Modeller Reference Manual

Version 14.7 Issue 1

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

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

Distributors Worldwide

Table of Contents

Chapter 1 : Introduction

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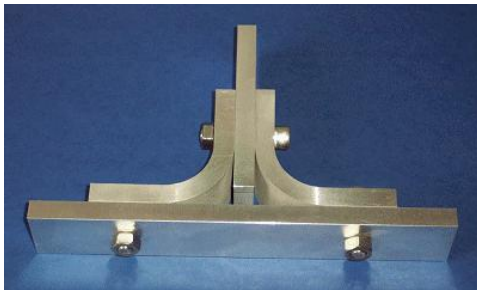
Engineering Problem   **Finite Element Model**
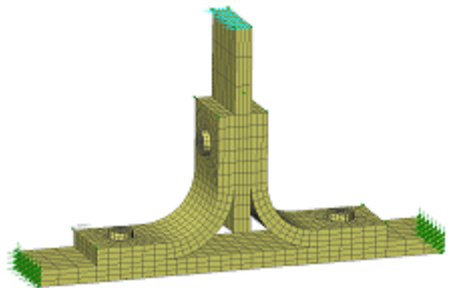
The real engineering problem responds in an infinite number of ways to external forces. The manner in which the Finite Element Model will react is given by the degrees of freedom, which are expressed at the nodes. For example, a three-dimensional solid element has three degrees of freedom at each node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

Finite Element Analysis with LUSAS

Finite element analysis using LUSAS software involves three stages.

1. **Creating the finite element model**
2. **Running the analysis**
3. **Viewing the results**

Each LUSAS software package consists of:

❑ **LUSAS Modeller** a fully interactive graphical user interface for modelling and viewing of results
❑ **LUSAS Solver** a powerful finite element analysis system

Modelling

Modelling involves creating a geometric representation of the structure, assigning attributes and outputting the information as a formatted data file (.dat) suitable for processing by LUSAS Solver. See **Using LUSAS** for a quick introduction to the LUSAS Modeller interface.

Creating a Model

A model is a graphical representation consisting of **Geometry** (Points, Lines, Surfaces and Volumes) and **Attributes** (Materials, Loading, Supports, Mesh, etc.). The model is created by **Defining** the Geometry and Attributes and **Assigning** the Attributes to the Geometry.

Geometry can be defined by entering coordinates, selecting Points on the screen, by using utilities such as transformations or by importing it from a CAD system. Attributes are first defined and then assigned to selected features.

To complete a model it may be necessary to define analysis control data. This is used to control the progress of an advanced analysis.

Running the Analysis

Once a model has been created click on the solve button to begin the analysis. LUSAS creates a data file from the model, solves the **stiffness matrix**, and produces a results file. The results file will contains some or all of the following data:

❑ **Stresses**	❑ **Reactions**	❑ **Strain energy**
❑ **Strains**	❑ **Yield flags**	
❑ **Displacements**	❑ **Potentials**	
❑ **Velocities**	❑ **Fluxes**	
❑ **Accelerations**	❑ **Gradients**	
❑ **Residuals**	❑ **Named variables**	

Viewing the Results

This involves using a selection of tools for viewing the results file produced from the analysis. Many different ways of viewing results are supported:

❑ **Undeformed/Deformed Mesh Plots**
❑ **Contours (Averaged and Unaveraged)**
❑ **Vectors**
❑ **Diagrams (Bending Moments and Forces)**
❑ **Animated Display of Modes/Load Increments**
❑ **Sections (Lines and Slices)**
❑ **Yield Flag Display**
❑ **Graphing**
❑ **Printed Output**

What Help and Documentation is Provided?

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

The documentation includes:

❑ **Installation Guide**
❑ **Getting Started Guide**
❑ **Modeller Reference Manual**
❑ **Examples Manual**
❑ **Application Examples Manual (Bridge, Civil & Structural)**
❑ **Application Manual (Bridge, Civil & Structural)**
❑ **Autoloader Reference Manual**
❑ **IMDPlus User Manual**
❑ **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)**
❑ **Glossary**

Installation Guide

- Details of installing LUSAS software for various licensing options.

- Available in PDF and printed form.

Getting Started Guide

- Contains a brief overview of LUSAS.

- Available in PDF and printed form.

Modeller Reference Manual

- Provides detailed reference material for modelling and results viewing with LUSAS Modeller.

- Provided in on-line help format and also available in PDF and printed form.

Examples Manual

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

- Available in PDF and printed form.

Application Examples Manual (Bridge, Civil & Structural)

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

- Available in PDF and printed form.

