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, or theta/phi, r/theta or r/phi.

- Using the transformation options for joint elements has no effect.

- For nodal results (displacements and reactions) only Global and Local element / node (transformed freedoms created by local coordinates assigned to the feature) options are available.

- With the exception of the **No transformation applied** option the type of transformation used is written to the Contour key.

Sign conventions

Sign conventions in LUSAS are based on established theory and historical precedent, which means different conventions are often adopted for different contexts.

- For beam element results, moments are taken about the axis i.e. Mx is the moment about the x axis.

- For shell elements, moments are taken in the direction of the axis i.e. My is the moment about the x axis.

- For results on slices through a model, the sign convention differs depending on whether the 'Slice Beam/Shell resultants' or 'Section through a 3D model' facilities are used.

  - For **Slice Beam/Shell resultants**, the orientation of the slice local axes follows the selected lines and is in accordance with beam theory where the beam local x axis is orientated along the beam. The slice local x axis will be defined perpendicular to both the slice local y and z axes.

  - For **Section through a 3D model** slice local x and y axes lie in the plane of the slice. The slice local z axis is perpendicular to both the slice local x and y axes. Reversing the local axes of the slice will re-orientate the z axis as well as the x and y local axes.

For information on sign conventions for all element types see *Element Reference Manual* Appendix D.

Shell plane for resultants

The following diagram illustrates the shell planes (shaded) that are available for a cylindrical coordinate system.

theta / z   r / theta   r / z
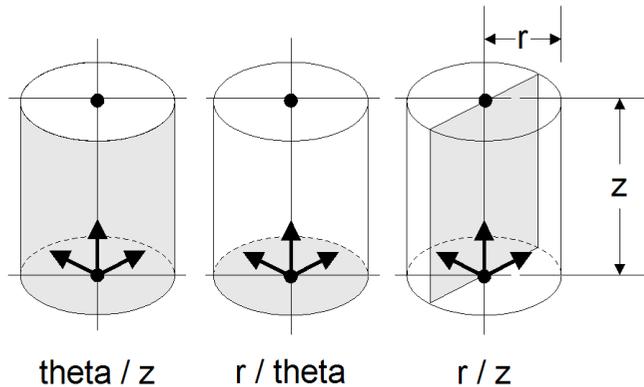
The following diagram illustrates the shell planes (shaded) that are available for a spherical coordinate system.

theta / phi   r / theta   r / phi
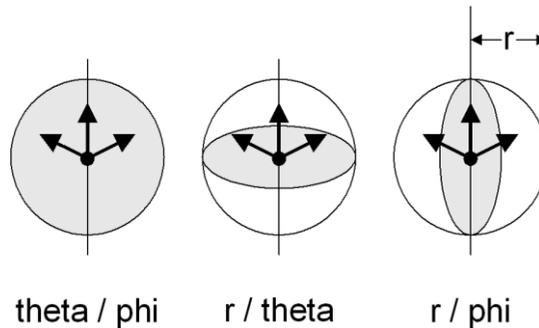
Raw and Derived components

When selecting a primary component of interest for investigating results for a particular effect, either 'Raw solver components' or 'Derived components' may be chosen.

- **Raw solver components** are those that are calculated for each element type by LUSAS Solver. For example, for a 2D Thick beam, they are Fx, Fy, Mz.

- **Derived components** are created by LUSAS Modeller by mathematically manipulating the raw components, and include components such as absolutes, equivalents, principals, Wood-Armer, User Defined Results, Steel frame and RC frame design calculations, and others.

In LUSAS Modeller raw components are always listed for selection according to the elements in use. Wherever there is no ambiguity of use, derived components will also appear alongside raw components in results drop-down menu listings, and can be selected without any further user-involvement.

By default, LUSAS will automatically suppress the listing of derived components where the definition of such may be misunderstood. This issue affects all smart combinations, and also when a combination includes envelopes within its definition. The issue only occurs when using a derived component as a primary component, or inspecting a derived component individually. Such derived components can be made available for selection by using the corresponding checkbox, once the program behaviour has been understood.

The following table shows when derived components are made available for selection with particular combination and envelopes scenarios..

Derived components are made available for	Derived components are suppressed for
Envelope of loadcases	Envelope of Smart Combinations
Envelope of envelope(s)	Basic Combination of Envelope(s)
Envelope of Basic Combination(s)	Basic Combination of Smart Combination(s)

Viewing results for suppressed derived components

Viewing, graphing, printing or reporting on results of any derived components that have been suppressed is possible by choosing the '**Allow derived components**' check box on the respective dialogs on which it appears. The results obtained will be reasonable in many circumstances; however, as described below, this is not always the case and so the ability to access results from derived components in the situations stated must be used with caution. When suppressed derived components are accessed, it is the user's responsibility to judge whether or not the approach used by LUSAS Modeller is appropriate to the structure and analysis in question.

For a Smart Combination, or any basic combination or envelope that contains one or more smart combinations

A smart combination uses an adverse (or variable) factor for a constituent loadcase if the effect of that constituent loadcase is onerous; where "onerous" means the constituent loadcase gives an effect of the primary component of the same sign as is being optimised for.

It may be expected that a smart combination will give you the most onerous possible permutation for the primary component. If the primary component is a linear solver component, then this is what will happen. However, if the primary component is a derived component, then this is not necessarily the case, particularly where different constituent loadcases have values of differing sign. This is because the derived component will be recomputed by the combination, from the raw solver components, after all the loadcases have been factored and combined.

Consider the following example:

- Loadcase 1 at node 1 has displacement DX = 10 and DZ = 10, giving RSLT = 14.14

- Loadcase 2 at node 1 has displacement DX = -10 and DZ = -10, giving RSLT = 14.14

- Smart Combination 1 has a beneficial load factor of 1 and an adverse load factor of 1.5 for Loadcase 1; and a beneficial load factor of 0 and an adverse load factor of 1 for Loadcase 2

- LUSAS computes RSLT for each constituent loadcase, in order to work out which are onerous. In this case, for a maximum smart combination, both loadcases are onerous, because they have positive values of RSLT, therefore they will both be given their adverse factors.

- LUSAS then computes the "raw" values DX, DY and DZ for the smart combination, using the factors as computed in the previous step. This gives DX = (10 * 1.5) - (10 * 1) = 5 and DZ = (10 * 1.5) - (10 * 1) = 5.

- Finally, LUSAS computes the requested derived component, RSLT, using the usual formula, in this case RSLT = 7.07.

- But, in this case, the most onerous permutation would have been for Loadcase 1 to receive its adverse factor, but Loadcase 2 to not, i.e. DX = (10 *1.5) - (10 * 0) = 15 and DZ = (10 * 1.5) - (10 * 0) = 15. Together these give RSLT = 21.21.

Hence the most onerous permutation may be missed.

For a basic combination that contains one or more envelopes

When a basic combination contains an envelope, the primary component to be used in the envelope must be set. It may be expected that the combination will give the most onerous possible permutation (which contains any one loadcase from each envelope) for the primary component. If the primary component is a linear solver component, then this is what will happen. However, if the primary component is a derived component, then this is not guaranteed to be the case. This is because each envelope is regarded as a 'self-contained' unit that provides an answer to the combination above it. As a result, the envelope computes the "worst" loadcase for the primary component (irrespective of any other constituents of the combination), and the combination uses the associated values from that loadcase, factors them, and combines them with the results from other loadcases / envelopes in the combination.

For derived components, it is possible that the effect of the "worst" loadcase from one envelope reduces that from another constituent, meaning that the worst permutation is missed.

Consider the following example:

- Loadcase 1 at node 1 has displacement DX = 10 and DZ = 10, giving RSLT = 14.14

- Loadcase 2 at node 1 has displacement DX = -10 and DZ = -10, giving RSLT = 14.14

- Loadcase 3 at node 1 has displacement DX = 1 and DZ = 1, giving RSLT = 1.414

- Envelope 1 contains Loadcases 2 and 3

- Combination 1 contains Loadcase 1 and Envelope 1 (both with a factor of 1)

- With RSLT as the primary component, Envelope 1 (max) will pick Loadcase 2, as it gives the biggest RSLT. This gives us a value for Combination 1 DX = 10 - 10 = 0 and DZ = 10 - 10 = 0. Together this DX and DZ gives RSLT = 0.

- But the most onerous permutation would have been for Envelope 1 (max) to pick Loadcase 3. This gives in Combination 1 DX = 10 +1 = 11 and DZ = 10 + = 11. Together these give RSLT = 15.56.

Basic and Smart Combinations, and Envelopes

Basic and Smart Combinations, and Envelopes enable the results from several loadcases to be presented as a single set of results.

Use the **Design Combination** wizard to assign LUSAS loadcases, envelopes or basic combinations to predefined load types for a supported design code. Based on the assignments and settings made, load combinations for those load types are automatically generated by LUSAS Modeller, rather than having to be user-defined one-by one. Only loadcases suitable for combining or enveloping will be made available for inclusion in a combination or envelope.

Basic combinations

Basic combinations are defined by selecting the **Analyses > Basic Combination** menu item. 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.

Options

❑ **Method** provides the means to specify a combination method

- o **Factored** (the default option) - The sum of all included loadcases, where each loadcase is multiplied by its specified load factor. For example: (Loadcase 1 x Factor 1 + Loadcase 2 x Factor 2...) etc.

- o **Average Combination** - The average of the sum of all included loadcases, where each loadcase is multiplied by its specified load factor. For example: (Loadcase 1 x Factor 1 + Loadcase 2 x Factor 2...) etc. / number of included loadcases

- o **SRSS Combination** A combination of the square root of the sum of the squares of all included loadcases, where each loadcase is multiplied by its specified load factor. For example: The square root of {(Loadcase 1 x Factor 1)^2 + (Loadcase 2 x Factor 2)^2... } etc.
- o **Absolute Combination** The absolute sum of all included loadcases where each loadcase is multiplied by its specified load factor. For example: The absolute value of (Loadcase 1 x Factor 1) + The absolute value of (Loadcase 2 x Factor 2) + etc.

Smart combinations

Smart combinations are defined by selecting the **Analysis > Smart Combination** menu item. 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.

Beneficial / Adverse and Permanent / Variable load factors

For new models, by default, the smart combinations dialog is presented to allow input in terms of Beneficial/Adverse factors rather than Permanent/Variable factors. However if the 'Display beneficial/adverse factors' check box on the 🐞 Combination and envelope options dialog is unchecked, the **Beneficial load factor** and **Adverse load factor** names may be optionally displayed as **Permanent load factor** and **Variable load factor** and appropriate factors input.

Note that in LUSAS, when inputting permanent and variable load factors:

- Permanent load factor = Beneficial load factor.
- Variable load factor = Adverse load factor - Permanent load factor.

For example, if a design code requires permanent actions to be factored by 1.35 where adverse and 1.0 where beneficial, and unfavourable variable actions to be factored by 1.5 where adverse, and omitted where beneficial:

- For permanent actions this is equivalent to a Permanent load factor of 1.0 and a Variable load factor of 0.35 since 1.0 will always be applied and 0.35 will only be applied if it creates an unfavourable effect.
- For variable actions this is equivalent to a Permanent load factor of 0.0 and a Variable load factor of 1.5 since the loading will only be applied if it creates an unfavourable effect.

Note also that for permanent and variable load factors:

- 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 a summary of 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 a summary of permanent and variable factors for negative load effects.

Saved models retain the input type and values specified.

Other options

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.
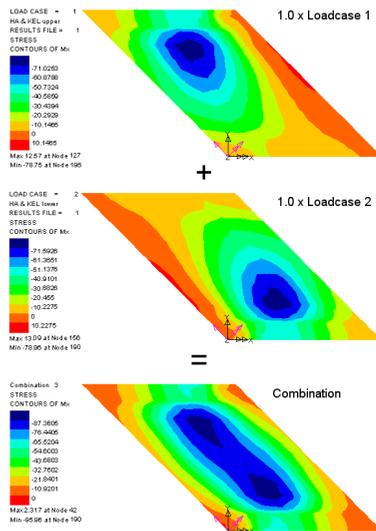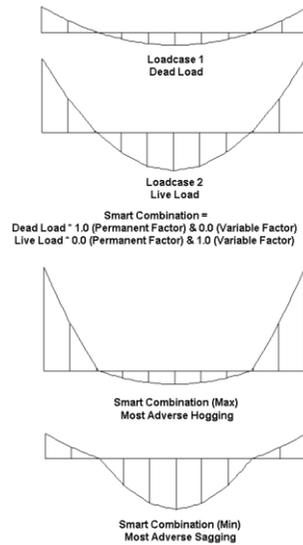
Basic combination   **Smart Combination**

Envelopes

Envelopes enable the maximum and minimum results from individual loadcases (possibly from more than one analysis) or from other combinations or envelopes to be determined.

Defining a Basic or Smart Combination, or Envelope

A basic combination, smart combination or envelope 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. Combinations and envelopes definitions are saved in the model file.
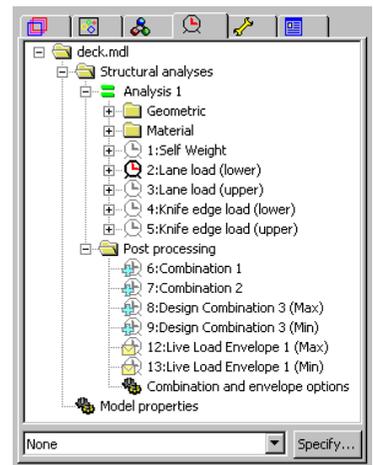
Superposition

Combinations are only generally applicable to linear elastic analyses. This is because the principle 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.

Manipulating Basic, Smart Combination, or Envelope Loadcases

Basic or smart combination or envelope editing commands are available from the context menu that is activated by right-clicking on a one of those loadcases in the Analyses Treeview. The following commands are available:

- ❑ **Set active** Sets a loadcase active for viewing attributes assignments and results for the current window.
- ❑ **Edit** View and edit the defined loadcase information.
- ❑ **Examine definition...** View, in a treeview form, the nested combinations and envelopes that make up the definition, together with the corresponding factors for each included loadcase.
- ❑ **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.
- ❑ **Create loadcase...** (for basic combinations only) 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.
- ❑ **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.

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 advanced settings may be made. See "Combination
and Envelope Options" on page 627 for more details.

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, or,

- Drag and drop a loadcase entry from the Analyses ⌖ Treeview into a view
  window to set that loadcase active.

When using smart combinations or envelopes the **Set Active** menu item will prompt
for an Entity and 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. Choosing the 'Max/min each component
individually' option performs the calculation independently for each displayed
component. Thus, different components are not coincident.

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 loadcase can be set active.

Contour, Diagram or Values plots can be displayed to show maximum, minimum or
extreme results for an envelope or smart combination. The maximum and minimum
sections of the diagrams for these plots are drawn in different (user-definable) colours
from the default orange colour used for standard diagram plots. By default red is used
for maximum values, and blue for minimum values. The use of colours to differentiate
between maximum and minimum values is also applicable for the Values layer. See
Smart Combination and Envelope Examples below.

Viewing derived components

When a derived component is used as a primary component, or when a derived
component is inspected when choosing the 'Max/min each component individually'
option for particular situations involving smart combinations and envelopes, LUSAS
Modeller will automatically suppress the listing of particular derived components in

the Component drop-down list that could produce misleading results as the unwary could get an answer that could be misunderstood or misinterpreted when reporting on results involving those components. This can be overridden. See **Raw and Derived Components** for details.

General notes relating to viewing results from Combinations and Envelopes

- 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 combining or enveloping results from multiple models the mesh must be identical across all models.

- 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.

- When multiple envelopes are included inside a single basic combination, and a derived component is set as the primary component, note that it is not possible to "find" the "worst" permutation of a loadcase from each envelope, when factored as specified. This is because each envelope is computed in isolation prior to combining the results, and the effect of one envelope can cancel out some or all of the effect from another envelope.

- For a smart combination, when a derived component is set as the primary component, or when viewing a derived component and choosing 'Max/min each component individually' option, note that it is not possible to "find" the "worst" permutation of factored loadcases because the calculation is not associative.

- When considering the activation and deactivation of elements in relation to enveloping results note that if an envelope is set-up containing loadcase 1, loadcase 2, and loadcase 3, where the elements being viewed are 'born' in loadcase 2, this is, by definition, the same as an envelope of loadcase 2 and

loadcase 3 because loadcase 1 is ignored if the elements providing results for the envelope have yet to be activated.

Basic Combination Examples

Loadcase 1 + Loadcase 2 = Basic combination

❑ **Loadcase 1**

❑ **Loadcase 2**
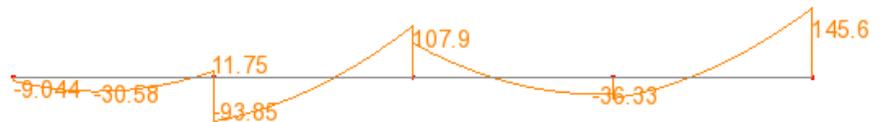
❑ **Basic combination**

Smart Combination Example

Smart Combination = Loadcase 1 (Dead load) * 1.0 (Permanent factor) and 0.0 (Variable factor) and Loadcase 2 (Live load) * 0.0 (Permanent factor) and 1.0 (Variable factor).

For loadcase 1 and loadcase 2 as shown in the basic combination example:

For a Smart Combination (Max) loadcase:

❑ **Show maximum only**

❑ **Show minimum too**

❑ **Show extreme**

For details of Smart combination calculations see **Appendix A**

Envelope examples

For loadcase 1 and loadcase 2 as shown in the basic combination example:

For an Envelope (Min) loadcase:

❑ **Show minimum only**
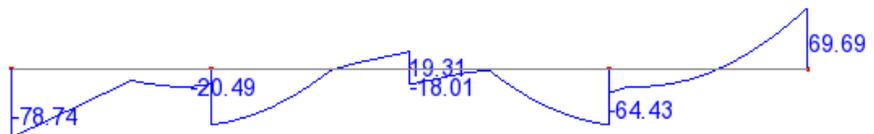
❑ **Show maximum too**

For an Envelope (Max) or (Min) loadcase:

❑ **Show extreme**
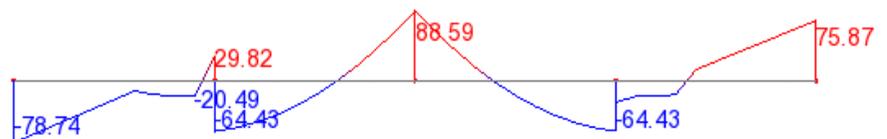
Combination and Envelope Options

The combinations and envelope options dialog can be accessed from the 🐝 Combinations and envelope options object in the Analyses Treeview.

Smart combinations

❑ **Display beneficial/adverse factors** (when checked) sets the smart combination labels on the smart combination dialog and associated grid dialog to be Beneficial/Adverse as opposed to Permanent/Variable.

Storing of results

Results for Modeller-calculated results components (such as those calculated for combinations and envelopes, and also for other derived components such as design code-based utilisations, Wood-Amer, user-defined results etc.) can be stored in order to speed-up the initial and subsequent display of those results. This can be done either by setting options to store calculated results in LUSAS Modeller interactively, or by specifying that all or selected results components should always be stored each time a model is solved. For either case, only those results not currently stored will be added to those that are.

Store calculated results for full model in interactive use

❑ **Never** This option disables all storing of results. All results are recomputed as and when needed, regardless of element visibility.

❑ **Only when viewing full model** Calculated results are stored provided the whole model is being viewed.

❑ **Always, even when viewing partial model** - Calculated results are stored for all elements, even if only a few are visible. Setting this option overrides and switches off any threshold settings (see below).

❑ **Whenever both the following thresholds are reached** - These two options provide a degree of refinement as to when calculated results are stored. Both threshold values must be met for storing of results to take place.

- **Proportion of contributing elements** Calculated results are stored for all elements when the number of elements being viewed is greater than the stated percentage of the total number.

- **Number of contributing loadcases** - Calculated results are stored for all elements when the number of loadcases that are returned by LUSAS Solver, and that are used to compute a combination or envelope, exceeds the stated number.

❑ **Show warning to visit this dialog when calculation times reaches** - Here a specified time can be stated. This option is checked 'on' by default, and is set to 30 seconds.

Calculate and store results for full model for specified PRW utilities

As well as, or independently from, storing calculated results for the full model in interactive use, results components for chosen entities, loadcases and locations (specified as part of creating a **Print Results Wizard** utility) can be calculated and be automatically stored straight after a model is solved, or at any time thereafter if this section of the Combinations and Envelope options dialog is visited. Any previously-defined Print Results Wizard utilities will be listed in the selection panel on this dialog. Additional Print Results Wizard utilities entries can be added by use of the 'New' button. Print Results Wizard utilities must be 'checked' in the selection panel in order for storing of those results to take place. Results can be specified to be stored:

❑ **After each solve** Calculates and automatically stores the specified results for a Print Results Wizard utility straight after a model is solved.