Application Manual (Bridge, Civil & Structural)

- Describes the bridge, civil and structural application specific features of LUSAS and their uses.

- Available in PDF and printed form.

Autoloader Reference Manual

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

- Available in PDF form.

IMDPlus User Manual

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

- Provided in on-line help format and also available in PDF and printed form.

Rail Track Analysis User Manual

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

- Available in PDF form.

Element Reference Manual

- Contains full element specifications. This is the place to go to find out which functionality your elements support and what output you will obtain from your element selection.

- Provided in on-line help format and also available in PDF and printed form.

Solver Reference Manual

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

- Available in PDF and printed form.

Theory Manuals (Volume 1 and 2)

- These contain more detailed theoretical information for the more experienced user. They cover topics specific to LUSAS and where appropriate list references to other publications. The topics covered include:

    - Analysis procedures including: linear, nonlinear, dynamics, eigenvalue extraction, modal analysis, all forms of field analysis, fourier analysis and superelement analysis.

    - Geometric nonlinearity.

    - Constitutive material model formulations.

    - Loads and boundary conditions with particular reference to general load types, constraint equations, slidelines and thermal surfaces.

    - More complex post processing calculations, including nodal extrapolation and calculation of Wood Armer reinforcement moments.

    - Element formulation theory.

- Available in PDF and printed form.

Verification Manual

- A manual of LUSAS testcase examples benchmarked against known solutions.

- Available in PDF form only.

CAD Toolkit User Manual

- Describes interfaces to LUSAS involving the use of external pre- and post-processing packages.

- Provided in PDF form only.

LUSAS Programmable Interface (LPI)

- Provides information for application programmers wishing to customise the LUSAS environment or interface LUSAS with other applications.

- Provided in on-line help format only. This can be accessed from the Start > All Programs > LUSAS > LUSAS Help menu.

Glossary

- Contains definitions of general terms used in all manuals.

- Provided in on-line help format. Included in the *Modeller Reference Manual.*

Chapter 2 : Using Modeller

Welcome to LUSAS Modeller

LUSAS Modeller is an easy to use Windows-based finite element modelling system.

Creating a Model

In LUSAS Modeller two types of models can be created:

- ❑ **Feature-based geometry models** - these comprise geometry features (points, lines, surfaces, volumes etc.), that are created either by using a whole range of tools under the Geometry menu, or the buttons on the Toolbars, or by importing third-party data that is supported
- ❑ **Mesh-only models** - these comprise only 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.

To both types of model **attributes**which describe the properties of the model (materials, thicknesses, loading, supports, mesh, etc.) are assigned. Attributes are defined from the Attributes menu. Once defined, attributes are listed in the Attributes ⚛ **Treeview**

Treeview

The Treeview is used to organise various aspects of the model in a graphical frame. It has six panels each with a Treeview showing Layers 🖼 , Groups 🔶, Attributes ⚛, Loadcases 🕒 , Utilities 🔧 and Reports 📖 respectively. The Treeviews use drag and drop functionality. For example, an attribute in the ⚛ Treeview can be assigned to geometry by dragging the attribute onto the previously selected **object**.
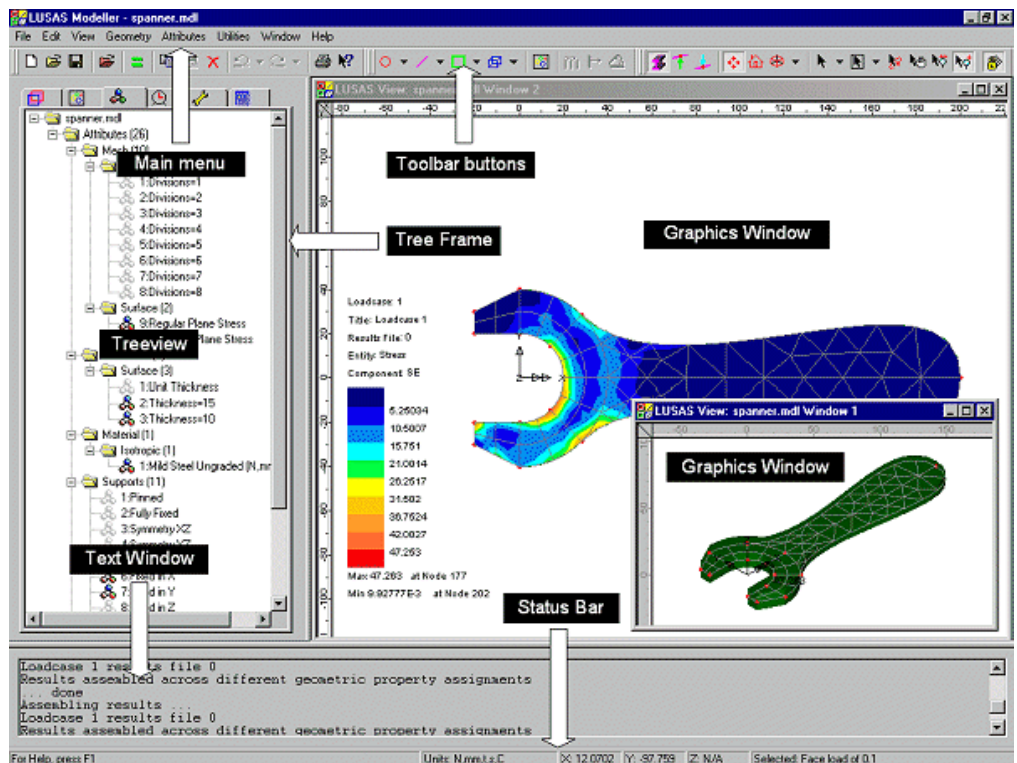
LUSAS Modeller Interface

Context Menu

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

Properties

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

Status Bar

The Status Bar displays progress messages and help text during a modelling session, the model units, the current cursor position in model units (if the model is displayed in an

orthogonal plane) and the item or number of items in the current selection. The **View> Status Bar** menu item may be used to hide or show the Status Bar.

Text Window

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

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

Toolbars

Toolbars contain the toolbar buttons. On initial start-up the Main, Define and View Toolbars are displayed. All toolbars can be shown, hidden, or customised, using the **View> Toolbar** menu item. When a modelling session is completed the current toolbar settings are saved and reloaded the next time Modeller is used.

User defined toolbars and buttons can also be added to the user interface. Actions are assigned to user defined buttons using the scripting language.

Tree Frames

By default a single Tree Frame is displayed with the Layers 🔲, Groups 🔲, Attributes 🔗, Loadcases 🔍, Utilities 🔧 and 📋 Reports Treeviews visible. Multiple Tree Frames can be utilised from the **View> Tree Frame** menu item. Treeviews can then be dragged and dropped between Tree Frames as required.

Browse Selection

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

Browse Selection Memory

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

Browse Cyclable Items

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

Data Tips

Data tips (also sometimes referred to as datatips) provide basic information about whatever is under the cursor. The data tip mechanism is only invoked when the cursor hovers over a model feature within the graphics window. If more than one item is present at the location on the screen, you may use the **Tab** key to cycle between all possible selectable objects, with the data tip updated each time. As additional feedback, if a object is selected, dynamic selection is invoked to highlight each object as it becomes the focus. Thus with two intersecting lines or overlapping surfaces, it is clear which one is being displayed in the data tip.

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

Getting Help

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

❑ The **Help** button on the Main toolbar is used to get context-sensitive help on the LUSAS interface. Click on the **Help** button, then click on any toolbar button or menu item (even when greyed out).

❑ **Help Topics** accessed from the **Help** menu provides access to the Help files. They include the *Modeller Reference Manual,* reference help files such as the *Element Reference Manual* and access to other manuals that are available in PDF format such as the Examples Manuals, Theory Manuals and Solver Reference Manuals.

❑ Every dialog also includes a **Help** button which provides information.