❑ **Now** Calculates and stores the specified results immediately, based upon the currently loaded results file.

Note that for both these options, storing of results takes place for the whole model ignoring any 'Extent' setting.

❑ Additional Print Results Wizard utilities can be defined by pressing the **New...** button at the bottom of the Combination and Envelope Options dialog. Pressing the 'New' button shows the normal Print Results Wizard dialog, but with inappropriate controls for caching purposes shown disabled.

❑ Selecting a Print Results Wizard entry in the selection panel, and pressing the **Edit...** button permits editing of the results requested.

Deleting all stored results

❑ **Delete all stored results** Deletes all stored results.

Design Combinations

The Design Load Combination wizard is available in Bridge and Civil software products only. It is accessed from the **Analyses > Design Combinations...** or **Design > Design Combinations...** menu items.

Design combinations are used to assign LUSAS loadcases, envelopes or basic combinations to predefined load types for a supported design code. Based on the assignments and settings made, the required set of load combinations for those load types are automatically generated by LUSAS Modeller.

See **Design Combinations** in *Application Manual (Bridge, Civil & Structural)* for more details.

Progress Bar

A progress bar is displayed at the bottom of the user interface to show the approximate progress being made by LUSAS Modeller in calculating results for a particular task such as creating a design envelope, or combination. Each progress step represents one results component being calculated for one constituent loadcase.

For small models with only a few loadcases the progress bar may not even be seen, but for larger models with a large number of loadcases, where numerous loadcases need to be combined and enveloped, or otherwise processed, the progress bar provides a reassurance that calculations are continuing.

For some tasks, such as creating a model report, several results chapters may have been requested, each containing several components requested for several envelopes, each made of several hundred constituent loadcases. In this situation the progress bar may move smoothly throughout the calculation of results. But for a single envelope calculation, such as when drawing contours or plotting values in the view window, or printing results using the print results wizard there may be a noticeable pause at the penultimate step when those results are combined together to make a single result. Note that depending upon the complexity of the task Windows may mark Modeller as 'not responding' during that time, but in fact the calculations will still be continuing. Allowance should be made for the results calculation to finish in such situations.

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 **Analyses> 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.
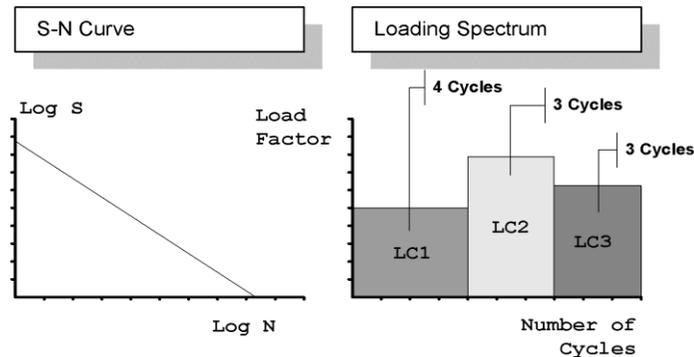
Notes

- 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.
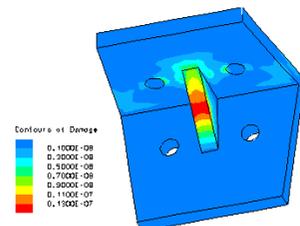
Viewing results

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 log10 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, moving load, and moving mass 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

The basic steps are as follows:

1. Decide whether to perform a response analysis on a single node (using the Graph Wizard), or the whole structure (by defining an 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.

LARGE MASS
(1e6 kg Joint Element)
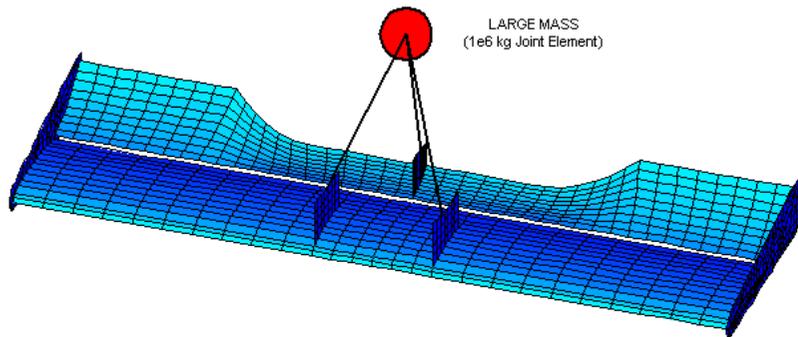
Modal Dynamics Results Types

❑ **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
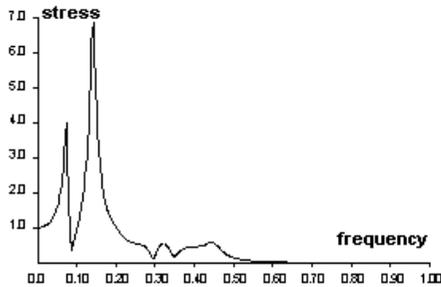
Response at a node calculated across a specified time history
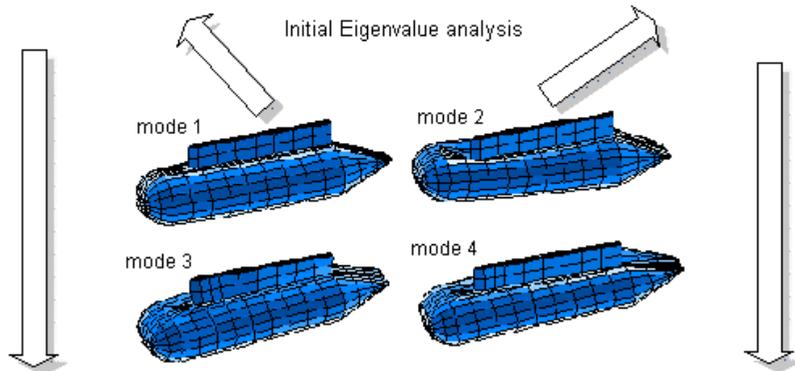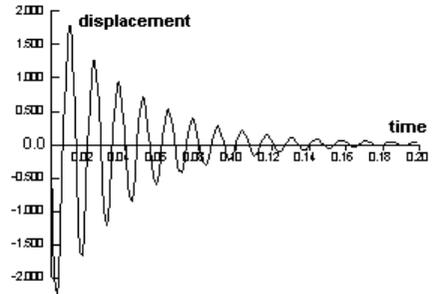
Results at a specific frequency, or time step

Response calculated at a specified frequency for the whole structure
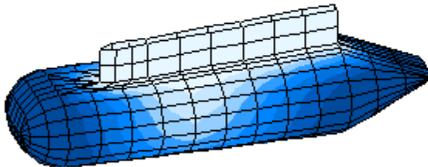
Response calculated at a specified time for the whole structure
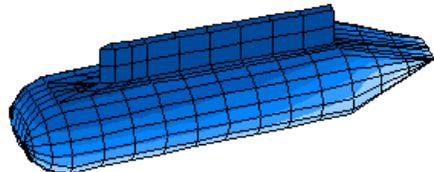
Frequency Power Spectral Density

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. Select the **File> Open** menu item and specify the results file name.

2. Select the **Utilities> Graph Wizard**menu item. 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.

IMDPlus Analysis

The IMDPlus software option solves 2D and 3D seismic, moving load and moving mass analysis problems using modal superposition techniques in the time domain. In order to use IMDPlus, as for dynamic analysis in general, a consistent set of units should be used, as for example Force = N, Mass = kg, Length = m, and Time = s if SI units were being used.

The various stages of an IMDPlus analysis are initiated from the **Analyses> IMDPlus** menu or from the **IMDPlus Toolbar**, which can be activated from the **View> Toolbars...** menu.

The basic steps that need to be undertaken for each of the three available analysis types are as follows:

Seismic Analysis

A seismic analysis obtains the dynamic response of a structure to acceleration time histories of support motion.

In order to perform an **IMDPlus seismic analysis**, a mass normalised eigenvalue analysis first needs to be undertaken to describe the frequency behaviour of the structure. Once completed, the response of the structure associated with up to seven earthquake acceleration time histories can be calculated and node / element results interrogated.

Procedure

Select the **Analyses > IMDPlus> Seismic** menu item and carry out the following steps:

1. Select the acceleration time histories

2. Select the eigenmodes to include in the solution

3. Specify the damping for the eigenmodes

4. If required, modify the advanced solution options

5. Specify the frequency interpolation technique and if required, the time step

6. Specify the nodes or elements to process along with the output requirements

Moving Load Analysis

A moving load analysis obtains the dynamic response of a structure to moving loads. In a moving load analysis the load can either be explicitly defined or a vehicle configuration (composite axle definition) can be used. In both cases the magnitude and configuration of the load remains constant throughout the analysis.

❑ **Explicit definition** The complete load configuration is defined through a discrete load definition in Modeller. This can be used for analysing the passage of a single load configuration across the structure

❑ **Composite axle definition** The complete load configuration consists of a unit load single axle configuration, defined as a discrete load definition in Modeller, and a vehicle configuration is that is defined separately. The vehicle configuration (composite axle definition) method is the recommended approach, as it allows multiple load configurations (such as multiple train rolling stock configurations) to be analysed rapidly without having to repeat initial steps for the IMDPlus solution.

In order to perform an **IMDPlus moving load analysis**, the path of the load first needs to be defined in addition to the frequency behaviour of the structure. Once the mass normalised eigenvalue analysis has been undertaken, the response of the structure to moving loads can be calculated and node / element results interrogated.

Procedure

Select the **Analyses> IMDPlus> Moving Load** menu item and and carry out the following steps:

1. Vehicle Configuration

If the loading is not explicitly defined in step 2, define moving load vehicle configuration(s) using the positions of the axles/bogies that form a vehicle together with the moving load factors to be applied at each position

2. Moving Load Generation

Generate the movement of a discrete load across the structure by using a number of distinct locations along the moving load path

3. Modal Force Calculator

Calculate the equivalent modal forces for each of these distinct locations

4. Moving Load Analysis

a. If required, select the moving load vehicle configuration to be used in the analysis

b. Select the eigenmodes to include in the solution

c. Specify the damping for the eigenmodes

d. If required, modify the advanced solution options

e. Specify the speed ranges to analyse along with time stepping parameters

f. Specify the nodes or elements to process along with the output requirements

Moving Mass Analysis

A moving mass analysis obtains the dynamic response of a structure to moving masses. In a moving mass analysis spring-mass systems are used in a vehicle configuration. Each spring-mass system includes an unsprung mass and up to two sprung masses connected by springs and viscous dashpots. The configuration of the systems remains constant throughout the analysis but as they move across the structure, the dynamic response of the unsprung and sprung masses affects the forces acting on the underlying structure due to inertia effects.

The working assumptions for the particular case of an IMDPlus moving mass analysis are as follows:

❑ **No Loss of Contact** There is no loss of contact between the unsprung masses (wheels) of the spring–mass systems and the structure at any time during a moving mass analysis.

❑ **Vertical Motion of Spring-Mass Systems Only** Only vertical motion of the spring-mass systems is considered in a moving mass analysis.

❑ **Mass of Spring-Mass Systems Not Included in Eigenvalue Solution** The mass of the spring-mass systems have no effect on the natural frequencies of the structure

❑ **Recommended Limit on Total Mass of Spring-Mass Systems** The accuracy of the moving mass solution reduces for light-weight structures or structures where the magnitude of the moving mass is similar to the magnitude of the structure mass. A literature search recommends that for most solutions of this type, the total mass of all the spring-mass systems that are actively interacting with the structure at any given time should not exceed 10% of the total mass of the underlying structure.

In order to perform an **IMDPlus moving mass analysis**, the path of a unit load first needs to be defined in addition to the frequency behaviour of the structure. Once the mass normalised eigenvalue analysis has been performed, the response of the structure to moving masses can be calculated and node / element / spring-mass system results interrogated.

Procedure

Select the **Analyses> IMDPlus> Moving Mass** menu and carry out the following steps::

1. Spring Mass Properties

Define spring-mass system property attribute(s) for use in the moving mass vehicle configuration

2. Vehicle Configuration

Define moving mass vehicle configuration(s) using the positions of the axles/bogies that form a vehicle together with the spring-mass system property attribute that is to be used at each position

3. Moving Load Generation

Generate the movement of a discrete unit load across the structure by using a number of distinct locations along the moving load path

4. Modal Force Calculator

Calculate the equivalent unit modal forces for each of these distinct locations

5. Moving Mass Analysis

a. Select the moving mass vehicle configuration to be used in the analysis

b. If required, include a path surface profile that defines the surface irregularities along the length of the moving load path

c. Select the eigenmodes to include in the solution

d. Specify the damping for the eigenmodes

e. If required, modify the advanced solution options

f. If required, modify the path surface profile options

g. Specify the speed ranges to analyse along with time stepping parameters

h. Specify the nodes, elements or spring-mass systems to process along with the output requirements

General IMDPlus Assumptions Applicable to All Analysis Types

The working assumptions for the IMDPlus modal dynamics facility are as follows:

- ❑ **Linear** The system is linear in terms of geometry, material properties and boundary conditions. Therefore geometrically nonlinear eigenvalue results are not applicable. Nor are nonlinear joint and slideline analyses suitable for this type of post-processing treatment
- ❑ **No Cross-Coupling** There is no cross-coupling of modes caused by the damping matrix. This is reasonable for all but the most highly damped structures or applications
- ❑ **Low Modes Dominant** The response is dominated by the lowest few modes.
- ❑ **Damping Ratios Are Below Critical** Damping ratios of 100% or more are not permitted due to the solution of the time domain response of the structure using either the Hilber–Hughes-Taylor (HHT) method or Duhamel's Integral.

See the *IMDPlus User Manual* for full details on IMDPlus analysis.

Worked examples

See Online help > Worked Examples > For IMD Plus Option

Target Values Analysis

Target values analysis 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.

On the Target Values dialog 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.

Cable tuning analysis

Whilst cable tuning analysis is possible using this general target values method it is greatly simplified by using the dedicated **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.

Defining a target values analysis

Setting up a target values analysis requires you to:

1. **Choose a solution type** An 'exact' method, an optimisation facility and two best-fit solution methods are available
2. **Define the loadcases that will be used in the target values analysis**

The Target Values dialog contains tabbed pages to allow these tasks to be completed. Note that the Target Values dialog is functionally similar to the Cable Tuning dialog, with the exception that a no Cables page is included to allow the selection and inclusion of lines representing cable stays. Instead, lines of interest are set target values

The target values dialog pages **Solution types**, **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 on the Target Values dialog all input is saved, even if it contains errors or 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.

Double-clicking a target values entry (or selecting Edit from its context menu) will show the settings used to create it.

Solving a target values loadset

To solve a target values loadset press the Solve button ▤ on the main toolbar menu.

When solved, target values loadcase results ⊙ are held in a Post processing folder in the Analyses ⌚ Treeview.

Viewing results of a target values analysis

- Right-click on the target values results entry ⟠ and select **Set Active** to view results.

- Double-click on the target values results entry ⟠ to display the forces calculated for each loadcase in order to achieve the target values.

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.

Target Values: Solution types

For an overview of the steps required to carry out a target values analysis see **Target Values Analysis**.

Solution types

The choice of solution type is made on the main page of the Target Values dialog.
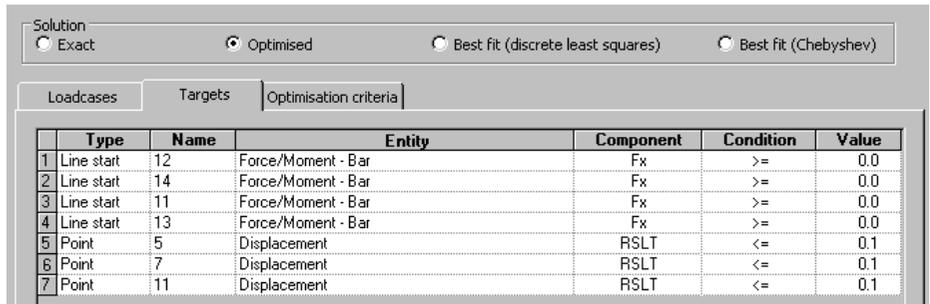
The type of solution that can be chosen is dependent upon the number of lines selected and the number of equality conditions as seen in the Condition column on the Targets grid. The solution type can be set to be:

❑ **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 loadcases that are defined as requiring a Calculated factor on the Loadcases tab. 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 number of loadcases that are defined as requiring a Calculated factor on the Loadcases tab. 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 number of loadcases that are defined as requiring a Calculated factor on the Loadcases tab. 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 **Solution type**, **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.

Target values: Defining the loadcases

This page of the target values dialog is similar in operation to the basic and smart combination dialogs.

Previously defined loadcases need to be selected for inclusion in the calculations to achieve specified target values. Each can also have a factor applied.

For an overview of the steps required to carry out a target values analysis see **Target Values Analysis**.
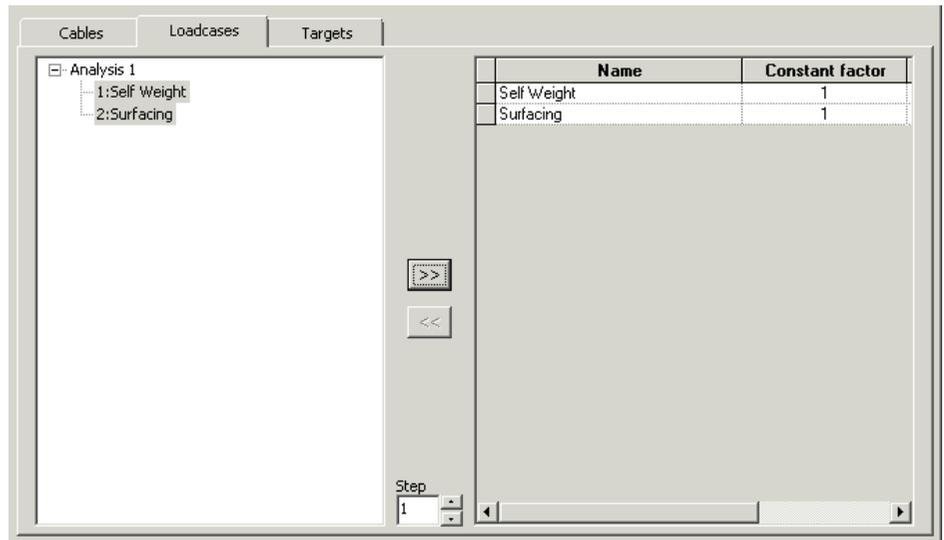
❑ Results can be combined from different results files if those files are open. Click in the drop-down list to select the results file. To include a results loadcase select in the left-hand panel and click the Add to ⟩⟩ button to add it to the **Included** panel. Use the **Shift** key to select multiple results cases.

❑ Combination and Envelope loadcases are saved with the model data; click in the drop-down list to see and select the model entry.

❑ Combinations may include results, other combinations or envelopes as necessary (but see note regarding superposition below)

❑ Included loadcases can be factored by selecting the loadcase and modifying the value in the **Factor** field

❑ The **Grid** button enables the combination data to be visualised in a table. This table can be modified directly, or by use the standard windows cut and paste options.

Loadcase settings

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.

Notes

- Loadcases marked as having a Constant factor are considered in the optimisation, but not modified by it. There may also be other loadcases in the model which form no part of the optimisation.
- Loadcases marked as having a Calculated factor may be factored, for both exact and non-exact solutions.
- There is no restriction on multiple combinations and envelopes that reference the same loadcase, or indeed that reference loadcases that are included in their own right. As a result there may be some permutations of loadcases for which a particular analysis solution cannot be found. Detecting a potential problem in advance of solving is not practical, so if this should occur any error is reported after a solution has been attempted.
- When a target values analysis is carried out LUSAS cannot verify that superposition holds when combinations contain loadcases from different analyses or when nonlinear behaviour is being considered. This may occur if lift-off supports are present, or when geometric or material changes take place between analyses.

Target values: Defining target values

This page of the Target Values dialog provides the means to add rows of feature type, entity and component data to a grid for comparison, row-by-row, against target values. The equality or inequality conditions available are governed by the solution type. An equality condition can be set for either a point, or the start or end of a line.

For an overview of the steps required to carry out a target values analysis see **Target Values Analysis**.

Use

Model features of interest (points or lines in a geometry model, or nodes in a mesh-only model) should be selected prior to selecting the **Add Selected** button. Note that the selection of features, and of general Modeller menu items, is also possible whilst the main Cable tuning dialog is displayed.