Modeller Licence Selection
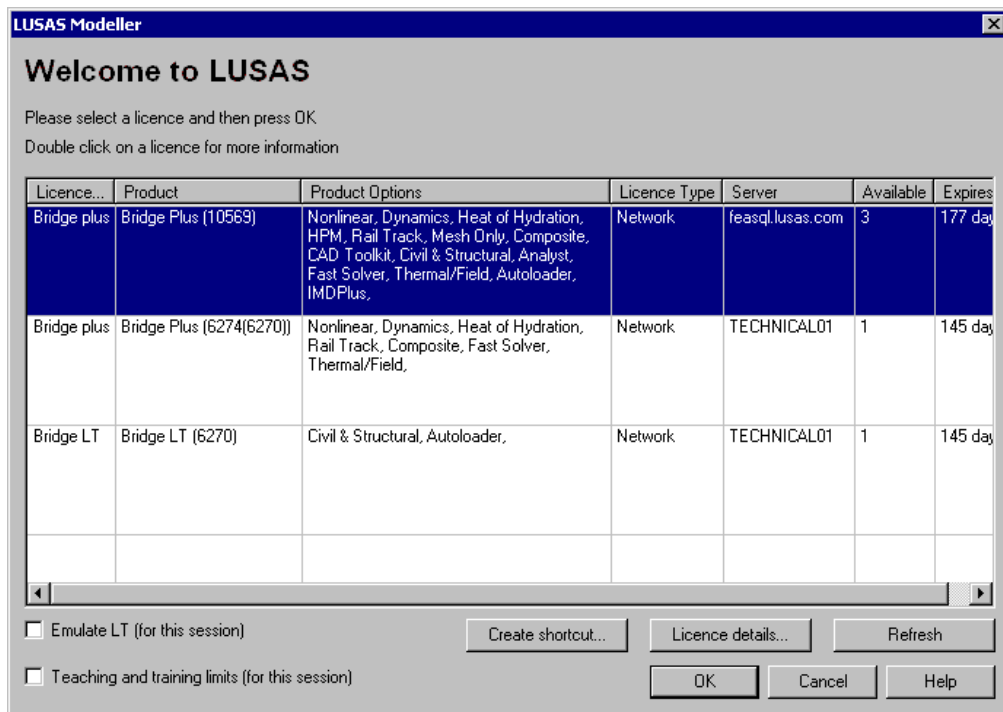
When running LUSAS, the Modeller licence selection dialog lists all software products that are available for selection, along with details of the licenced product options, licence type, server name, number of available licences and the number of days left until expiry. Invalid licences (those that have 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 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 Configuration Utility to add licences to the list of those available.

License order and usage

Standalone or network licences are 'tumbled' based upon Configuration Utility settings. The order of the licence types listed on the Modeller licence selection dialog follows the order of the keys set on the Licensing page of the LUSAS Configuration Utility. If the display of the main licencing dialog has been disabled the first licence in the list of those available will be used in preference to all others. As licences are used the number of licences available for

others to use is updated accordingly. When LUSAS Modeller is run the system will work its way down the list of licences until a valid and available licence is found.

Creating shortcuts

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

  Example of shortcut created:

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

Licence details

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

Refresh

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

Emulating LT Behaviour

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

Teaching and training limits

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

Creating a New Model

The New Model dialog is displayed every time a new model is to be created. The dialog enables the model file to be named and located in a selected project folder and for initial model set-up information to be specified.

The model title, job number, units and designated vertical axis are also defined. The style of user interface can be also set (dependent upon licence key options). This simplifies the user interface and also defines the class of model to be created. The model title and the style of user interface can be subsequently modified on the **General** tab of the **Model Properties** dialog accessed via the **File> Model Properties** menu item but note that changing an interface type will not remove any properties created and assigned to the model when using the previous interface.

Within LUSAS consistent units must be used. A wide range of consistent units is available from the units drop-down list.

By selecting a startup template, useful predefined attributes can be setup in the model. User-defined startup templates can be created and accessed via the button alongside the Startup template drop-down list.

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

Model Types

Two types of model can be created in LUSAS:

❑ **Feature-based geometry models** - these are based on defined features (such as points, lines, surfaces and volumes) or are created from imported geometry and require the definition of mesh objects such as elements with their associated nodes. Prior to version 14.7 of LUSAS these were the only model type available in LUSAS.

❑ **Mesh-only models** - these comprise only mesh objects (elements and their associated nodes, edges and faces) and are created by importing only those types of LUSAS or third party data that are supported. Initial support for these models was introduced in version 14.7 of LUSAS.

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

Attribute assignments to different model types

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

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

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

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

Feature-based geometry models

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

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

Mesh-only models

Mesh-only models are comprised of nodes and elements and do not contain any geometric feature types, or indeed any geometric data at all. They 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 Mesh** 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 user interface option 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.

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.

Notes for mesh-only models

- The **File > Import Mesh** menu item is only available if no Geometry data is present in the Modeller window.

- Mesh-only models cannot contain geometry layer data. As a result, the option to add a Geometry layer is not accessible.

- By default, when element types from non-LUSAS created datafiles are mapped to an equivalent LUSAS elements, structural element types are created. If the Coupled user interface option 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.

Restricted / unsupported functionality for mesh-only models

With reference to the user interface menus for feature-based geometry models:

- The Utilities > Slice resultants beams and shells menu item is not currently supported

- The Utlities > IMDPlus > Moving load wizard functionality is not currently supported

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

- The Bridge > Prestress Wizard > Tendon profile and Multiple tendon prestress menu items are not currently supported.

- The Bridge > Prestress Wizard > Single Tendon design code options are not currently supported.

- Single tendon definition is only permissible by importing data from a spreadsheet.

- Reference paths can only be created currently by typing in the coordinates defining the path.