When the Add Selected button is pressed the **Add Targets From Selection** dialog assists in the population of the grid by allowing feature type, entity and component selections to be made for the chosen selection of features.

- When added singly the feature name, entity and component should be chosen from the combos on the Add Selection dialog for each.

- When added for a multiple selection of features common target data can be set on the Add Selection dialog for all the similar features in the selection.

When data is already in the grid the **Add** button will duplicate the last row. When the **Add** button is used without data in the grid the feature name must be specified manually but thereafter selections can be made in the combo boxes for each column.

Typical target values settings are shown:

Checking input

- ❑ The **Validate input** button is used to check if the number of equality conditions and the number of lines representing cables are valid for the chosen Solution method.
- ❑ The **Identify selection** checkbox 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.

Notes

- Nodes can only be selected in mesh-only models.

- If Solution type is set to **Exact** only the '=' condition is available.
- If Solution type is set to **Optimised** or **Best Fit** the entries in the Condition column can be set to '=', '>=' and '<='.
- Type, Entity and Component settings in the grid can be modified using the drop-down menu for each cell.
- Types **Line start** and **Line end** are similar to specifying a **Point**, but clarify which element is to be examined, for example where multiple lines meet at a point.
- The **user defined results** facility may be used to construct components that are more complex than the standard provided components. This allows targets to be given in terms of potentially complex expressions involving components from more than one results entity.
- Points and Lines cannot be deleted from the model if they are present in a target values loadset.

Target values: Optimisation criteria

This page of the Target values dialog provides the means to define optimisation criteria based upon solution type.

For an overview of the steps required to carry out a target values analysis see **Target Values Analysis**.

The following are shown only for the Optimised Solution type:

- ❑ **Minimise the largest calculated factor** The calculated factor for the cable stay with the largest factor is minimised.
- ❑ **Minimise sum of the calculated factors** The sum of the calculated factors for all cable stays is minimised.
- ❑ **Minimise the sum of the criteria specified below** Minimises the sum of additional values (at given locations) as specified in the grid to optimise the solution.
- ❑ **Maximise the sum of the calculated factors** The sum of the calculated factors for all cable stays is maximised
- ❑ **Maximise the sum of the criteria specified below** Maximises the sum of additional values (at given locations) as specified in the grid to optimise the solution.

Notes
Specified criteria may refer to any results component and can be completely independent of the targets, or may duplicate one of them. The **user defined results** facility may also be used to construct components that are more complex than the standard provided components so the optimisation criteria can be given in terms of potentially complex expressions involving components from more than one results entity.

Steel Frame Design

Steel frame design code checking facilities are available in Bridge and Civil software products only. See **Steel Frame Design - Code Checking** in the *Application Manual (Bridge, Civil & Structural)* for more details.

Wood-Armer Reinforcement Calculations

Wood-Armer attributes, when defined and assigned to features in a model, permit calculation of top and bottom reinforcement moments for plate, shell and grillage elements, and additionally top and bottom reinforcement forces for shell elements according to Wood-Armer / Clark-Nielsen theory.

Wood-Armer theory 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 Wood-Armer design moments [Mx(T), My(T), Mx(B), My(B)] may be obtained. 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-Nielsen 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 outer faces 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, 'equivalent plate width', has to be defined for the grillage to compute these equivalent moments. The 'equivalent plate width' is defined in the **grillage geometric attribute**.

Wood Armer reinforcement properties can be specified by selecting the **Attributes > Wood Armer** menu item. See **Wood-Armer / Clark-Nielsen Attributes** for more details.

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.
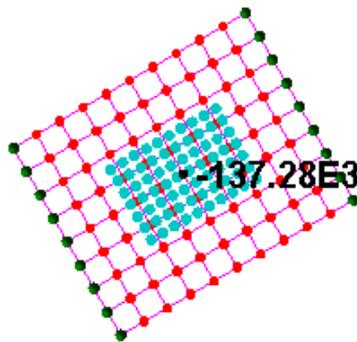
Wood-Armer moment (MXB) in global X axis for a grillage model
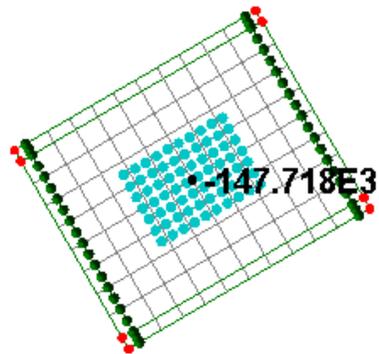
Wood-Armer moment (MXB in global X axis) for the equivalent plate model

<div align="center">
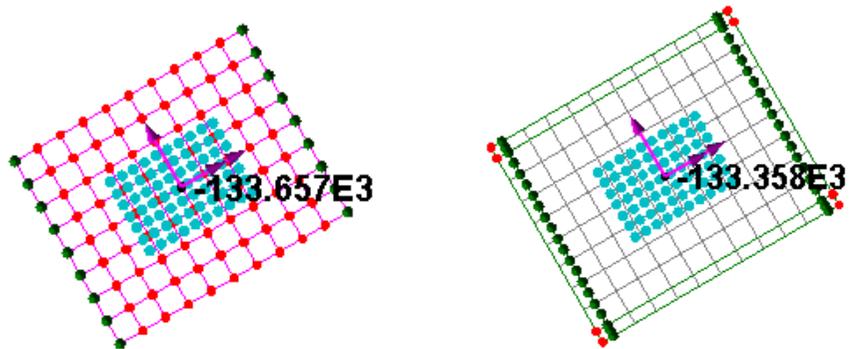
**Wood-Armer moment (MXB) in transformed angle
direction for a grillage model**   **Wood-Armer moment (MXB) in transformed
angle direction for the equivalent plate model**

Comparison of untransformed and transformed values for grillage and plate models

</div>

See the *Theory Manual* for further information on Wood-Armer calculations.

Reinforced concrete slab design facilities are available in Bridge and Civil software products only. They can be accessed using the **Design> RC Slab Design** menu item.

See **RC Slab Design : Overview** for more details

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** or the "Transient Smoothed Multi Crack Concrete Model (Model 105)" on page 282

❑ For slabs modelled by shell or plate elements, and for design codes that support crack width calculation. 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. 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 **Analyses> Basic Combination…** 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 Factor Plots

Design factor plots are available for use with the Analyst software product only. They provide a means of assessing the reserve strength capacity of a component or structure. To use this facility the results file must be loaded with the model file. The material strength and design factor are defined from the **Attributes> Design> 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:

One of the following failure theories incorporated into LUSAS Modeller must be chosen:

❑ 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)

Note: In the case of the Maximum Stress Theory, the component on which to test - either the maximum principal stress component S1 or the minimum principal stress component S3 must be defined.

Contours may be displayed of the following:

❑ Failure/Yield Index

❑ Factored Failure/Yield Index

❑ Factor of Safety

❑ Reserve Factor

❑ Margin of Safety

The calculation and failure limit value for the above options are:

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.

Note that only plane strain, plane stress and 3D continuum elements are supported.

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 failures. 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 must be given. It is this name that will be used in other dialogs (e.g. contour properties) to refer to the expression. The name must comply with strict naming rules (no spaces, no punctuation) and must be unique. A description can be added if required. The description is not used by Modeller.

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.

User Defined Results

	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. EN1992-1-1 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

Add   Delete   Copy to Clipboard   Insert from Clipboard   OK   Cancel   Help

- Clicking on the drop-list button ▼ in the Entity cell lists all entities available for the elements in the model.

- 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.

See **Expressions and Functions Supported** in Appendix D for more details.

Name scoping

Name scoping (an association of a name to an entity, such as a component or variable for reference purposes) can be used within the User Defined Results facility to help avoid ambiguity of an included component or variable. Name scoping is optional as long as there is no ambiguity.

A User Defined Results expression such as "DX * 2 + Iyy * 3 + LDNUMB" is valid. However, some named variables could be ambiguous - for example more than one attribute type defines the variable T. To resolve this ambiguity, name scoping can be used. A scope is a prefix to the variable name that identifies where Modeller should search for the variable that follows. Adding name scoping to the previous example woul d give "entity.DX * 2 + geometric.Iyy * 3 + loadcase.LDNUMB".

Currently only uniquely assigned attribute types (those attributes where one assignment replaces another) have been added as name scopes. One example is design attributes.

A double name-scope is needed for User Defined Results components and scripted results components. As well as the main scope "UDR" (or "scripted"), an additional scope is used which is the name of the named set of expressions. So, for example, a typical component name might appear as "UDR.set_1.C_1" when used as a variable

within the definition of another component. Note that where spaces or dots occur within the name of a set, these are replaced with underscore characters.

Variable that can be added to a UDR expression	Name scope
LUSAS results component	"entity"
UDR component	"UDR"
Material properties	"material"
Geometric properties	"geometric"
Loadcase constant	"loadcase"

For a list of User Defined Results scopes see **Input and Output of Real Numbers in LUSAS** in Appendix D.

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.
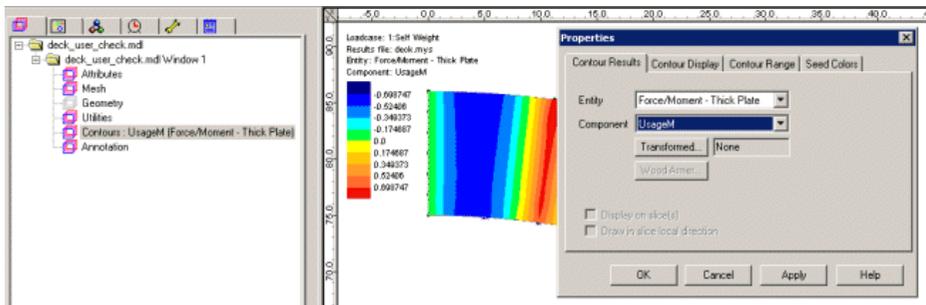
Visualising The Results

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 a direct method influence analysis loadcase is active the deformed mesh layer is renamed 'Influence shape'.

❑ **Influence shape** Draws the influence shape for an active influence analysis.

- ❑ **Contours** Display the results on the model as colour fringes or lines of equal value.
- ❑ **Vectors** Displays vector results quantities as an arrow (or sets of arrows) on the model.
- ❑ **Diagrams** Displays beam element shear force and bending moment diagrams.
- ❑ **Values** Marks result values on the model as symbols and/or values.
- ❑ **Annotation** This layer is used to display contour key and other annotation data that may be added to the results viewing window.

Isolated results can be viewed on contour and values layers for any defined location by using **Inspection Locations.**

For design checks, results for line beams can be obtained at nodal or internal points.

Results Orientation

> **When viewing results in LUSAS 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

Visualisation of results 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, or by dragging and dropping a loadcase name into the View window. 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 results 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.
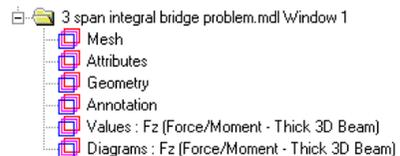
Deformed Mesh Plots

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.
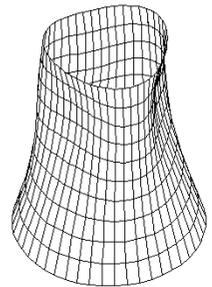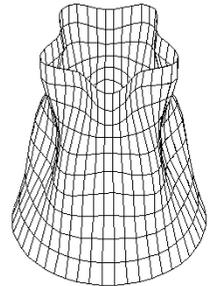
Initial deformations

❑ **Exaggerate initial deformations** Initial deformations (defined when specifying an initial state for a new analysis) are displayed using the mesh scaling value stated on this dialog.

Notes

- 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 clarifying the deformations.

- Results visualisations (contours, values etc) 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.

- When defining an initial state for certain analyses, 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. See "Specifying an initial state for an analysis" on page 536 for more details.

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 can be added 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 nodal** (unsmoothed) results. At discontinuities in the model such as those formed by geometry orientation, geometric assignments, and material assignments, for instance, a discontinuity in the contours drawn will be seen.

❑ Contours may be plotted at **Inspection locations.** For these, a single filled circle of a colour representing the value of the result is drawn at each location. For line beam models, when plotting these 'spot' contours for stresses for fibre location results, the circle is drawn at the assigned location on the nodal line and not at the true fibre position.

❑ For grillages 'spot' contours can be shown of Wood-Armer and Clark-Nielsen components, as well as for equivalent plate moments.

❑ For line beam design checks at nodes 'spot' contours are used. For clarity, contours for coincident nodes are shown drawn slightly offset from their true positions.

❑ Contours may be plotted to show **Feature maximum**, **Feature minimum**, or **Feature extreme**. For these options, the whole of each feature will be drawn in the same contour colour, in a colour that relates to the single highest value found in any node / element on that feature. See **Feature based results** below.

❑ 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 or minimum results value and 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.
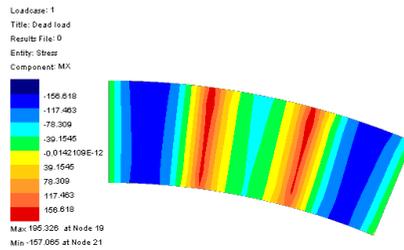
Colour filled contour plot   **Filled circle 'spot' contour plot**
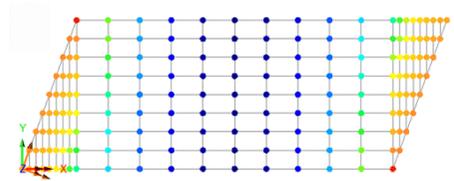
Setting the Contour Range

By default the values 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. Contour range settings can be controlled from the Contours properties dialog, accessed from the Contour layer entry in the ▣ Treeview.

- 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 colour when printing.

See "Concrete Models (Nonlinear)" on page 263 for material models that model both cracking and crushing behaviour of concrete and permit plotting of contours and values of crack widths.

Feature-based results

Feature-based results can be displayed on results contour plots, or be printed in results tables created by the Printed Results Wizard, or be selected to be included in Reports when adding a results chapter.

❑ **Feature maximum** uses, prints or displays on a contour plot the 'maximum' value for a feature where the maximum may be a positive or negative number.

For instance, for values of -8 and +3, +3 would be the feature maximum. For values of -8 and -3, -3 would be the feature maximum

❑ **Feature minimum** uses, prints or displays on a contour plot the 'minimum' value for a feature where the minimum may be a positive or negative number. For instance, for values of -8 and +3, -8 would be the feature minimum. For values of -8 and -3, -8 would be the minimum.

❑ **Feature extreme** uses, prints or displays on a contour plot the value with the largest magnitude for a feature, but (unlike an absolute value) preserves its sign, so for example a value of -8 is considered more extreme than +6.

Feature-based results examples
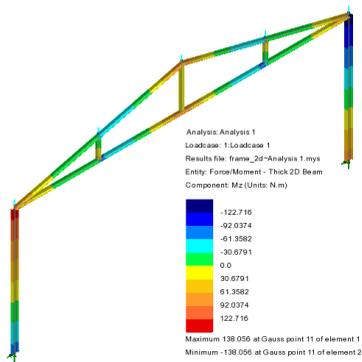
Unaveraged nodal results
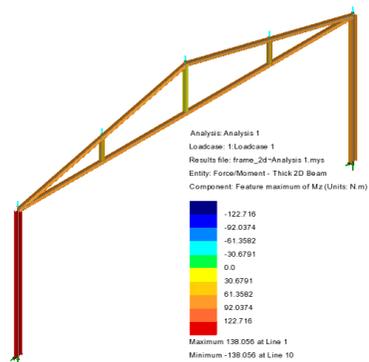
Contours of Feature maximum
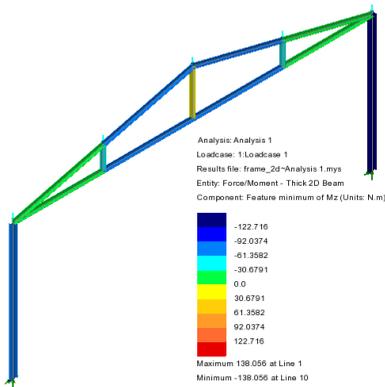
Contours of Feature minimum
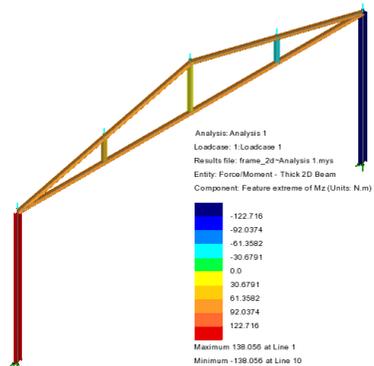
Contours of Feature extreme

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. At discontinuities in the model such as those formed by geometry orientation, geometric assignments, and material assignments, for instance, vectors are not drawn at the discontinuity.

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.
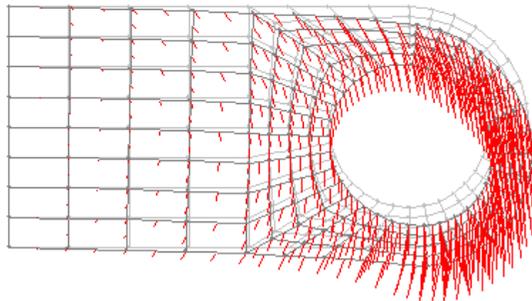
Values

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 on the selection and at **inspection locations**.

The Values layer is also used to display symbols showing locations of yield, concrete cracking planes, and symbols denoting regions of localised crushing. See **Nonlinear Material Plots** for details.

At discontinuities in the model such as those formed by geometry orientation, geometric assignments, and material assignments, for instance, multiple values will appear at nodes on the discontinuity in a multi-line list.
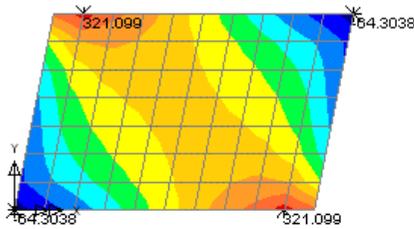
Maximum and minimum values marked
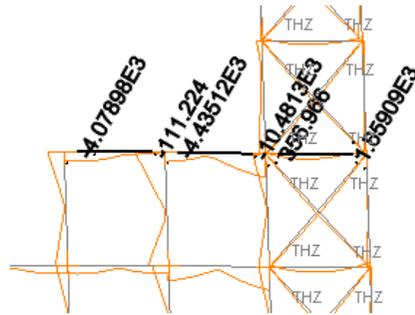
Values plotted for selected (shrunk) elements

Visualisation of yielded material
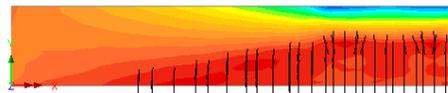
Visualisation of concrete crack patterns on a 2D model

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 results on the screen for a subset of a model. Setting this value to 0% shows only the extreme maximum and/or minimum value.

- To prevent text labels from overlapping each other the elements on which values are drawn may be shrunk by specifying a percentage of element's length remaining on the Mesh layer properties dialog. The same facility may be useful (e.g. at an averaging discontinuity) to identify to which element each value belongs. 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 selected 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. A significant difference between

values at the same node, coming from adjacent elements, indicates either an averaging discontinuity or a mesh refinement issue.

Diagrams

Bending moment, stress and force diagrams may be drawn for any 2 or 3 dimensional frame structure comprised of **bar** or **beam** elements. Diagrams can also be drawn to show beam/shell slicing results, and any surface quantity at any given graph location. The results components available depend upon the element type (see *Element ReferenceManual*). The diagrams may be drawn using the element axes or screen axes.

Diagram values can be displayed for the whole model, or for selected elements. Diagrams can be orientated by element axes or made to lie flat on the screen, regardless of model or element orientation. Peak values (as opposed to all values) may be optionally plotted. All diagrams can be plotted on the undeformed or deformed shape, and drawn either to match the element length, or to a percentage of that length to help clarify the display.
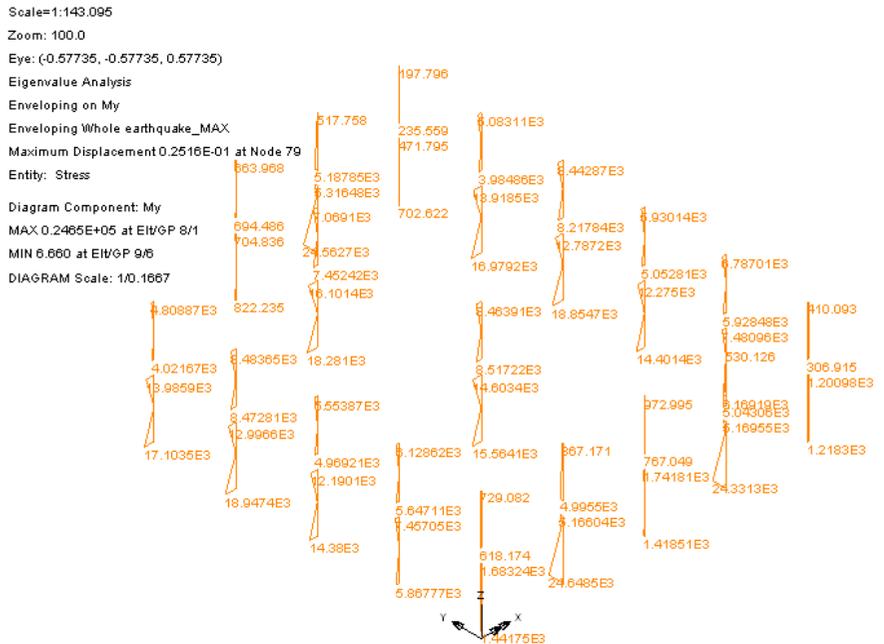
Viewing results at inspection locations

An inspection location is a defined position of interest on a model for results extraction / viewing purposes. Inspection locations can be defined and assigned to a model either before or after a model has been solved, and are saved when a model is saved. Model features can have multiple inspection locations assigned to them.