Model Properties

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

❑ **General**
❑ **Notes**
❑ **Geometry**
❑ **Meshing**
❑ **Attributes**
❑ **Solution**
❑ **Defaults**
❑ **Solver System Variables**

General

- **Title** Brief description of the current model as entered in the New Model startup dialog.
- **User Interface** Enables the user interface to be simplified by specifying the type of analysis model being generated. Options dependent upon licence key is use are Coupled, Structural, Thermal. Associated main menu items are removed if licence key does not include a thermal option. Note that changing an interface type will not remove any properties created when using the previous interface.
- **Auto backup** Saves the model automatically as it is being developed or modified.
- **Precision shown in dialogs** Controls the number of significant figures or decimal places displayed in the dialogs.

- **Pens** Sets the Pens that are used to draw Points, Lines, Surfaces and Volumes.
- Click on the **Choose Pen** button to use a different pen, or to modify the Pen Library.
- Changing a Pen allocation can be applied to existing features and/or new features using the two check boxes.
- **Units** Specifies the modelling units.

Notes

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

Geometry

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

❑ **Merge Options**

- **Action** controls the criteria that must be satisfied before features sharing a common definition will be merged.

- **New geometry unmergable** sets the merge status of all new features to Unmergable rather than the default which is Mergable.

- **Tolerance** controls the distance within which Point features must lie before they will be considered for merging. Note: The merge tolerance should only be changed with extreme caution because changing it from its default value can lead to instability of the underlying geometry engine.

See **Merging Features** for more details.

❑ **Active Local Coordinate**

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

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

Advanced Geometric Properties

❑ **Splitting Defaults**

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

❑ **Creation Defaults**

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

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

❑ **Hole Removal Defaults**

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

❑ **Merge Defaults**

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

❑ **Drawing defaults (faceting)**

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

Meshing

❑ **Equivalence** Defines the default nodal equivalence tolerance used in a equivalence attribute. If **automatic** is switched on equivalencing is carried out automatically for all nodes in the model and no equivalence attribute assignment is required. See **Nodal Equivalencing** for details.

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

❑ **Irregular tet meshing** specifies the number of the passes and attempts to be made by the tetrahedral mesh generation when attempting to mesh a volume.

❑ **Create a group of objects that failed to mesh** creates a group named **$failedToMeshObjects** which contains all features which failed to mesh.

Advanced meshing parameters

❑ **Draw failed parts of mesh only** - draws those parts of the mesh that failed to mesh

❑ **Linearise element edges**

❑ **Constrain adjacent linear/quadratic edges** forces mid-side nodes on lines or surfaces to be averaged between the corner/end node positions. This option is turned on by default.

❑ **Element edge collapsing** invokes **edge collapsing** which removes elements/faces with short edges and small subtended angles by merging them with neighbouring elements.

Attributes

❑ **General Options**

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

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

❑ **Slideline Options**

- **Suppress stringent slave search** For simpler geometries, such as flat surfaces in contact, a slight reduction in processing time may be achieved by suppressing the "stringent" local node search but this is not usually recommended. (LUSAS Solver option 184). See the *Theory Manual* for more details.

- **Suppress initial slide-surface stiffness check** Slideline stiffnesses are automatically scaled at the beginning of an analysis if the average master/slave stiffnesses differ by a factor greater than 100. This is to account for contact between bodies that have significantly different material properties. See the *Theory Manual* for more details.

- **Suppress initial penetration check** The coordinates of all contact nodes that have penetrated prior to the commencement of an analysis, are reset back to the closest point on the contacted surface. This option should be set if the node resetting is to be suppressed, such as when performing an interference fit analysis. See the *Theory Manual* for more details.

Solution

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**.

❑ **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**.

❑ **Element Options**

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

- **Assign 6 degrees of freedom to all thick shell element nodes** By default this option is on. It has the effect of adding a rotational spring stiffness to the drilling rotation of the thick shell elements making the analysis more stable. The value of the spring stiffness can be adjusted using the system parameter STFINP. For problems in which geometric nonlinearity (Option 87) is being used more accurate results may be obtained by switching this option off and letting LUSAS automatically establish the need for 5 or 6 degrees of freedom at a node. (LUSAS Solver option 278).

- **Axisymmetry about Global X axis** When selected, LUSAS considers the line of axisymmetry in an analysis to be about the global X axis and not the default, which is the global Y axis. (LUSAS Solver option 47).

- **Lumped Mass Matrix** Formulate lumped mass matrix instead of consistent mass matrix for elements. (LUSAS Solver option 105).

- **Write strains to output file** causes element strains to be written to the Solver output file.

- **Preserve loading whilst elements deactivated** retains the loading assigned to all elements in model (activated and deactivated) until a subsequent load case is applied. Typical usage would, for example, be when carrying out a staged construction analysis. Self weight would be assigned to all elements (activated and deactivated) in the loadcase representing stage 1 of the analysis. As stage 2 and subsequent loadcases are activated the loading initially applied to elements in stage 1 would automatically be applied as the elements become active. Note that when using this option no additional loading should be applied to any loadcases following the one that contained the load

assignments. If done so, the initial applied loading for those loadcases will be lost.

❑ **Integration options**

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

- **Fine integration for mass HX16 and HX20** Formulate mass matrix with fine integration. (LUSAS Solver option 91).

- **Coarse integration for semi-loof shells** Invokes coarse numerical integration rule for semiloof elements. This option under-integrates the semi-loof shell element which may have the effect of removing low energy mechanisms when the element is very thin and/or pressure loaded. (LUSAS Solver option 19).

- **Newton-Cotes Integration for beam elements** Newton-Cotes is a form of numerical integration or quadrature. It is often used for through-thickness integrals since sampling points are located at the extremes of the range. (LUSAS Solver option 134).

❑ **Nonlinear Options**

- These options define the type of geometric nonlinearity to be used in the analysis. The default is for no geometric nonlinearity. See Geometric Nonlinearity. Consult the *Element Reference Manual* to check which geometric nonlinearity type is supported for selected elements.

Geometric nonlinearity

- **Total Lagrangian** A strain formulation that has its reference as the initial configuration at the start of the analysis. (LUSAS Solver option 87).

- **Updated Lagrangian** A strain formulation that has its reference as the end of the last converged increment. (LUSAS Solver option 54).

- **Eulerian** A strain formulation that has its reference as the current configuration. (LUSAS Solver option 167).

- **Co-rotational** Form of geometric nonlinearity in which large displacement effects are related to a set of axes that follow and rotate with the element. (LUSAS Solver option 229).

Solution control

Allows fine control over advanced nonlinear solution procedures.

- **Continue solution after convergence fails** This option continues with a nonlinear analysis even after an increment has failed to converge. It is useful for creating a results dump of an unconverged results file to visualise problem areas. This option should be used with care. (LUSAS Solver option 16).

- **Continue solution if more than one negative pivot occurs** will step over the LUSAS error that stops the analysis if more than one negative pivot is encountered. This can be useful if pivot problems are encountered at an early stage but the problem is free from them during later stages. This option should be used with care as it is likely to hide more fundamental analysis problems. (LUSAS Solver option 62).

- **Non-symmetric solution** (LUSAS Solver option 64). This is set automatically by LUSAS Modeller when, for example, carrying out a nonlinear concrete material modelling of cracking.

❑ **Coupling Options**

These allow control of a coupled analysis.

- **Coupling type** defines which analysis to run first, Thermal or Structural.

- **Parallel coupling** requires both analyses to run simultaneously such that the temperatures from the thermal analysis are read into the structural analysis and the displacements from the structural analysis into the thermal analysis. If the temperatures are to be calculated in a thermal analysis and then transferred to a structural analysis this option is not required.

- **Initialise reference temperatures** takes the first temperature distribution from the thermal analysis and uses them as the reference temperatures in the structural analysis.

- **Suppress recalculation of view factors in coupled analysis** Turns on/off the view factor recalculation. The option should be turned on when the radiation surface geometry is unchanged by the structural analysis to suppress the re-calculation of view factors. (LUSAS Solver option 256).

- **Step coupling** is used for coupling thermal to structural analysis such that the nonlinear increment is used to control the coupling steps.

- **Time coupling** is used for coupling thermal to structural/transient analysis such that the time is used to control the coupling steps.

- **First data read** and **first data write** are the time/increment to read/write the first set of data.

❑ **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.

Defaults

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

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

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

Solver System Variables

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

Using Windows

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

Windows consist of the following components:

❑ **Drawing Layers** By default the first panel of the tree frame displays each model window currently open, and shows the view **layers** contained in that window.

❑ **Window Properties** Double-clicking in a window or right-clicking in a window with no selection, will display the properties of that window. The **Window Properties** can be altered as required.

Saving a View

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

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

❑ **Layers** The window layers contained in the Treeview.

❑ **Colours** The colourmap as used for plotting results contours.

❑ **Page size & borders** The page size and border setting as defined from the File> Page Setup menu item.

❑ **Pen Library** The pen library referenced every time a item is drawn.