Appropriate results can be viewed for all defined inspection locations by selecting the Inspection Locations option from a relevant results dialog. Inspection location results can be displayed/inspected for:

❑ **Graphs**
❑ **Values layer**
❑ **Contours layer**
❑ **Print results wizard**
❑ **Reports**

See **Inspection locations** for more details.

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.

Symbols adjacent to each group name in the 🔷 Treeview show the status of model visibility and results display.

When viewing results

🔷 This is the default group icon.

When 'Show visibility feedback in treeview' is selected and a results file is loaded the following icons are provided:

✓ (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.
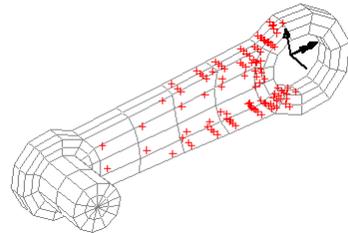
This display is especially useful when used in conjunction with contours or an animated sequence when the onset and spread of yield can be highlighted.

❑ **Crack patterns and Crushing** Crack patterns and crushing symbols can be displayed for smoothed multi-crack concrete material models. 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.
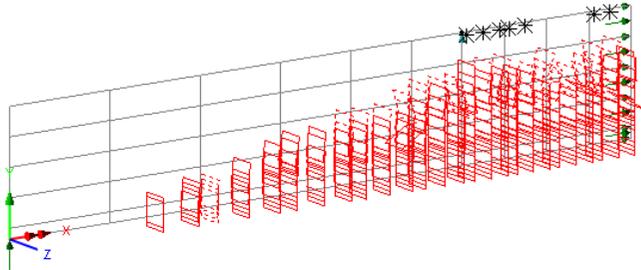
Open and closed crack planes and crush symbols resulting from the loading and unloading of a 2D concrete shell model

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 "Smoothed Multi Crack Concrete Model (Model 109)" on page 290 for details of plotting contours and values of crack widths following a structural analysis.

- See "Transient Smoothed Multi Crack Concrete Model (Model 105)" on page 282 for details of plotting contours and values of crack widths following a hygro-thermal coupled analysis.

Results on Sections / Slices through a Model

Results can be plotted on sections or slices through a model that has results available 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 section cut through a 2D continuum model, or on a shell, or a plate, or on a slice section which, in turn, was cut through a solid model. Results are calculated at the intersections of the line section and the element edges hence a finer mesh will produce more sampling points. Any results values along the slice may also be computed.

Section through a 3D Model

Slice sections may be cut at arbitrary positions through a solid 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.
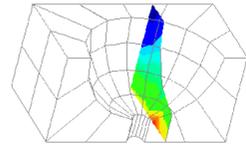
The location of an arbitrary slice section through a model is not saved with the model. However, when cutting a slice section through the model an option to create an annotation polygon is provided. This annotation polygon effectively defines the location and orientation of the cutting plane and does 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, followed by the **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 solid models. For surface models refer to **Graph Through 2D** below.

- Slice local x and y axes lie in the plane of the slice. The slice local z axis is perpendicular to both the slice local x and y axes. Reversing the local axes of the slice will re-orientate the z axis as well as the x and y local axes. For information on sign conventions for all element types see *Element Reference Manual*.

- 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 solid model.

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 by the mesh layer properties.

- The slice local axes and origin are displayed by selecting the **Draw axes** menu item from the slice context menu.

- The properties of the slice may be printed by selecting the **Print Properties** context menu item for the slice.

- The resultant local forces on the slice may be printed by selecting the **Print Local Forces** context menu item for the slice.

- Groups created from slice sectioning a 3D model are not currently retained when a model is saved.

- The orientation of the slice local axes is based upon [describe]. Slice local x and y axes lie in the plane of the slice. The slice local z axis is perpendicular to both the slice local x and y axes. Reversing the local axes of the slice by selecting the **Print Properties** context menu item for the slice will re-orientate the z axis as well as the x and y local axes.
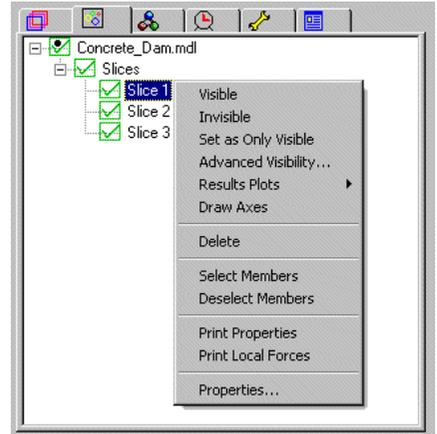
Graph through 2D

Arbitrary line sections may be taken through any surface model, or on a slice cut through a solid model. Line sections may be cut at arbitrary positions through the model using the cursor, with or without reference to an underlying grid. A slice section can also be created with reference to a selected line, arc or combined line, optionally when a line is given, the program needs to "map" the line onto the surface to be sliced. This is a trivial operation for a planar surface with a co-planar line. However, when the surface is complex, and/or the line is elevated away from the plane of the surface, a projection direction is used to map the line onto the surface. Slice section lines can also be created from intersecting surfaces (advanced use) or at the location of an existing graphed location.

When defining the line section by cursor, or for lines generated by projection, the section line created will, by default, be drawn in the Utilities layer. This may be used later for repeating the cut if a graph along the same line is required. The visibility of these lines is controlled by the Utilities layer properties dialog, as well as by each graph entry in the Utilities treeview.. It should be noted that the order of the layers in the Layers treeview may hide line sections drawn in the Utilities layer, and require that layer to be drawn on top of a contours layer in order for the line sections to be seen.

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. Values for graphing are obtained for each location where the slice section line intersects with the mesh. When a slice section line passes through a hole no values are calculated or plotted for the extent of the hole.

A **width for corridor averaging** value can optionally be specified to increase the number of results used when creating a graph of slice results in order to avoid localised results concentrations. When defined, results are obtained along three lines - the defined slice line and two parallel lines each offset perpendicularly from it by half the corridor width value. For each location distance along the slice three values are obtained. These are then weighted according to whether a value lies on the corridor centreline (x0.5) or at its edge (x0.25) before being averaged to form a single value for each slice line location for graphing purposes. When a slice section line corridor width has been specified and the edge of either side of the corridor overlays a hole, no values are averaged or plotted for the extent that the corridor overlays the hole. If a continuous unbroken graph is required in this situation either a narrower corridor width or no corridor width should be specified.

The process of cutting a slice and making appropriate loadcase and extent selections ultimately creates a graph utility in the Utilities 🔧 Treeview. This utility contains two graph datasets; the first containing the distance along the line section, and the second containing the specified results along the line. The graph can be drawn at any time by right-clicking on the graph utility and selecting the 'Show graph' option. A context menu option 'Visualise' displays the slice section location for a graph on the model.

Once a graph has been defined, its location can be used to draw contours, values or diagrams of any results quantity on the model by visiting the appropriate layer properties dialog, and choosing the option 'Graph locations' from the relevant droplist.

Notes

- The Graph Through 2D option is not available when a model is being viewed using the perspective facility.

Loadcases and Extent

Loadcases

The loadcases that the slice/graph is to report on can be specified by selecting either:

- ❑ **<Current active loadcase name>** - reports for this loadcase name only. See note.
- ❑ **Active** - reports for the Active loadcase only. See note.
- ❑ **All** - reports for all loadcases.
- ❑ **Specified** - reports for a specified named loadcase.
- ❑ **Create new window for each loadcase** - when checked, displays each graph in a separate window. Unchecked, all graphs are displayed within one window.

Note

Graphing of data for **<Current active loadcase name>** will graph only that loadcase. By setting the **Active** loadcase option will allow multiple graphs to be created or included in separate reports, simply by changing the Active loadcase in the Analyses Treeview, before each graph or report is created.

Extent

The extent of data that the slice/graph is to report on can be specified by selecting either:

- ❑ **Full model** - default option which reports for all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.
- ❑ **Visible model** - reports for all elements that are visible in the Modeller view window at the time the graph is created
- ❑ **Elements showing results** - reports for those elements that are displaying results in the Modeller view window at the time the graph is created.
- ❑ **Selection** - reports for only those elements that are selected at the time that the graph is created.
- ❑ **<Group name(s)>** - (if present) lists available group names.
- ❑ **New...** - allows creating a group name of visible items, or an empty group, as part of the graphing process

Notes

- Sometimes, due to the zoom setting, grid points are too close together to be usable for slicing purposes. In these cases simply increase the grid size so that individual grid points can be selected.

- Graphing of slice section data cannot take place on multiple stacked surfaces.

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.

Beam stress results

- When plotting results of beam stress for results components that contain a moment term such as Sx (Fx,My,Mz) or Sx(My) the results contours displayed on a fleshed model are accurately represented by the colours shown in the contour key. However, because the contours are being drawn for all locations on the section (an infinite number of y/z locations) at the same time, the maximum and minimum values cannot be stated on the contour key because they are based upon estimated values.

- Contours plotted on an unfleshed model for beam stress results are precise and are drawn for a specified active fibre location.

- Values, diagrams and Print Results Wizard tabulated values for beam stress results are precise and are plotted for a specified fibre location

Examples follow:

Loadcase: 1:Loadcase 1

Results file: tubular_roof_frame.mys

Entity: Stress - Thick 3D Beam

Component: Sx(Fx, My, Mz)

	-7.12163E6
	-5.34122E6
	-3.56081E6
	-1.78041E6
	-0.465661E-9
	1.78041E6
	3.56081E6
	5.34122E6
	7.12163E6
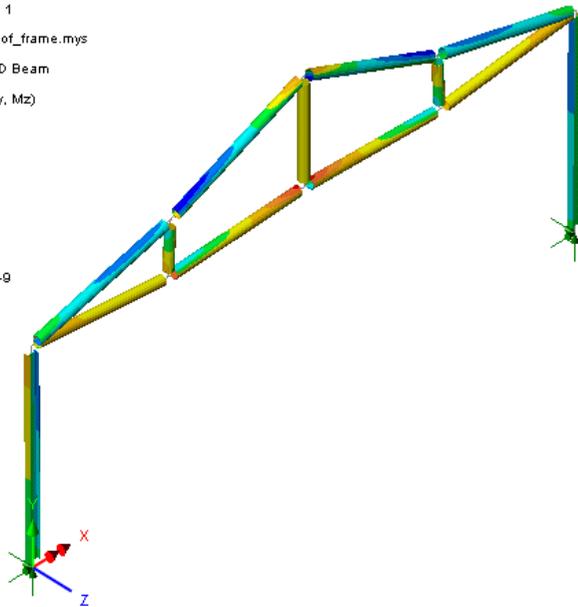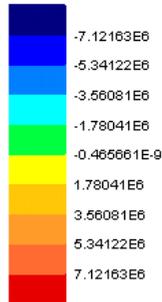
Filled contours plotted on fleshed beam sections

Loadcase: 1:Loadcase 1

Results file: tubular_roof_frame.mys

Entity: Stress - Thick 3D Beam

Component: Sx(Fx, My, Mz)

	-7.12163E6
	-5.34122E6
	-3.56081E6
	-1.78041E6
	-0.465661E-9
	1.78041E6
	3.56081E6
	5.34122E6
	7.12163E6
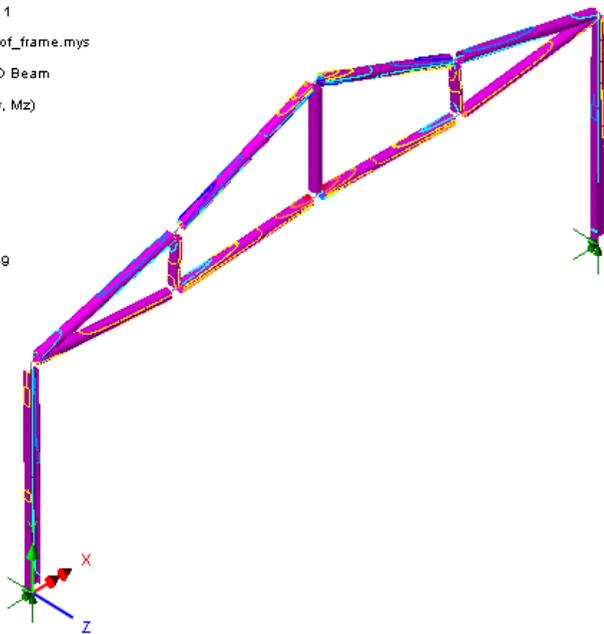
Line contours plotted on fleshed beam sections
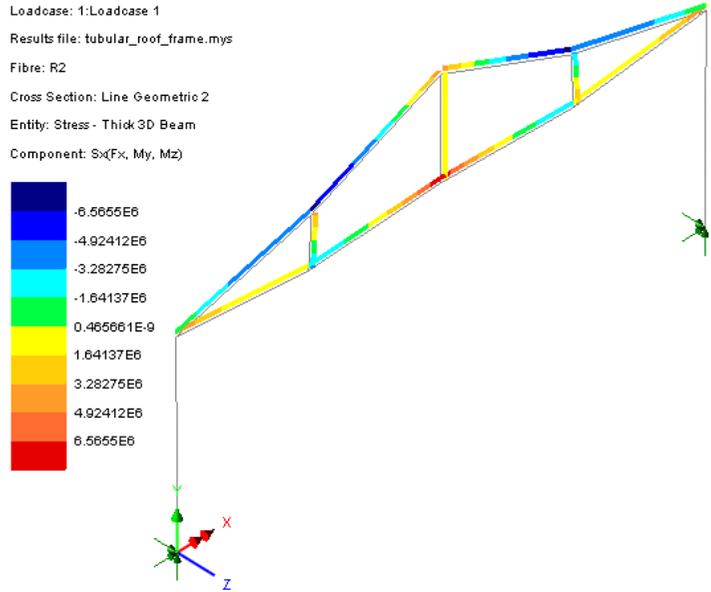
Line contours plotted at top beam fibre location
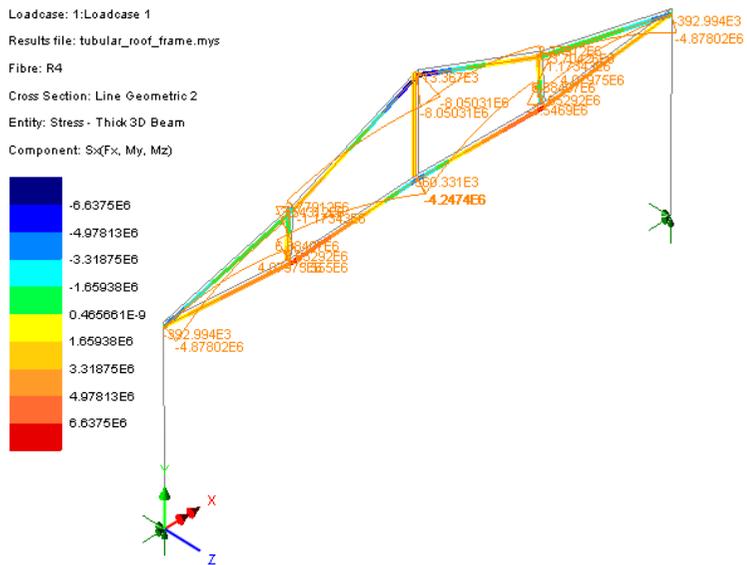
Line contours and diagram stress results plotted at bottom beam fibre location

Fibre locations

When viewing selected contour or diagram results for beams,or tabulating values using the **Print Results Wizard**, a fibre location must be used to specify the place(s) on the beam section at which the results should be calculated. 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.
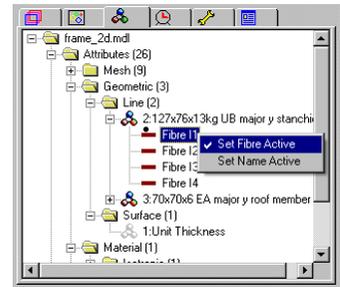
Notes

- 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 comprising linear isotropic materials by use of the **Utilities> Slice Resultants Beams/Shells** menu item. This allows the conversion of the results of a complex shell model into an equivalent beam analogy for use with design codes of practice. This facility can only be used with models with an **analysis category** of 3D.

Slice path

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

Slice locations

The locations of slices along the path can be defined by entering values using any of the following methods:

- ❑ **Incremental distances from start of path** (for example, entering '1@10;2@5' will set slice locations at distances of 10, 15, and 20 from the start of the path). Distances can be both positive and negative but the running total distance should remain within the length of the path.
- ❑ **Absolute distances from start of path** (for example, entering '10;15;20' will set slice locations at distances of 10, 15, and 20 from the start of the path). Distances must be positive and within the length of the path.
- ❑ **Parametric distances from start of path** (for example, entering '0.1;0.2' will set slice locations at distances of 10% and 20% of the distance along the path)
- ❑ **Constant spacing** (for example, entering 1.25 will set repeated slice locations at that distance along the path)
- ❑ **Include additional slices along path** if selected will create additional slices for each point on the line defining a path. For instance, for a single line representing a slice path, two additional slices will be created. For a combined line comprising three lines, four additional slices will be created.
- ❑ **Distance from reference origin to start of path (chainage)** 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 follows the selected lines and is in accordance with beam theory where the beam local x axis is orientated along the beam. The slice local x axis will be defined perpendicular to both the slice local y and z axes.

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:

❑ **Slice width** If selected, an overall width (centred on, and perpendicular to the slicing path) of elements to include in the calculations can be specified. If a slice width is not stated, all valid elements (in the specified extent) will be included in this direction. For both options, the slice is infinitely deep in the slice local z axis.

❑ **Include whole elements only** If the slice 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 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

❑ **Extent** The extent of the model that will be considered to contribute to the slice can be specified by selecting either:

- **Full model** - default option which includes all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.

- **<Group name(s)>** - (if present) lists available group names. Note that restricting the extent for each slice utility by specifying a group improves the calculation speed.

- **New...** - allows creating a group name of visible items, or an empty group, as part of the results/graph generation process.

❑ **Rotation about x** - Rotates the slice place about the x-axis of line beam. See notes if skewing of slice plane along beam is required.

❑ **Name** A name given to identify this slice data when an entry is added to the Utilities 🦴 Treeview.

Visualisation of slice locations

After making appropriate selections a Beam/Shell Slicing entry will be added to the Utilities 🦴 Treeview. A context menu for this entry provides the means to Rename, Delete or Edit the slicing data, and to turn off or turn on the Visualisation of the slice planes and their related information. Note that the location of the generated slice planes can only be visualised on the model after the results have been requested through any of the drawing layers. The size of the drawn slice plane depends upon the elements that are sliced. By default, slice planes and local slice axes are drawn in red, but these and other visualisation settings relating to slice axes, slice planes and slice

labels can be specified from the Utilities layer properties dialog, which is accessed from the Utilities ⬜ Treeview.
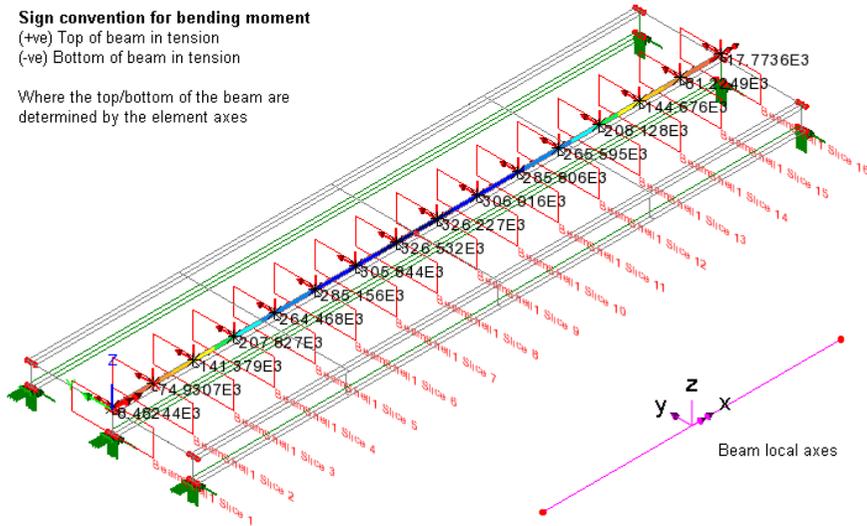
Isometric view of beam and shell model showing locations of slice planes, slice local axes, and slice labels. Inset shows beam local axis.