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

Pen Library

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

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

Loading a View

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

Copying Windows

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

Using Layers

Model information is held in separate pre-named layers to aid selective viewing of both model and results data. Layers can be added or removed from a view window by right-clicking on a blank part of the graphics area, or by right-clicking the window name in the 🔲 Treeview and selecting **Properties** from the context menu, or by selecting the **View> Insert Layer** menu item.

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. Turning a layer off retains the layer in the Layers 🔲 Treeview but removes the display of that layer from the Graphics Area.

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.

Note. 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.

Layers available

The following layers can be added /removed, have their display turned on/off and be generally manipulated from a view window:

- ❏ **Geometry**
- ❏ **Mesh**
- ❏ **Attributes**

❑ **Labels**
❑ **Annotation**
❑ **Utilities**
❑ **Contours**
❑ **Vectors**
❑ **Deformed mesh**
❑ **Diagrams**
❑ **Values**

Note. The order in which the layers are drawn is defined by the order or the layers in the ▣ Treeview. The top layer in the Treeview is drawn first to the screen. 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 ▣ Treeview by selecting it, then dragging and dropping it to a new position to cause it to be drawn following a particular layer.

Selecting Model Features

Items displayed in the graphics window may be selected with the cursor by clicking on them individually or by dragging over an area. 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..

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.

> 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.

Notes

- 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 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.

- Items in the 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 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.

Selection Filters

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.

Feature-based models

❑ Normally the 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
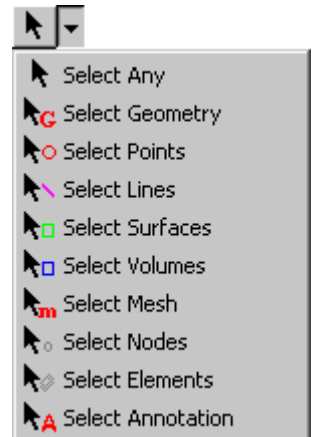
S – Surface selection filter

V – Volume selection filter

M – Mesh selection filter

A – Annotation 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

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:
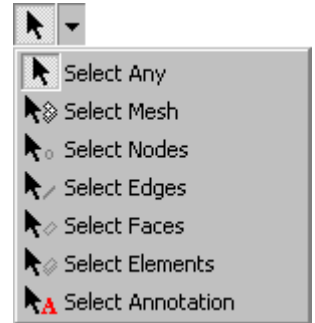
M – Mesh selection filter

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 the selection keys available.

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

Cycling through the Selection

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

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

1. Hovering the cursor over a feature displays a data tip. This shows the feature that will be initially selected with a cursor click. Other items at the location are shown.

Each click of the mouse or press of the TAB key will select the next feature available.

Surface 1
Lines : 1, 2, 3, 4
Msh:Regular Plane Stress
Gmp:Thickness=15
Mat:Mild Steel Ungraded (N,mm,t,s,C)

There are 3 items at this position
Press Tab to cycle through them

Browse Selection...
Browse Cyclable Items...
Browse Selection Memory...
Browse Visible Items...
Next
Previous
Closest
Furthest

X: -12.253   Y: 32.2875   Z: N/A   Selected: Surface 1 (3 items cyclable)

2. The currently selected feature is also displayed in this area of the Status Bar. A right-click here displays a context menu allowing even greater control over the selection of items.

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

Associate Selection Downwards

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

Associate Operations Downwards

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

Selection Memory

Some operations require features to be stored in selection memory. 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 Window properties. To display the Window properties choose properties from the window context menu enabled by right-clicking in the graphics area.

Groups

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

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

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

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

Uses of Groups

❏ Enabling unique components to be identified, manipulated, hidden, or have results plotted only on those features.

❏ Allowing the assignment of attributes to a group in one step. The appropriate geometry features will be used if attributes can not be assigned to all geometry types.

❏ Identifying all the features that failed to mesh during any command that invokes meshing. See **Fixing Meshing Problems**

❏ When defining slice sections through a model in order to view the internal arrangement or to plot results.

❏ To allow easy manipulation of master and slave results following a slideline analysis.

Defining Groups

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

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

Naming Groups

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

Manipulating Groups

Groups are listed in the 🔲 Treeview. From a group's context menu the following commands will act on that group in the current window:.