Viewing and printing results for defined slice locations

- When slice locations are defined and results are available, results for the slice locations can be viewed by selecting the **Beam/Shell Slice Resultants** entity from the layer properties dialogs for results (Contours, Diagrams etc) that support this.

- Use the **Print Results Wizard** to select Beam/Shell Slice Resultants of interest for viewing in tabular form or adding to a model report

Notes

- For use with linear isotropic models only.

- An assumption that plane sections remain plane is required for the calculations to take place.

- Linear variation of stress is assumed for this calculation method 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. If nonlinear material is present care should be taken to ensure the results obtained are suitable.

- The slicing path must be defined using straight lines, arcs or combined lines containing only these two line types.

- 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.

- The slice z-axis is taken from the line that represents the slice path. It cannot be changed.

- 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 x sides of the slice.

- Engineering thick beam elements are supported (BMI21, BMI22, BMI31 and BMI33) along with thin beam (BS3, BS4, BSL3 and BSL4) elements

- 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 are used wherever possible.

- Beam / shell slice results are not available for eigenvalue results as these are potentially misleading.

- If skewing of a slice plane along a beam is required, create a line at an appropriate skew angle from a point at one end of the beam and of sufficient length to extend beyond the extent of the model. This line can then be selected as a path line from which orthogonal slice sections will be generated according to the extent stated.

Worked Example

- See Online help > Worked Examples > For LUSAS Bridge and LUSAS Civil & Structural > 'Section Slicing of 3D Shell Structure' , which shows 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 deselecting 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.
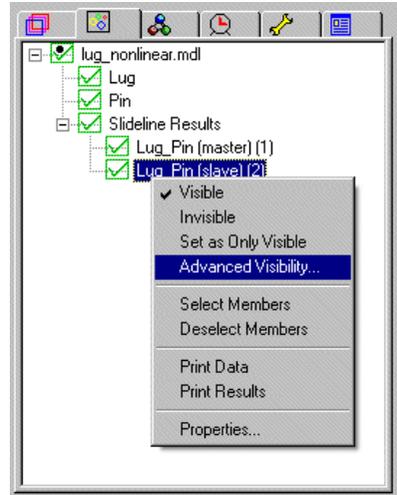
Printing Slideline Results

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
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 Utility provides the means to draw XY graphs of model or results data, edit an existing graph wizard utility-created graph, or simply create graph datasets for later use. It is accessed from the **Utilities > Graph Wizard** menu item. Graph datasets exported or copied to spreadsheets, graphs can be printed or included in reports.

Graphs of results can also be created along a line through a 2D continuum model or on a line through a section from a 3D model. See **Results on Sections Through a Model**.

The following results graphs may be plotted using the Utilities > Graph Wizard menu item 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** Use a previously defined graph dataset.

❑ **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.

❑ **Reuse values from existing graphs** Graphs two previously defined datasets.

❑ **Thermal surfaces** Graphs results along a thermal surface.

❑ **Slideline (assigned to line)**Graphs results along a slideline

❑ **Prestress** Graphs prestress losses in tendons

Graph Properties

The **Graph Wizard Utility** takes you through each step of creating the X and Y graph datasets and placing them into a graph.

When the graph wizard has been successfully completed (and 'Save in treeview' has been selected), the defined graph wizard utility name and its associated graph datasets are added to the Utilities Treeview, and if 'Display now' was chosen the chosen graph data is displayed. A context menu for the named graph wizard utility provides the following menu items:

- ❑ **Find** highlights the line section that was used to define the graph on the model. See also Visualise below.
- ❑ **Edit Attribute** provides access to the settings made in defining the graph.
- ❑ **Copy**, **Paste**, **Rename** and **Delete** options are self-explanatory.
- ❑ **Auto-Update** provides a switch to turn on / turn off automatic updating of the graph if the model has changed since its graph datasets were last displayed. Graphs set to not auto-update are shown by a locked graph icon 🔒 in the Utilities 🔧 Treeview.
- ❑ **Print** prints the values for all graph datasets associated with the graph entry in a separate view window.
- ❑ **Export** writes the graph data to a comma separated file (.csv)
- ❑ **Visualise** provides a switch to turn on or turn off the visualisation of the line section used to define the x-axis of the graph. Making the Utilities layer teh top-most layer in the Layers ⬜ Treeview will enable the visualisation to be seen.
- ❑ **Show Graph** allows predefined graphs to re-displayed at any time - without the wizard having to be re-accessed. Double-clicking on the named Graph wizard utility does the same.
- ❑ **Add to Report** adds the graph to a chosen location within a report.

On the Graph view window, toolbar buttons provide the means to print, print preview, add a graph to a report 📊, or edit the graph data ⌨ or its general properties and style 🖹.

A context menu for each named graph dataset 🔧 entry in the Utilities 🔧 Treeview provides the following optios:

- ❑ **Rename** Self-explanatory
- ❑ **Print** prints the values of just that dataset to the view window.
- ❑ **Export** exports the graph dataset values to a comma separated file (.csv). For graph datasets created for models prior to LUSAS version 15.1 it is also possible to export all graph datasets to a .csv file by using the **Export all** 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.

Graph datasets

Graph datasets contain the data that is graphed and cannot be deleted in isolation from the graph for which they are defined. Deleting a graph will delete associated graph datasets. Pressing the Undo button immediately after deleting a Graph Wizard-created graph entry will re-create the graph entry, but not the graph datasets. Reaccessing the

graph entry and creating a graph from its saved values will re-create the graph datasets for that graph. Graphs that are set to not auto-update are shown with a locked graph icon .

Graph visualisation
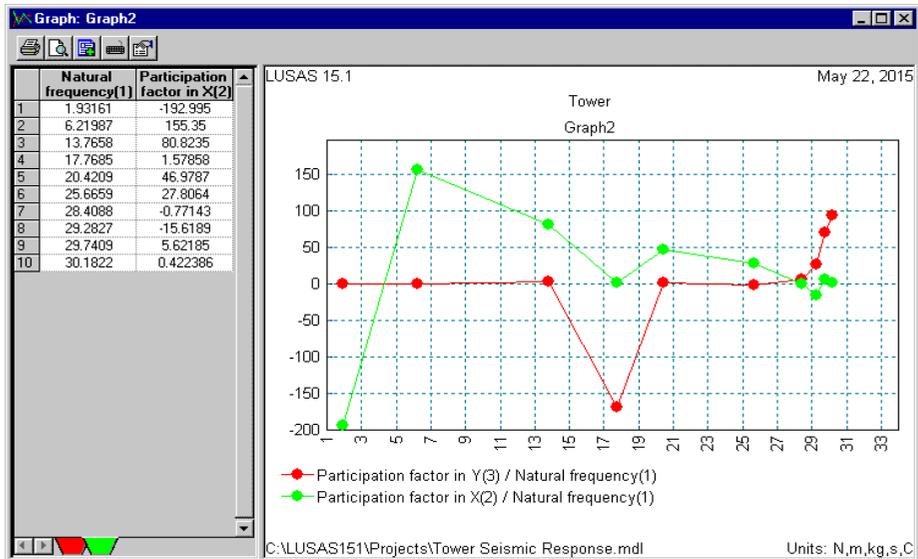
The graph window is split into the **graph data table** on the left and the **graph area** on the right.

Plotted graph data can be enlarged by zooming 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.**

Toolbar buttons on the Graph window provide printing, adding to report, and editing options.

Plotting Families of Curve Data On The Same Graph

If a graph already exists, then further curves may be added to the graph being created by choosing the **Include existing graphs** 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.

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. Graph dataset values can also be exported to a spreadsheet. See *Context menus* above.

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.

Adding Graphs to a report

Graph entries present in the Utilities 🔧 Treeview may be added to a report using the **Add to report** button. Each time a report is generated the graph data will be updated to match the current state of the model unless the graph is set to not auto-update.

Editing Graph data and properties

Once a graph has been plotted, the data that is graphed and the appearance of the graph may be modified.

A graph definition can be changed by using the **Edit Graph Definition** button or selecting the **Edit > Edit Graph Definition** main menu item. This allows total re-definition of a graph using the Graph Wizard.

Editing Graph Properties

To change the style or appearance of a graph, or to edit individual data points on the graph (for example to add an origin to a curve), select the **Edit Graph Properties** button, or select **Edit > Edit Graph Properties** from the main menu item or from the graph's own context menu. Grid details, X and Y Axis styles, and curve styles can be modified from respective tabs. By checking (ticking) the **Editable graph table** box on the General Graph Properties page. The following facilities are also 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.

<div style="background-color:#8fc4e8;padding:1em;">

Case Study. Plotting Families Of Curves

To compare results at different nodes, a graph could be plotted of the stress throughout an analysis at three nodes, say 6, 25, and 60. With a suitable results file, the nodes of interest would first be selected with the cursor,before proceeding as follows:

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 **Include 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**.

</div>

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 **Tools> 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.
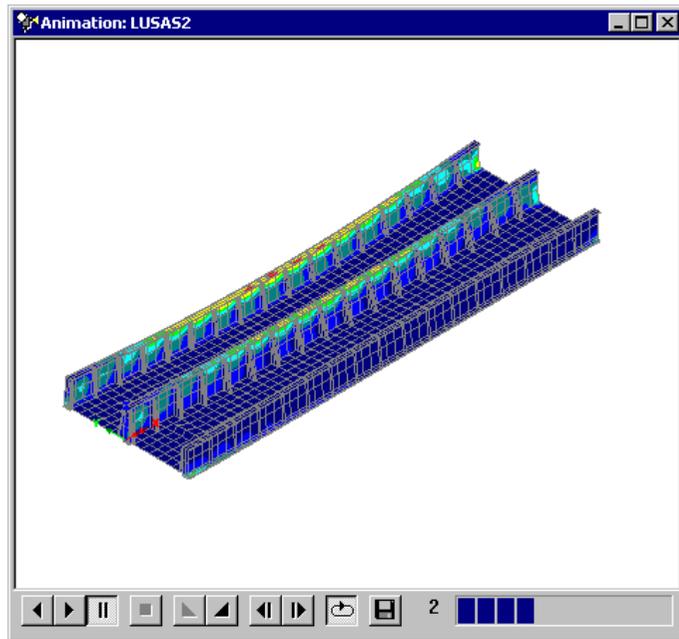
Notes

- 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.

Playing Animation Files (AVI format)

Windows media player is required to play AVI files.

Print Results Wizard

The Print Results Wizard allows selected results values, from one or more loadcases, to be assembled, ordered and displayed in a tabular listing format. By default, a Print Results Wizard entry is added to the Utilities 🗡 Treeview, which, if clicked, display the selected results. An option to additionally or only display the results to the screen is available. To assemble results use the **Utilities > Print Results Wizard** menu item or select the Print Results Wizard 🖽 toolbar button.

The Print Results Wizard dialog can also be accessed from the Add Chapter or Edit context menu items for a report that is present in Reports 🖼 Treeview. Selecting the results tab on the Chapters dialog, followed by pressing the New button, displays the

Print Results Wizard dialog to enable the results that will be saved in that results chapter of the report to be specified.

When selecting a loadcase and a results entity, results components of interest can be chosen. When the loadcase selected is a smart combination or an envelope, the reported results can be either coincident to a chosen primary entity and component, or individually computed (in which case they are not coincident). The location and extent of results can be specified, and the output type and order that the results are arranged can be chosen. Results precision settings can also be defined.

The results for each primary component for each loadcase are displayed within separate tabs in the print results window that is ultimately displayed, with tab names based upon the loadcase name and any selections made. Once listed the results can be printed, saved to a spreadsheet or text document, or be added to a **model report** by using the Add to Report button. Contributing envelopes, combinations or loadcases, and the values from each, along with the relevant factors applied can be identified for a single results value, along with a summary of loadcases and factors in use.

To stop the creation of printed results at any time, press the Ctrl + Break keys.

Results Types

- ❑ **Components** Default, always enabled.
- ❑ **Eigenvalues** Only enabled if eigenvalue results are present.

The dialog options described below cover both results types, where applicable.

Results Loadcases

When more than one results loadcase is present a choice can be made:

- ❑ **<Current active loadcase name>** - reports on results for this loadcase name only, when the results listing is created. This option creates a utility that is always tied to this named loadcase, and ignores the loadcase that is active when the results are subsequently viewed.
- ❑ **Active** - reports on results for the loadcase that is active at the time the results are being viewed. This may be different from the loadcase that was active when the utility was defined. Setting the **Active** loadcase option will allow results for different loadcases to be created or included in separate reports, simply by changing the Active loadcase in the Analyses Treeview, after each report has been created.
- ❑ **All** - reports on results for all loadcases.
- ❑ **Selected** - reports on results for a specified named loadcase or loadcases. Loadcases can be selected for inclusion in a results listing by selecting a loadcase in the lower 'Available' panel of the dialog and pressing the 'Send to' button to add it to the 'Included' panel. Loadcases can be removed from the

'Included' panel by selecting the row (by its left-most cell) and clicking the 'Send back' button. Entering a Step value of 'n' in the Loadcases panel of the dialog for a selected analysis will add the first and then every 'n'th loadcase of that analysis to the 'Included' panel. Only loadcases that are suitable for combining or enveloping will be listed in the 'Available' panel.

❑ **Final increment** - reports on results for the final results increment from a nonlinear analysis.

Viewing primary and coincident effects

When either a smart combination or an envelope is chosen to display results, one or more primary components can be chosen in the Primary panel of the dialog. Selections then made in the 'Reported component' panel of the dialog are reported as coincident effects. In summary:

- If the Primary checkbox is not checked the results reported on will not be coincident to each other. Instead each will be computed independently.

- If the Primary checkbox is checked only the primary component(s) selected is/are enveloped and all other reported components selected will be coincident effects.

Entity

For results components (not including eigenvalue results) the selection of a results **Entity** displays all available components that can be reported on in the adjacent 'Reported components' panel. Only those components applicable for the elements used in the model will be displayed. The results entities and results types available (when applicable) are:

Eigenvalues

❑ **Eigenvalues (Frequency)** - 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 mass participation factors** - sum the 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, Force/Moment

- ❑ **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.

Location

When applicable, printed results can be obtained for the following locations:

- ❑ **Nodal** prints nodal results for those nodes chosen in the Extent droplist.
- ❑ **Averaged nodal** prints average nodal (smoothed) results for those elements chosen in the Extent droplist. Averaged nodal values are computed from element nodal values (see below). Averages are computed as long as adjacent elements are suitable for averaging (reference to averaging options). Where averaging is not appropriate, multiple values will be presented for each node.

❑ **Gauss Point** or **Internal Point** prints the Gauss point values (for surfaces and volumes) or the internal point values (for line elements) that are internal to those elements chosen in the Extent droplist.

❑ **Inspection Locations** are defined positions of interest on a model for results extraction / viewing purposes.

❑ **Element Nodal** prints unaveraged nodal results for those elements chosen in the Extent droplist.. Nodal results are extrapolated from the Gauss point values within each element.

❑ **Feature maximum** prints the 'maximum' value where the maximum may be a positive or negative number. For instance, for values of -8 and +3, +3 would be the feature maximum. For values of -8 and -3, -3 would be the feature maximum

❑ **Feature minimum** prints the 'minimum' value where the minimum may be a positive or negative number. For instance, for values of -8 and +3, -8 would be the feature minimum. For values of -8 and -3, -3 would be the minimum.

❑ **Feature extreme** prints the value with the largest magnitude, but (unlike an absolute value) preserves its sign, so for example a value of -8 is considered more extreme than +6.

Note that no results summary information can be obtained for feature maximum, feature minimum or feature extreme results, as they results are already considered to be summaries. Note also that feature results will be based on nodal results for continuous results like displacement, and on gauss point results for element results like forces and moments.

Extent

The printed results for selected loadcases are governed by the following extent options:

❑ **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.

❑ **Visible model** - prints results all elements that are visible in the Modeller view window.

❑ **Elements showing results** - (default option) prints results only for those elements that are displaying them in the Modeller view window.

❑ **Selection** - prints results for all selected elements that are visible in the Modeller view window.

❑ **<Group name>** - prints results for all elements in a chosen group regardless of whether the elements are visible or displaying results in the Modeller view window. Only active if group names have been defined.

❑ **New...** - permits creation of a new group of selected or visible items on an empty group.

Output

❑ **Tabular** provides a table of values for all elements / nodes in the extent specified.

❑ **Summary** provides a table of only those nodes / elements in the extent that have the most extreme values

❑ **Tabular and Summary**(the default) provides both of the above.

Order

This specifies the table layout and the tab name to be used on the results listing. The results are grouped into pages (shown as tabs) for the first named item. Each tab then shows a table of all the second named item. For example, selection of 'Loadcase / Feature' creates one page (tab) for each loadcase. Within each page is a table of features, showing values from that loadcase. Selection of 'Feature / Loadcase' shows the same values organised as one page per feature, with each page showing a table of loadcases.

❑ **Loadcase/Feature** (default option)

❑ **Feature/Loadcase**

❑ **Loadcase/Mesh**

Transformation of results

❑ **Transform** Printed results may be transformed to be relative to a specified local coordinate, according to element local directions for stresses, or relative to the local element material directions. Numerous options are available by pressing the **Set...** button. See **Results Transformation** for details.

Additional results options

Displaying extreme envelope / smart combination results

❑ **Display extreme envelope / smart combination results** (Only available if an envelope or a smart combination 'Maximum' or 'Minimum' loadcase has been set active prior to selecting the Print Results Wizard, or selected as a specified loadcase during its use). Includes extreme values obtained by considering both the maximum and minimum value at each location, taking the larger absolute numerical value in each case, and preserving the sign of whichever value is used.

❑ **Display for slice(s)** 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 this 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.

- ❑ **Coordinates** when enabled, includes the global X, Y and Z positions of the nodes or gauss points, or inspection locations (as appropriate) 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.
- ❑ **Display now** When checked, this generates and displays the results data immediately. No printed results utility is saved unless 'Save in treeview' is checked. See below.
- ❑ **Save in treeview** When checked, creates a Print Results Wizard 🔳 entry in the Utilities 🔧 Treeview. These entries can be added to a model report.
- ❑ **Allow derived components** When a derived component is used as a primary component, or when a derived component is inspected for particular situations involving smart combinations and envelopes, LUSAS Modeller will automatically suppress the listing of derived components in the Components panel list as the unwary could get an erroneous answer when reporting on results. This can be overridden by making this selection. The ability to access derived components must be used with caution because results may be misinterpreted in some situations.See **Raw and Derived Components** for details.
- ❑ **Defaults** resets the dialog to default values. Otherwise, when re-accessed, previous settings are retained.

Precision

- ❑ The number of **Significant figures** or **Decimal places** can be specified for values written to the results listing.
- ❑ **Show trailing zeros** - includes the required number of zeros for a chosen number of significant figures.
- ❑ **Threshold value** This is an absolute value, up to which, all values between zero and the threshold value will be written as zero. For example, setting a threshold value of 1E-6 would ensure that any values, which are effectively zero, but would otherwise be written in full in a results listing, would be written as zero (0.0). Once set, a threshold value is retained but the Threshold value check-box is un-set to prevent potentially inappropriate or unnoticed uses of that value on further visits to the dialog.
- ❑ **Name** A name given to identify this table data when an entry is added to the Utilities 🔧 Treeview.

On completion

After making appropriate selections in the Print Results Wizard a table of selected results will be displayed, and a Print Results Wizard 🔳 table entry is added to the Utilities 🔧 Treeview.

A context menu for Print Results Wizard ▦ table entry in the Utilities 🔧 Treeview entry provides the following menu items:

- ❑ **Edit** provides access to the settings made in defining the results data listing.
- ❑ **Copy**, **Paste**, **Rename** and **Delete** options are self-explanatory.
- ❑ **Show Results** allows predefined tables of results data to be re-displayed at any time - without the wizard having to be re-accessed. Double-clicking on the named Print Results Wizard ▦ table entry in the Utilities 🔧 Treeview does the same.
- ❑ **Add to Report** adds the results generated by this print results wizard dataset to a chapter in a **model report**.

Display, manipulation and interrogation of printed results table data

The results for each selected loadcase, or feature (as chosen using the Order control) 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. At discontinuities in the model such as those formed by geometry orientation, geometric assignments, and material assignments, for instance, multiple values will appear for those nodes lying on the discontinuity. These values are denoted by (A), (B), (C) etc within the data.

Table related buttons

On the Printed Results table view window, toolbar buttons provide the means to save, print, preview, add the data to a report, edit its general definition, or

- Press the **Save** button to save the report in a spreadsheet format

- Press the **Print** button 🖨 to print the results.

- Use the **Print preview** button 🔍 to view how the page will print. Zoom and unzoom options are provided. Note that the page orientation is pre-set to be Portrait and is not governed by any settings made on the Page Setup dialog that would be accessed via File > Page setup.

- Use the **Add to Report** button 🗐 to add the results listings as a results chapter in a model report.

- Use the **Edit definition** button ⌨ to revisit the Print Results Wizard dialogs.

- Use the **Examine calculations** button 🔍 to show how a selected value in the print results wizard output has been calculated by listing the envelopes,

combinations or loadcases, and the values from each, along with the relevant factors applied, that produce the selected result.

- Use the **Total Contributions** button ⌕ for a selected results value to reduce the contributing envelopes and combinations to a single table of loadcases and factors in use. Where it is notionally valid to sum the contributing unfactored load effects, these values are also listed in the table. A loadcase may also be created for a result from a smart combination, which includes all the loading assignments and factors which gave that particular result (at a particular node).

Context menu options

When the Printed Results window is displayed a context menu can be invoked (by right-clicking within the window), which allows for the printed results data to be manipulated or interrogated:

- ❑ **Save as Microsoft Excel** The contents of the current tab or all tabs can be saved to a spreadsheet or to a text file.
- ❑ **Sort Ascending / Sort Descending** 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.
- ❑ **Copy** Selected cells or the whole grid can be copied to the clipboard.
- ❑ **Print** The Print option sends the contents of the active tab to the printer.
- ❑ **Definition** Re-invokes the Print Results Wizard dialog for editing the output
- ❑ **Examine Calculations** For a selected result value the contributing envelopes, combinations or loadcases, and the values from each, along with the relevant factors applied can be examined.
- ❑ **Total Contributions** For a selected results value reduces the contributing envelopes and combinations to a single table of loadcases and factors in use. For a smart combination, a loadcase can be created, which includes all the loading assignments and factors which gave a particular result (at a particular node).

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 each component individually, the loadcase from which the results are extracted cannot be tabulated because each individual results component may have come from a different loadcase.

Notes

- Tabulation of beam stress results (such as Sx (Fx,My,Mz) and others) is done for a selected **fibre location**. Note that the specified fibre location does not currently appear in the results listed.

- Selected slice and slideline data and results may be printed from the group context menu.

- To include model images in a model report see **Generating Reports**

- Opening pre-version 16 models that included results chapters will cause equivalent Print Results Wizard entries to be created in the Utilities 🔧 Treeview. These will be saved when the model is saved.

Examine calculations

Selecting the **Examine Calculations** 🔍 option for a selected value in a **Print Results Wizard** output table shows how the result has been calculated. Any envelopes, combinations or loadcases that have been used in the calculation will be listed, along with the values from each, and any relevant factors that were applied.

In the re-sizable dialog presented, the treeview initially shows only the parent loadcase, along with a grid below, showing the loadcases or results components that are contributing to the result shown for that selected entry in the dialog's treeview.

- By selecting a nested loadcase, the grid will update to show the loadcases or results components that are contributing to that selected entry in the treeview. Note that the value shown by a selected entry in the dialog's treeview is identified by bold text in the grid.

- By expanding loadcases and entries of interest in the treeview, loadcases, components and values contributing to loadcases and components higher up the treeview can be identified.

The lowest "level" of the treeview is either an unaveraged raw solver component (e.g. Mz from loadcase 1) or a user input such as a factor from a combination definition.
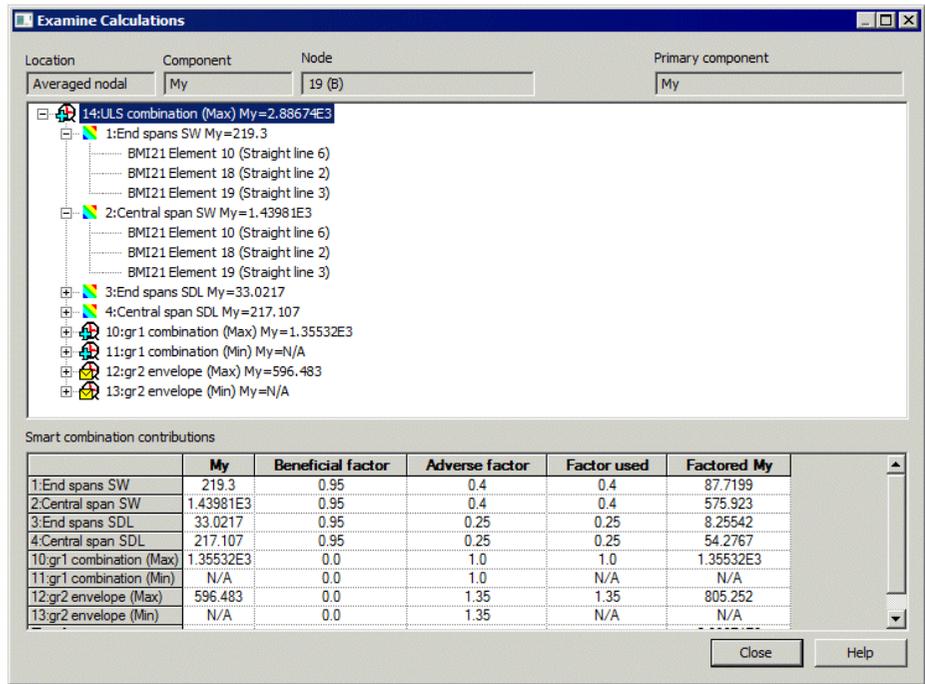
Notes

- A results component marked with an asterisk shows that it is a primary results component.

- It is not possible to examine the calculations used for **derived components**. It is only possible to examine the input that the calculations use. This will include the full set of raw solver components, but might also include attribute values such as Young's modulus or thickness.

- In some situations results components may be reported in the treeview as 'N/A', meaning not available, or not applicable. To give an example of when this may occur, it is important to remember that a combination is built from the sum of factored loadcase values, whether it is smart or basic. The only difference is the way the factors are calculated. For a basic combination, the factors are user inputs. For a smart combination, a relatively complex process calculates the factors that are used, meaning that some constituents are ignored completely. For instance, it is normal practice, when adding a smart combination inside an envelope, to add both max and min "sides" of that smart combination to the envelope. Subsequently, when calculating max envelope results, LUSAS correctly ignores the min "side" of the smart combination, and vice versa, and reports 'N/A' for such ignored components.

- In some situations results components may be reported in the grid as 'Not used'. One example of when this would occur is if the location of the calculation being examined was at a averaging discontinuity, perhaps because of different material or thicknesses being assigned to surfaces either side of the location.

Total contributions

Selecting the **Total contributions** ![search icon] option for a selected results value in the **Print Results Wizard** output table reduces the contributing envelopes and combinations to a single table of loadcases and factors in use. Where it is notionally valid to sum the contributing unfactored load effects, these values are also listed in the table generated.
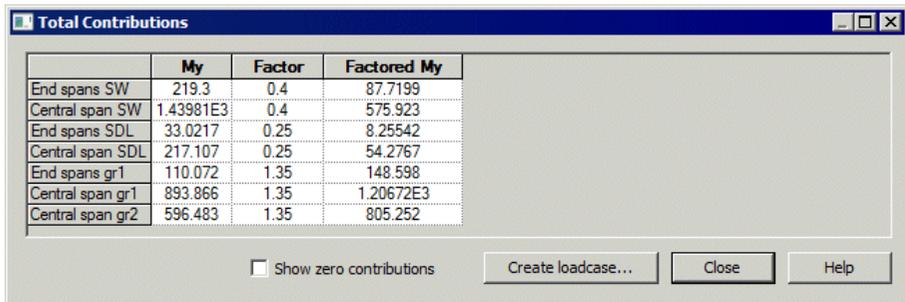
	My	Factor	Factored My
End spans SW	219.3	0.4	87.7199
Central span SW	1.43981E3	0.4	575.923
End spans SDL	33.0217	0.25	8.25542
Central span SDL	217.107	0.25	54.2767
End spans gr1	110.072	1.35	148.598
Central span gr1	893.866	1.35	1.20672E3
Central span gr2	596.483	1.35	805.252

- ❑ **Show zero values** Any loadcase that has been considered and rejected (e.g. by an envelope) will not be listed in the table. They can be optionally shown with a factor of 0.0.
- ❑ **Create loadcase** Creates a loadcase, for a smart combination, which includes all the loading assignments and factors which gave a particular result (at a particular node). This may be useful for those wishing to use an onerous loadcase identified by a linear analysis as part of a further nonlinear analysis, for example. When solved, that loadcase would yield all the coincident effects that occur at the time when the original chosen component is at its maximum (or minimum) value at the location chosen. No default name is provided for this loadcase. A suitable Analysis entry and loadcase name must be specified.

Printing and Saving Views of a Model

Printing a view of a model

Views of a LUSAS model may be printed directly to the default printer by using the print ![printer icon] 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 a view of a model

For use in model reports

See **Saved Views** for more information.

Saving a view as a specified file type

Views of the LUSAS model can be saved as BMP, JPG, PNG files, or as LUSAS Picture files using the **File> Picture Save** menu item.

BMP, JPG, and PNG files can be saved to a user-defined pixel width (or height) where the height (or width) not specified is proportional to the size of the graphics window when the file was saved. JPEG and PNG files are the most efficient to save in terms of file size. BMP, JPG, and PNG files can be included in a model report, but use of saved views should be used instead if automatic updating of images in a report is required.

Views saved as a **LUSAS Picture File** can only be viewed using the LUSAS picture file utility and legacy viewer, Expose. Note that Graphs cannot be saved in LUSAS picture file format. Picture files cannot be included in a model report.

Copying the View Window

The contents of the View window can also be copied (and subsequently pasted) into other applications by clicking the right mouse button in the View Window and selecting **Copy** from the context menu. In this instance the pixel width (or height, if greater) is set to be a maximum of 1800 pixels with the other dimension proportional to the view window size.

The contents of the view window can also be transferred to the Windows clipboard using the copy 📋 button.

Generating Reports

A customised report can be built in the Reports 📄 Treeview by defining chapters that reference the type of 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 particular results entities. Data can be listed for all or selected parts of a model. Chapters containing user content in the form of captured or saved images can be included, as well as graphs, saved
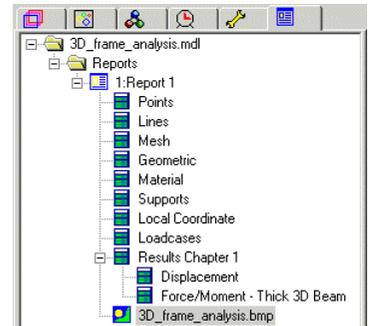
views of the model, or one-click reports.

Note that with the exception of user-captured or user-inserted images, and graphs that have been specifically set to not auto-update, all remaining assembled report data is automatically updated each time a report is viewed to reflect the current state of a model.

Report templates

A report template (which is created, modified and saved in the Reports 📄 Treeview for each model) holds the information required to generate reports that can include model properties 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. Each report template can include any number of chapters that define the model attributes and analysis results.

A new (empty) report template is created by selecting the **Utilities > Report...** menu item and selecting the **New report** menu item, or from right-clicking **New Report** from the **Reports** context menu in the Report 📄 Treeview.

Any number of report 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.

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) and other formats.

If results are required only to be displayed on-screen (rather than be assembled for printing in a report) the **Print Results Wizard** may be a better choice.

Creating a Report

A new (empty) report template is created by selecting the **Utilities> Reporting...** menu item and selecting the **New Report** menu item, or from right-clicking on the Reports folder in the Reports 📄 Treeview, and selecting the **New Report** menu item from the context menu.

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 most values seen in the report will be converted appropriately. See notes for details of which types of values are not converted.

❑ **Front page view** This permits a **saved view** of the model to be used as an image on the first page of the report for identification purposes.

❑ 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. **Show trailing zeros** includes the required number of zeros for a chosen number of significant figures.

❑ **Threshold value** This is an absolute value, up to which all values between zero and the threshold value will be written as zero. For example, setting a threshold value of 1e-6 would ensure that any values in the range of -1e-6 < 0 < 1e-6 , which are effectively zero but would otherwise be written in full in a results listing, would be written as zero (0.0). Once set, a threshold value is retained but on further visits to the dialog the Threshold value checkbox is un-ticked to prevent potentially inappropriate or unnoticed uses of that value.

❑ **Precision for all chapters** This, if selected, uses the values for significant figures or decimal places chosen on this dialog throughout the report. This option overrides any different values set inside each report chapter.

❑ **Page break between chapters** forces page breaks where stated.

❑ The **Report** name is the name added to the Reports 🖻 Treeview. By default reports are named Rpt1, Rpt2 etc. if no name is specified.

Once a New Report entry has been added to the Reports 🖻 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 Reports 🖻 Treeview when a model is saved.

Note that all model units will be converted to a different system of report units with the following exceptions:

- Graphs and graphed data are not converted to selected report units. However, graphing units are stated clearly within the image of the graph itself.

- Slab design units are only partially converted to report units. The units of codified values are retained. For example if a value such as reinforcement bar area per metre width is 1200 mm^2/m and a report in feet is requested, then the value will be converted to (approx) 400 mm^2/ft. The units of steel area are not converted because they represent a codified value.

- User Defined Results cannot be converted.

Report symbols explained

See "Treeview Panels and Icons" on page 19 for details of all symbols used in the Reports 🖻 Treeview.

Adding a Report Chapter

Model properties, loadcase results, user content, graphs, prestress data, saved views of a model, notes and other data can all be added to a model report as a report chapter in the Reports 🗐 Treeview.

Chapters can be added to a report by selecting the **Add Chapter** menu item from the Report name context menu (accessed by right-clicking the mouse button), or from the context menu of an existing report chapter entry in the Reports 🗐 Treeview - in which case the new chapter will be inserted to follow that selected chapter.

The following can be added using the Chapters dialog:

❑ **Model Properties** - Model geometry, attribute and model-related loadcase data.

❑ **Results** - any or all loadcase results data.

❑ **User Content** - user-captured or supplied images, or text.

❑ **Graphs** These can also be added using the **Add to Report** button in the generated window when a Graph is displayed, or from a context menu option for the graph entry in the Utilities 🔧 Treeview.

❑ **Prestress** Tendon properties, profile, loading, losses and setting -out data for all tendons defined and assigned in a model can be tabulated.

❑ **Saved Views** - a saved view of the model that is updated to reflect the current state of the model each time a report is generated. A saved view can also be added using the **Add to Report** context menu option for a saved view entry in the Utilities 🔧 Treeview.

❑ **Notes** - general user-defined notes relating to one or many chapters. Notes can also be added using the **Add to Report** context menu option for a saved view entry in the Utilities 🔧 Treeview.

❑ **Other** - includes Response Spectra utilities, if created.

Additional related information

Chapters for particular results can also be added by selecting the **Add to Report** button in the generated report window for that result, or from the Utility name context menu for that result entry. See below for details.

The following can be added to a report by using the **Add to Report** context menu item for the named entry in the Utilities 🔧 Treeview:

❑ **Print Results Wizard** results

The following can be added to a report by using the **Add to Report** button in the generated window for that report:

❑ **One-click report**

❑ **Steel design code member report**

Add or Edit a Model Properties Chapter
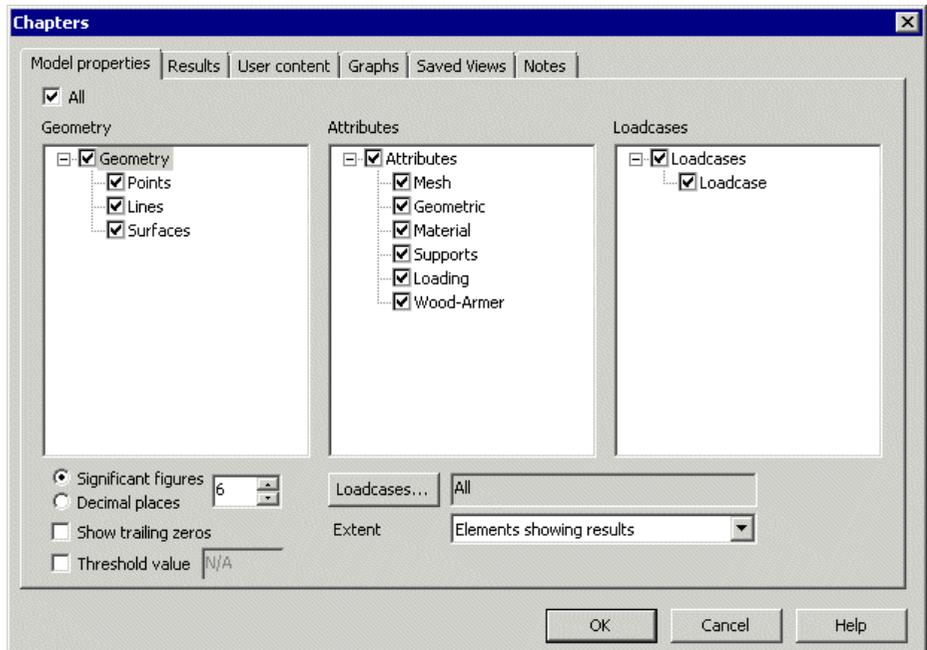
The **Model Properties** tab of the Chapters dialog shows Geometry, Attributes and Loadcase panels that allow model geometry, attribute and model-related loadcase /IMD loadcase information to be added to the report via the use of check boxes.

❑ The number of **Significant figures** or **Decimal places** can be specified for values written to the results listing.

❑ **Show trailing zeros** - includes the required number of zeros for a chosen number of significant figures.

❑ **Threshold value** This is an absolute value, up to which, all values between zero and the threshold value will be written as zero. For example, setting a threshold value of 1e-6 would ensure that any values in the range of -1e-6 < 0 < 1-e6 , which are effectively zero but would otherwise be written in full, would be written as zero (0.0). Once set, a threshold value is retained but the Threshold value check-box is un-set on further visits to the dialog to prevent potentially inappropriate or unnoticed uses of that value.

❑ Use the **Loadcases…** button to display a dialog which restricts the chapter to display data 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.

❑ Use the **Extent** droplist to define the scope or the extent of the chapter to be created, for example to restrict the chapter to display data for the **Full model**, the **Visible model**, **Elements showing results**, a **Selection**, or a **Specified group** only. See **Chapter Extent** for more details.

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 Loadcase Results Chapter
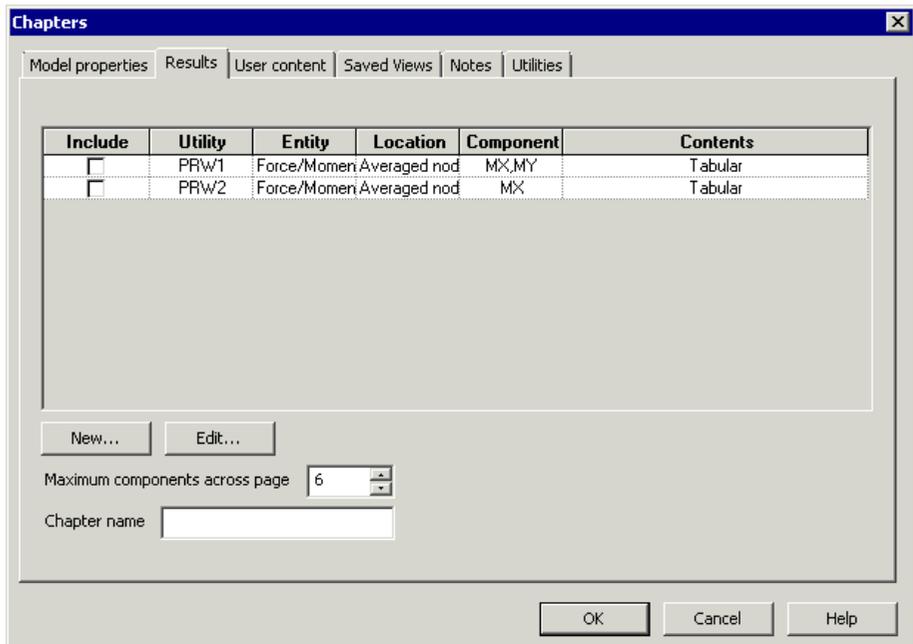
The **Results** tab of the Chapters dialog is always shown regardless of any results being available. This enables a report to be assembled prior to solving the model, should that be desired.

Any previously created Print Results Wizard utilities will be listed in the grid for inclusion in a report chapter. Click the **Include** checkbox to add each defined listing of results to the report as a sub-chapter.

❑ Use the **New...** button to add new entities for inclusion in the report as results sub-chapters by using the **Print Results Wizard**. When added, results sub-chapters are shown in the grid. A corresponding Print Results Wizard entry is added to the Utilities 🔧 Treeview for the selection made.

❑ Use the **Edit** button to change the results selection to be included in a report chapter. Prior to pressing this button a row of the grid must be selected by clicking inside the first cell of the row.

❑ The **Include** checkbox (seen when results are present) will include or exclude results listings from the report.

❑ **Maximum components across page** restricts the results components listings to a specified number of columns.

❑ **Chapter name** can be entered or edited if the default or previously entered name is to be altered. It is suggested that the Results Chapter include a reference to the Loadcases and Extent included.

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.

Deleting loadcase results chapters

It is not possible to delete loadcase results chapter entries from the grid on this dialog directly, or from the Report 📇 Treeview itself. Instead, the 'Include' checkbox can be un-checked on this dialog to remove a Print Results Wizard utility from the model report chapter, and the Print Results Wizard utility can also be deleted from the Utilities 🔧 Treeview using the context menu provided, if it is not otherwise needed.

Add or Edit a User Content Chapter
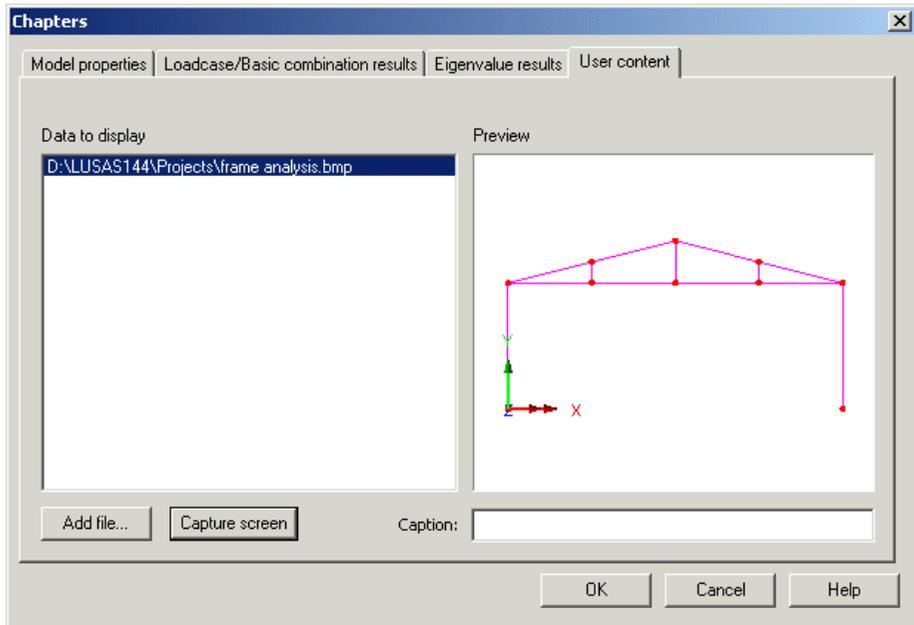
The **User Content** tab of the Chapters dialog allows images or text files to be added to the report

- ❑ The **Add file...** button allows adding existing saved images or text to the report.
- ❑ The **Capture screen** button takes a snapshot of the current view window and allows it to be saved as a fixed-size BMP, JPG or PNG 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. **Note that screen captures do not automatically update to reflect any changes made to the model or its re-calculated results if the report is re-generated**. If it is required to have all model and results-related images automatically update each time a report is generated add a **Saved view chapter** instead.
- ❑ 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.

Add or Edit a Graph Chapter
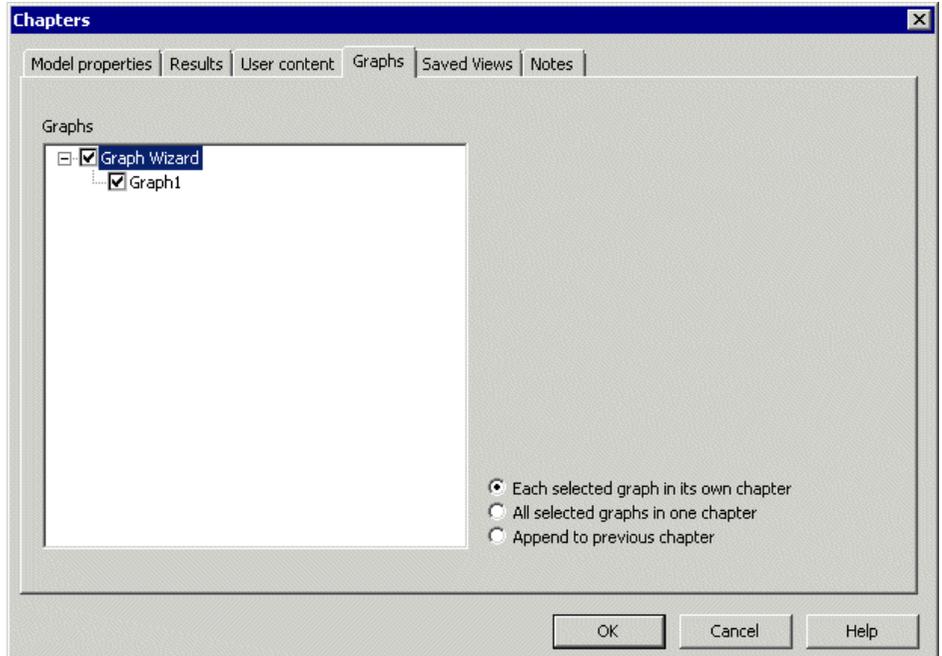
The **Graphs** tab of the Chapters dialog is only present if Graph Wizard-created graphs are present in the Utilities 🔧 Treeview.

Graphs defined by Graph Wizard can be added to a report as a report chapter. Graphs that were defined and added to the Utilities 🔧 Treeview are initially listed in the Graphs panel of the Chapters dialog. Check (tick) the appropriate graphs to include those graphs as a chapter in the report. Note that the name of a graph is automatically used as a title beneath the inserted graph in the report.

Self-explanatory options when adding graphs to a report are:

- ❑ **Each selected graph in its own chapter**
- ❑ **All selected graphs in one chapter**
- ❑ **Append to previous chapter**

Notes:

- 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.

- Each time a report is generated graphs included in a report will update to reflect any changes that may have been made to the model since the graph was last displayed, unless the auto-update feature has been turned-off.

- Graph chapters can also be added using the **Add to report** button in the generated window when a Graph is displayed, or from a context menu option for the graph entry in the Utilities 🔧 Treeview.

Add or Edit a Prestress Results Chapter
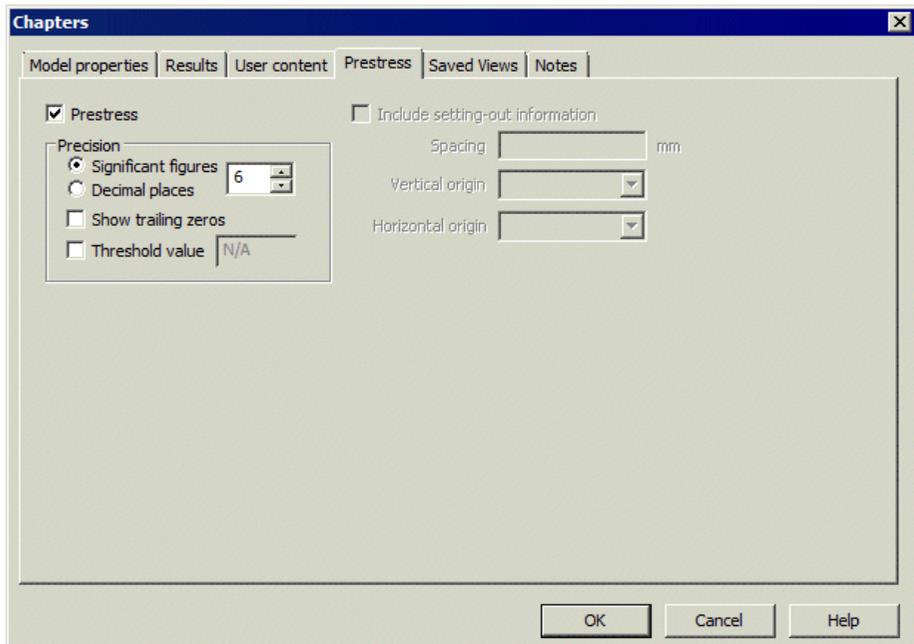
The **Prestress** tab of the Chapters dialog is shown if at least one tendon loading has been assigned in a model.

Tendon properties, profile, loading, losses and setting -out data for all tendons defined and assigned in a model can be reported in a prestress chapter.

Precision

Independent precision options for a prestress chapter can be set if the precision settings for the report as a whole are not set. See **Creating a Report**.

❑ The number of **Significant figures** or **Decimal places** can be specified for values written to the results listing.

❑ **Show trailing zeros** - includes the required number of zeros for a chosen number of significant figures.

❑ **Threshold value** This is an absolute value, up to which, all values between zero and the threshold value will be written as zero. For example, setting a threshold value of 1E-6 would ensure that any values, which are effectively zero, but would otherwise be written in full in a results listing, would be written as zero (0.0). Once set, a threshold value is retained but the Threshold value check-box is un-set to prevent potentially inappropriate or unnoticed uses of that value on further visits to the dialog.

Including setting-out information

❑ **Include setting-out information** The calculated tendon profile shape (after all smoothing has been applied) can be included in a tendon summary report in a format that is suitable for setting-out the tendon on-site.

• Tendon profile setting out data can be tabulated for a defined **Spacing** along a beam with reference to **Vertical origin** and **Horizontal origin**. The origin settings can be set to be an Assigned line or the top or bottom, left or right, or centre of a geometrically defined cross-section. Note that 'Section centre' may differ from 'Assigned line' by any eccentricity that may be applied.

• Each tendon assignment writes a separate setting-out chapter where the setting-out tendon profile information is stated as relative distances between setting-out points. In some cases, several assignments may generate duplicate tables of data, for example where the same profile has been used on many lines, but the details of the assigned lines may be different so a separate report is provided as a safety measure.

• At any given point on the structure the interpretation of 'Section top', 'Section bottom', 'Section left' or 'Section right' depends upon the specified vertical direction (set using Utilities > Vertical Axis), the assigned line, and the direction of increasing x distance along the tendon profile. 'Section top' is always interpreted as vertical, which (for non-level decks) is not necessarily perpendicular to the section at that point. A 'forwards' direction is always along the assigned line, but may be opposite to the line's x-axis, as it must follow the tendon's increasing x value. 'Section left' and 'Section right' are calculated from the cross-product between the 'up' and 'forwards' directions at each point.
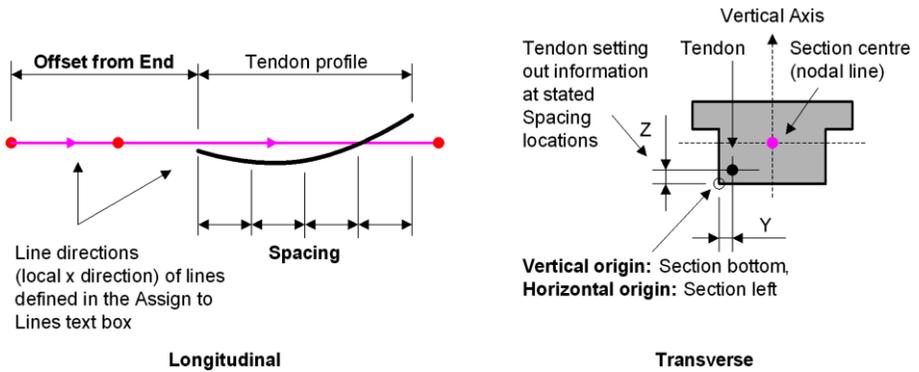
Selected tendon profile setting-out terms explained

When a report is generated, prestress results data for all tendon loading can be presented in a report.

Prestress

Tendon Summary - Tendon 1 assignment 1

General	Value	Units
Design code	EN1992-1-1:2004 / 2014 Eurocode 2	
Initial tendon force	4.10000E3	kN
Tendon Area	3.60000E3	mm²
Modulus of elasticity for tendon	195.000E6	kN/m²
Jacking	Jacking at end 1	None
Slip at end 1	5.00000E-3	m
Jack angle at end 1	33.0000	deg
Instantaneous losses	**Value**	**Units**
Unintentional angular displacement	0.0100000	rad/m
Duct friction coefficient	0.190000	
Time-dependent losses	**Value**	**Units**
Tensile strength of prestressing steel	1.86000E3	N/mm²
Relaxation class	Class 2	
Relaxation loss at 1000hrs	2.50000	%

Tendon Profile (original input) - Tendon 1 assignment 1

Segment type	X	Y	Z
Start	0.0	0.0	0.750000
Straight	6.00000	0.0	0.750000

Minimum radius = 0.0
Smoothing type: Cut corner

If the prestress results become out-of-date an appropriate message will appear in the report. A re-solve of the model will update the report the next time it is requested.

Add or Edit a Saved View Chapter
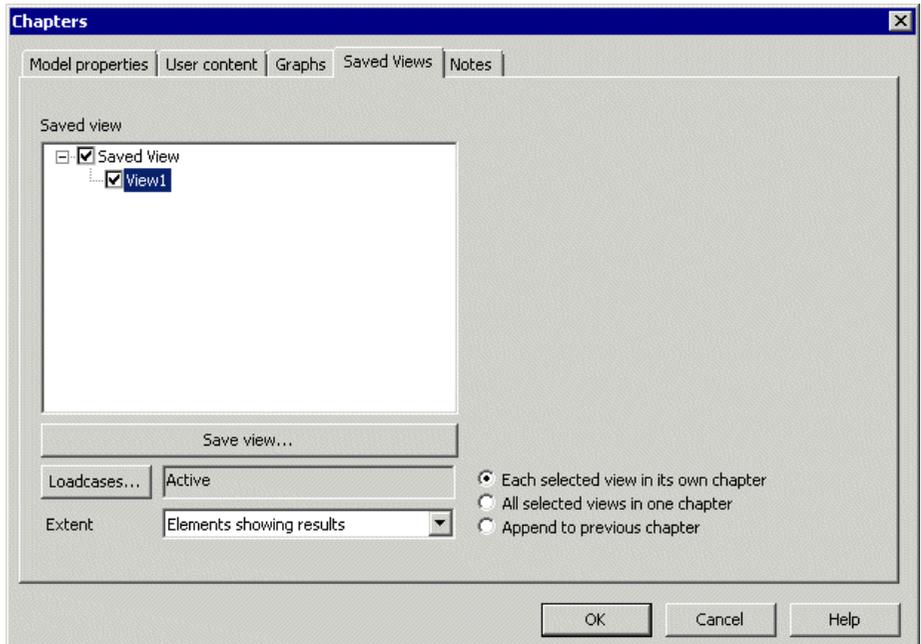
The **Saved Views** tab of the Chapters dialog is always present.

Saved views or the current view can be added to a report as a report chapter. Any saved views that were defined and added to the Utilities 🔧 Treeview are listed in the Saved views treeview panel of the Chapters dialog.

Check (tick) the appropriate checkbox to include the contents of that view as a chapter in the report. Note that a title for the view is automatically added beneath the inserted image of the view in the report. This title is comprised of the Saved view name followed by a loadcase name in parentheses as, for example, 'Contours Fx (Loadcase 1)'.

❑ Use the **Save view...** button to save the current active view into the Saved view list.

❑ Use the **Loadcases…** button to display a dialog which restricts the chapter to display results for **All**, **Active** or **Specified** loadcases or combinations.

❑ The **Extent** droplist controls the scope or the extent of the chapter to be created, for example to restrict the chapter to display results for the **Full model**, the **Visible model**, **Elements showing results**, a **Selection**, or a **Specified group** only. See **Chapter Extent** for more details.

Self-explanatory options when adding saved views to a report are:

❑ **Each selected view in its own chapter**

❑ **All selected view in one chapter**

❑ **Append to previous chapter**

Notes:

- It is possible to visit this dialog several times to create multiple chapters, each of which can have different ordering, scope and loadcase choices.

- Once created, the order of chapters in a report can be modified at any time by dragging and dropping within the Report 🖹 Treeview.

- Each time a report is generated the contents of a saved view chapter are updated to reflect the current state of the model and/or results referenced by the saved view.

- Saved view chapters can also be added using the **Add to Report** context menu option for a saved view entry in the Utilities 🔧 Treeview.

Add or Edit a Notes Chapter
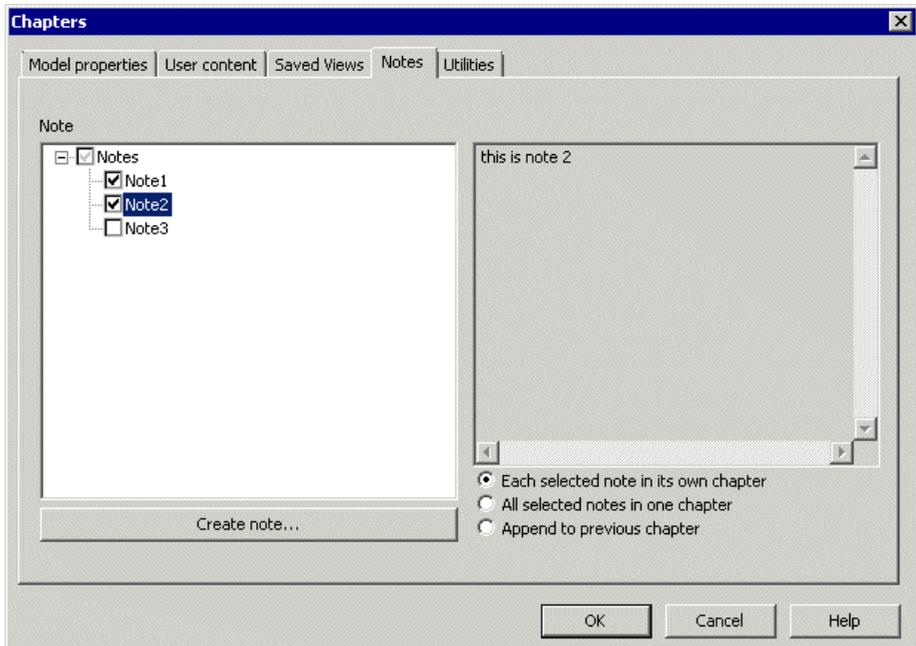
The **Notes** tab of the Chapters dialog is always present.

Notes can be added to a report as a report chapter. Any notes that were defined and added to the Utilities 🔧 Treeview by using the **Utilities > Notes...** menu item are listed in the Notes treeview panel of the Chapters dialog.

- ❏ Use the checkboxes to select which notes to include in this chapter.
- ❏ Use the **Create note** button to define additional notes. When these notes are defined they will be saved as entry within the Utilities 🔧 Treeview.

Note adding options are:

- ❏ **Each selected note in its own chapter**
- ❏ **All selected notes in one chapter**
- ❏ **Append to previous chapter**

Notes:

- It is possible to visit this dialog several times to create multiple chapters.

- Once created, the order of chapters in a report can be modified at any time by dragging and dropping them within the Report 📄 Treeview.

- Notes chapters can also be added using the **Add to report** context menu option for a note entry in the Utilities 🔧 Treeview.

Add or Edit an 'Other' Chapter
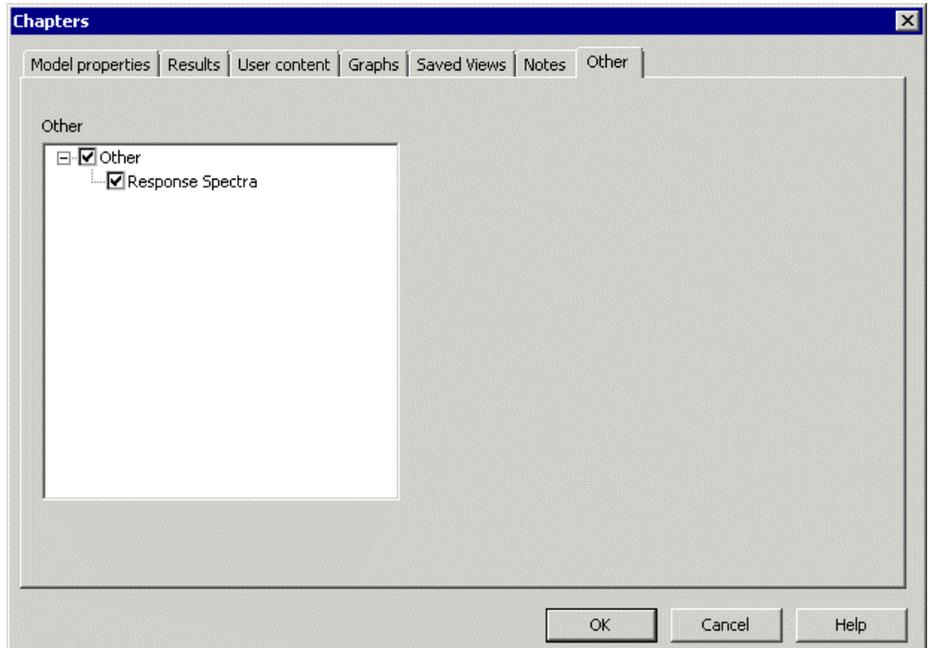
The **Other** tab of the Chapters dialog is only shown if any of the tree-frame entries it can list are available.

Response Spectra utilities are an entry that will be listed in the tree-frame for optional inclusion in a report chapter for this tab setting. Checking (ticking) each entry will add a single report chapter for each, but containing all the defined utilities for these selections. Relevant data will be tabulated and accompanying response spectra graphs automatically added to the report.

Notes:

- Once created, the order of chapters in a 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 particular report chapters can report on can be specified by selecting one of the following options on the relevant dialog:

- ❑ **Full model** - default option which reports for all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.
- ❑ **Visible model** - reports for all elements that are visible in the Modeller view window at the time the report is viewed.
- ❑ **Elements showing results** - reports for those elements that are displaying results in the Modeller view window at the time the report is viewed.
- ❑ **Specified group** - reports for a specified named group. Options are: Selected Items, Visible Items, Empty Group.

Loadcase / Basic Combination Selection

The loadcases, combinations or envelopes that the model properties chapter is to report on can be specified by selecting either:

- ❑ **<Current active loadcase name>** - reports on results for this loadcase name only.
- ❑ **Active** - reports on results for the loadcase that is active at the time the results are being viewed. Setting this option will allow results for different loadcases to be created or included in separate reports, simply by changing the Active loadcase in the Analyses Treeview, after each report has been created.
- ❑ **All** - reports on results for all loadcases.
- ❑ **Specified** - reports for a specified named loadcase.

If multiple loadcases are specified, multiple entries will appear in the report.

Viewing a Report

Reports are viewed from the Report ▣ Treeview by choosing the **View Report...** menu item from the report name context menu. By default, and after time is taken to assemble and format the report data, a report consisting of all the selected chapters will be created as a PDF file and displayed on screen if a PDF viewer is installed on the PC is use.

Exporting Report Data

Model report data may be exported to a chosen file location in a variety of formats by use of the **Export** context menu item for each report entry, as listed in the Reports ▣ Treeview.

File formats permitted include:

- ❑ **Portable Document Format (*.pdf)**
- ❑ **Rich Text Format (*.rtf)**
- ❑ **Microsoft Excel 97-2003 (*.xls)**
- ❑ **Tab Separated Text (*.ttx)**
- ❑ **Print Files (*.prn)**

Note: 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.

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 beneficial and adverse 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 beneficial factors given for negative load effects and using adverse factors for positive load effects.

Loadcase	Nodal result	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results
Dead load	-20	1.0	1.15	Beneficial	-20 x 1.0 = -20
Deck surfacing	-10	1.0	1.75	Beneficial	-10 x 1.0 = -10
Superimposed load	-15	1.0	1.2	Beneficial	-15 x 1.0 = -15
				Smart combination (Max)	**-45**

Smart combination (Min) will assemble results from the loadcases using just the beneficial factors given for positive load effects and using beneficial + adverse factors for negative load effects.

Loadcase	Nodal result	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results
Dead load	-20	1.0	1.15	Adverse	-20 x 1.15 = -23
Deck surfacing	-10	1.0	1.75	Adverse	-10 x 1.75 = -17.5
Superimposed load	-15	1.0	1.2	Adverse	-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 beneficial and adverse 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 beneficial factors given for negative load effects and using adverse factors for positive load effects.

Loadcase	Nodal result	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results
Temperature	-5	0	1.3	Beneficial	-5 x 0 = 0
Wind	-5	0	1.4	Beneficial	-5 x 0 = 0
Settlement	-10	0	1.2	Beneficial	-10 x 0 = 0
Live load 1	-20	0	1.5	Beneficial	-20 x 0 = 0
Live load 2	-15	0	1.5	Beneficial	-15 x 0 = 0
Live load 3	10	0	1.5	Adverse	10 x 1.5 = 15
Live load 4	-5	0	1.5	Beneficial	-5 x 0 = 0

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the beneficial factors given for positive load effects and using adverse factors for negative load effects.

Loadcase	Nodal result	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results
Temperature	-5	0	1.3	Adverse	-5 x 1.3 = -6.5
Wind	-5	0	1.4	Adverse	-5 x 1.4 = -7
Settlement	-10	0	1.2	Adverse	-10 x 1.2 = -12
Live load 1	-20	0	1.5	Adverse	-20 x 1.5 = -30
Live load 2	-15	0	1.5	Adverse	-15 x 1.5 = -22.5
Live load 3	10	0	1.5	Beneficial	10 x 0 = 0
Live load 4	-5	0	1.5	Adverse	-5 x 1.5 = -7.5

Smart combination (Min) = -85.5

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 beneficial and adverse 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 effects will be combined for the Max combination and the four most negative effects will be combined for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the beneficial factors given for negative load effects and using adverse 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	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	**Used**
Wind	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	**Used**
Settlement	-10	0.7	1.5	Beneficial	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	1.5	Beneficial	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	1.5	Beneficial	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	1.5	Adverse	10 x 1.5 = 15	**Used**
Live load 4	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	**Used**

Smart combination (Max) = 4.5

Smart combination (Min) will assemble results from the loadcases using just the beneficial factors given for positive load effects and using adverse 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	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	1.5	Adverse	-5 x 1.5 = -7.5	**Used**
Wind	-5	0.7	1.5	Adverse	-5 x 1.5 = -7.5	Not used
Settlement	-10	0.7	1.5	Adverse	-10 x 1.5 = -15	**Used**
Live load 1	-20	0.7	1.5	Adverse	-20 x 1.5 = -30	**Used**
Live load 2	-15	0.7	1.5	Adverse	-15 x 1.5 = -22.5	**Used**
Live load 3	10	0.7	1.5	Beneficial	10 x 0.7 = 7	Not used
Live load 4	-5	0.7	1.5	Adverse	-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 beneficial and adverse 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. With the number of "loadcases to consider" set to four, only the four most positive effects will be combined for the Max combination and the four most negative effects 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 beneficial factors given for negative load effects and using adverse 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 beneficial 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	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used
Wind	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	1.5	Beneficial	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	1.5	Beneficial	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	1.5	Beneficial	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	1.5	Adverse	10 x 1.5 = 15	**Used**
Live load 4	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the beneficial factors given for positive load effects and using adverse 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 beneficial 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	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	1.5	Adverse	-5 x 1.5 = -7.5	**Used**
Wind	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	1.5	Adverse	-10 x 1.5 = -15	**Used**
Live load 1	-20	0.7	1.5	Adverse	-20 x 1.5 = -30	**Used**
Live load 2	-15	0.7	1.5	Adverse	-15 x 1.5 = -22.5	**Used**
Live load 3	10	0.7	1.5	Beneficial	10 x 0.7 = 7	Not used
Live load 4	-5	0.7	1.5	Beneficial	-5 x 1.5 = -7.5	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 beneficial + adverse load factors will only be considered for the number of loadcases specified as the number of Variable loadcases to consider. The remaining three loadcases are considered using the beneficial factor. With the number of "loadcases to consider" set to four, only the four most positive effects will be combined for the Max combination and the four most negative effects 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 beneficial factors given for negative load effects and using adverse 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 beneficial 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	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used
Wind	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	1.5	Beneficial	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	1.5	Beneficial	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	1.5	Beneficial	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	1.5	Adverse	10 x 1.5 = 15	**Used**
Live load 4	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the beneficial factors given for positive load effects and using adverse 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 beneficial 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	Beneficial factor	Adverse factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	**Used**
Wind	-5	0.7	1.5	Beneficial	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	1.5	Beneficial	-10 x 0.7 = -7	**Used**
Live load 1	-20	0.7	1.5	Adverse	-20 x 1.5 = -30	**Used**
Live load 2	-15	0.7	1.5	Beneficial	-15 x 0.7 = -10.5	**Used**
Live load 3	10	0.7	1.5	Adverse	10 x 1.5 = 15	Not used
Live load 4	-5	0.7	1.5	Beneficial	-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

$$K = \begin{bmatrix} EA/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 12EI/L^3 \end{bmatrix} \begin{matrix} (u) \\ (v) \\ (\theta) \end{matrix}$$

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 is 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 is explicit dynamic elements which 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.

- 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:
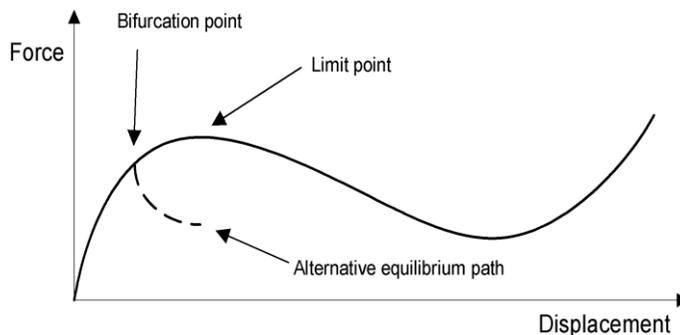
Such a bifurcation point could permit another, non-physical, solution path to be followed because, numerically, it requires less energy.

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 : Model Selection and Viewing 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**
Select geometry (any feature type)	= + **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 (displayed for mesh-only models only) = + **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.

Planar selection options

These can be used to define planes in which, or axes along which features can be selected. When used in isolation all features/objects in a chosen axis or plane will be selected within a cursor selection. When a planar option is set prior to using the previously mentioned Feature/Mesh Object Selection option individual points or lines or elements etc can be selected within the axis or plane chosen.

Axes and planes are defined by model **view axes** and the model **vertical axis**.

Specific cursor	= Notes
Select Any	Selects any features/objects within the selection made
Select Vertical only	Select features/objects that are vertical with respect to the model vertical axis
Select Horizontal only	Select features/objects that are horizontal with respect to the model vertical axis
Select X axis only	Select features/objects in the X direction only with respect to the model vertical axis and in relation to any active local coordinate set
Select Y axis only only	Select features/objects in the Y direction only with respect to the model vertical axis and in relation to any active local coordinate set

 Select Z axis only — Select features/objects in the Z direction only with respect to the model vertical axis and in relation to any active local coordinate set

 Select XY plane only — Select features/objects that are in the XY plane with respect to the model vertical axis and in relation to any active local coordinate set

 Select YZ plane only — Select features/objects that are in the YZ plane with respect to the model vertical axis and in relation to any active local coordinate set

 Select XZ plane only — Select features/objects that are in the XZ plane with respect to the model vertical axis and in relation to any active local coordinate set

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

Cumulative selection

Note that cursor selection options are cumulative, so it is possible, for example, to make a rectangular selection, of only all the vertical surfaces within a model. This would be achieved by selecting the Select Vertical only toolbar button, prior to either selecting the Surface selection toolbar button, or selecting the Rectangle Select toolbar button whilst holding down the **S** key to limit the selection to just Surfaces.

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 Selection panel which can be displayed from the **View> Selection Panels** menu. 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.
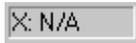
 = View along the +X axis towards the origin
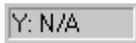
 = View along the +Y axis towards the origin

 = 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:
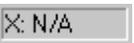
Shift key + = View along the -X axis towards the origin
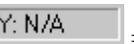
Ctrl key + = View along the -Y axis towards the origin
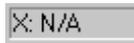
Ctrl + **Shift** key + = View along the -Z axis towards the origin

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 / Refresh

Appendix D : Real Numbers and Expressions in LUSAS

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 supported:

Arithmetic

```
(A), -A, A+B, A-B, A*B, A/B, A^B, ceil(A), floor(A),
abs(A), max(A, B, C...), min(A, B, C...), sum(A, B,
C...) average(A, B, C...) ,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, ceil, floor, abs,
```

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
`sign()`	Returns 1 for positive numbers, 0 for zero and -1 for negative numbers.
`convertFromSI`	Takes two arguments. The first can be any expression. The second is assumed to be a packed integer, which within LUSAS can come from the standard KSharedEnum implementation, or alternatively from the new packDimension() function below. The input value will be converted from "N,m,kg,s,C" into model units.
`case()`	Takes any number of arguments, assumes that the last argument is an error condition, and that all other arguments are in pairs of condition and value. If the first condition is true, the first value will be returned, otherwise control passes to the second condition – and so on until the last argument, which will be returned when none of the conditions is met.
`error()`	Takes only one argument, which may be a string or an integer error code – i.e. x = error("text to describe the error") or x = error(12345). In practice, the function never returns a value – instead it stops execution of the current expression, and stores the given message / error code. At the end of drawing the screen / preparing the report / etc, Modeller checks to see if any errors have been stored, and if so displays them to the user.
`none()`	Takes no arguments. In practice, the function never returns a value – instead it stops execution of the current expression, and prevents the program displaying any results for the current location.
`isNumber()`	Takes one argument. It returns 'true' if the only argument is a valid number. It returns 'false' in response to any

	numerical error, such as isNumber(log(-1)) or isNumber(none()).
`isDefined()`	Takes one argument. It returns 'true' if the only argument is a valid variable name. E.g. isDefined("displacement.DX") is true, whilst isDefined("displacement.DZ") would return 'false' in a 2d analysis.
`member.parametric()` `parent.parametric()`	A parametric distance is given along the parent line, which enables the program to indentify an element. The parametric distance is converted to a parametric distance along that element, and then the element is asked to return the value of the given named value. E.g. member.parametric(0.5, "My") returns the mid-span bending moment for a line. If the parent line belongs to a combined line, then the member variant treats the parametric distance as parametric within that combined line. If the parent line does not belong to a combined line, the two functions return the same result.
`member.max()` `member.min()` `parent.max()` `parent.min()`	These simply loop all the elements along the parent / member and return the numerically largest or smallest number encountered.
`lookup()`	Takes three arguments. The first argument is the name of a table of values, which must be attached to an attribute assigned to the element in question. The second and third arguments are simple indices into this table. For example, lookup(geometric.myTable, 2, 3) returns the value at location (2,3) in the variable "myTable" which must be in the geometric attribute assigned at the node, element node, gauss point, or internal calculation point, depending upon which has been selected.
`packDimension()`	converts simple separate integers into the packed integer required for the convertFromSI function. E.g. stressUnit = packDimension(1,-2,0,0,0) means "force / area".
`interp()`	Takes 7 arguments. The first argument is the name of a table of values. The second argument is the name of a vector of row header values, in which to search for the third argument. Similarly, the fifth argument is the name of a vector of column header values, in which to search for the sixth argument. The fourth and last arguments are mode switches for row and column respectively. A mode of 0 requires an exact match. A mode of 1 returns the nearest match. A mode of 2 returns an exact match if possible, but

otherwise the highest value which is smaller than the given value. A mode of 3 returns an exact match if possible, but otherwise the smallest value which is higher than the given value. A mode of 4 performs linear interpolation between adjacent cells. Having determined the modes, the program searches the row and column headers for the input values. Having identified row and column numbers, it can then return (or interpolate) values from the main table.

equroot	Takes two arguments. Returns a single root of a polynomial defined by the given array of coefficients.
pwl	Takes two arguments. Returns y given x given an array of values defining a 2D graph
pwlt	Takes two arguments. Returns y given x given an array of values defining a 2D graph and returns an error if an attempt is made to interpolate beyond the extent of the graph.

Variables

parent.type **parent.length**	The value of type is 1 for points, 2 for lines, 3 for surfaces and 4 for volumes, allowing expressions like "IF (parent.type = 1, ..." to be created. Having checked that the parent type is indeed a line, the variable 'length' may safely be used, as: L = IF(parent.type = 1, parent.length, 0) will set L to 0 for points, surfaces and volumes.
member.length	Returns the length of the combined line parent of the line, if there is one, or the length of the line if there isn't.
parent.x	Returns the parametric distance along the parent at which the calculation is being performed.
member.x	Returns the parametric distance along the parent at which the calculation is being performed.

User Defined Results Scopes

Name scoping (an association of a name to an entity, such as a component or variable) can be used within the User Defined Results facility to help clarify the actual meaning of an included component or variable. Name scoping is optional as long as there is no ambiguity.

"Parent"	The feature containing the location being evaluated, offering values such as type and radius.

"material"	The material attribute assigned to the feature containing the location being evaluated, offering values such as E and rho.
"composite"	The composite attribute assigned to the feature containing the location being evaluated, offering values such as layer name and material.
"damping"	The damping attribute assigned to the feature containing the location being evaluated, offering values such as ar and br.
"geometric"	The geometric attribute assigned to the feature containing the location being evaluated, offering values such as A and Iyy.
"age"	The age attribute assigned to the feature containing the location being evaluated, offering the single value age.
"mesh"	The mesh attribute assigned to the feature containing the location being evaluated, offering values such as ratio and size.
"equivalence"	The equivalence attribute assigned to the feature containing the location being evaluated, offering the single value tolerance.
"design"	The design code attribute assigned to the feature containing the location being evaluated, containing values specified by the design code author.
"entity"	The result entity within which this UDR expression appears, offering values such as DX or Fy as appropriate
"loadcase"	The loadcase specified within the UDR expression, offering values such as ldnum.
"global"	The global coordinates of the location being considered, offering the values X, Y and Z.
"scripted"	The scripted result entity.
"displacement"	The results entity displacement, offering values such as DX, DY,DZ and RSLT
"forceMoment"	The results entity "force/moment", offering values such as Fx, My, and SED
"stress"	The results entity stress, offering values such as SX, Sy and Sx(Fx)
"stressTop"	The results entity stress(top), offering values such as SX and

	SY (and similarly for Middle and Bottom)
"strain"	The results entity strain, offering values such as EX and EY.
"strainTop"	The results entity strain(top), offering values such as EX and EY (and similarly for Middle and Bottom.
"Potentials"	Consistent with its structural equivalent.
"Flux"	Consistent with its structural equivalent.
"Gradient"	Consistent with its structural equivalent.
"reaction"	The results entity reaction, offering values such as FX and RSLT

Notes

- The following attribute scopes cannot currently be used since they can be assigned multiple times, and as a result their values cannot be considered to be unique: "loading", "support", "slideline", "constraint", "retained freedom", "activate", "deactivate", "influence", "search area".

- The following results entity scopes cannot currently be used since their names cannot be considered to be unique: "loading", "support", "slideline", "velocity", "acceleration".

Index

3

6

A

D

E

F

facet, 115

failed analysis, 598

Fast Parallel Solvers, 593

fatigue analysis, 629

feature based models, 31

Feature Selection, 747

feature-based models, 31

FiberSIM
    defining composite stack, 412

Fibre directions
    visualisation, 416

Fibre locations, 679
    definition and adding, 216

field analysis, 571

field loads, 381

File
    backups, 81

File and folder naming
    260 character path limit, 84

file locations, 80

file types, 75, 76, 85, 86, 88, 89, 90, 92, 94

find
    by advanced selection, 51
    location of assigned influence
    attribute, 453

Fixing Mesh Problems, 190

FLD loading, 362

fleshing, 221

Floor loading, 363

flow, 351

Flux patch load, 393

foam material model, 321

Fourier expansion, 688

Fourier analysis, 580

Fourier results, 653, 687

frequency analysis, 559

frequency response, 635

Friction Pendulum, 331

Frictional slideline, 418

Functions
    supported, 755

G

general loads, 369

geometric Attributes, 217

Geometric Attributes, 220

geometric nonlinearity, 547

Geometric Properties, 212

geometry, 113

geometry orientation, 162

geostatic control, 559

gnl, 547

graph wizard, 673, 687

graphing, 687, 688

gravity loading, 358

Grid layout, 144

group symbols explained, 58

groups, 55

Guyan reduction analysis, 431, 562

H

HA/HB loading, 369

Hashin failure contours, 656

Hashin material model, 309

Heat of Hydration, 297

heat transfer, 456

Heat transfer, 456

hexahedral, 140

I

J

M

N

O

P

W

Y

Z

LUSAS