- ❑ **Visible** Makes all members of the group visible.
- ❑ **Invisible** Makes all members of the group invisible.
- ❑ **Set as Only Visible** Makes the whole model invisible and then sets all the members of the group to be visible.
- ❑ **Advanced Visibility** Enables the visibility of the higher and lower order features of the members of a group to be manipulated.
- ❑ **Results Plots** permits results to be selectively plotted for the chosen group. For details see **plotting results for groups** in the results viewing section.
- ❑ **Change element type** (mesh-only models) permits changing the element type of a group of imported elements by description, or by entering a specific known element name.
- ❑ **Current Group** Set the group to be the **current group**.
- ❑ **Rename** Enables a group to renamed.
- ❑ **Delete** Deletes the group. The group is deleted but the contents of the group are not.
- ❑ **Select Members** Adds the members of the group to the current selection.
- ❑ **Deselect Members** Removes the members of the group from the current selection.
- ❑ **Add to Group** Adds the currently selected items to the group.
- ❑ **Remove from Group** Removes the currently selected items from the group.
- ❑ **Properties** Displays a list of the member items of the group.
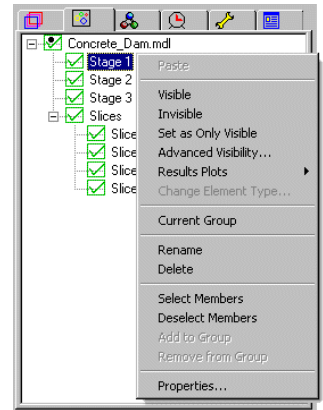
When the group defines a slice section the otherwise generally available Rename, Delete, Add to group, and Remove from group options are unavailable but additional slice-related options are include on the context menu:

❑ **Draw axes** Draws the local axes for the slice section. Note that options on the Contours properties dialog allow for plotting of results on slice sections and at slice axis directions.

❑ **Print Properties** Displays information about the slice including cross-sectional area of the slice, centroid and section property data.

❑ **Print Forces** Displays force and moment information for the slice.
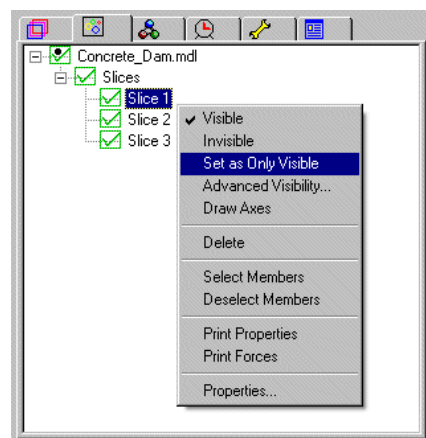
Group symbols explained

A symbol adjacent to each group name in the 🔯 Treeview shows the visibility and status of each group.

When modelling:

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

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

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

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

When a results file is loaded:

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

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

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

Changing the Visibility of Features

By default all model geometry is visible (providing the geometry layer is present and turned 'on' in the 🗔 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 graphics 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 graphics 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 graphics window context menu allows fine control on the visible / invisible items by controlling the visibility of higher order and lower order features. i.e. This allows all Lines attached to a Point to made visible or all Surfaces connected to a Line to be made invisible.

Notes:

• All visible features will always have their lower order features visible. i.e. If a line is visible its defining points will also be visible.

• When a feature is made visible any associated elements or nodes will also be made visible. Use the Advanced Visibility dialog to override this behaviour.

• An element can only be made invisible if its defining feature is invisible. To make a chosen element invisible, select the element and then use the Advanced Visibility dialog to make the element invisible by selecting the **Also apply to higher order** option. This makes the selected elements and the defining features invisible without making the unselected elements invisible.

Rotating, Zooming and Panning

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

❑ **Dynamic rotation** is carried out by selecting the dynamic rotate button, or by pressing the scroll wheel button and either the left or right mouse button or by holding down the **R** key, while moving the mouse in normal cursor mode. The model will be rotated about the centre of the model unless any part of the model is selected in which case the model will rotate about the centre of the selection. The model can additionally be rotated around any of the screen axes by pressing additional keys. See **Rotating the Model**.

❑ **Dynamic zoom** is carried out by selecting the dynamic zoom button, or by scrolling the mouse wheel or by holding down the **Z** key while moving the cursor in normal cursor more. If any part of the model is selected the location of the centre of the selection will remain fixed.

❑ **Dynamic pan (drag)** is carried out by selecting the pan button, or by depressing the scroll wheel button or holding down the **D** key while moving the mouse in normal cursor mode.

Notes:

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

Rotating the Model

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

❑ **Dynamic rotate** Allows the model to be rotated dynamically using the cursor. The model rotates around various multiple axes when the cursor is moved. By holding down the **Control** key, the **Control** and **Shift** keys or the **Shift** key while using the dynamic rotate the model can be rotated independently about the screen Z, Y and X axis respectively. Click on the normal cursor to return to selection mode. Note also that holding down the **Alt** key provides a one-key option for rotating about the the screen Y axis.
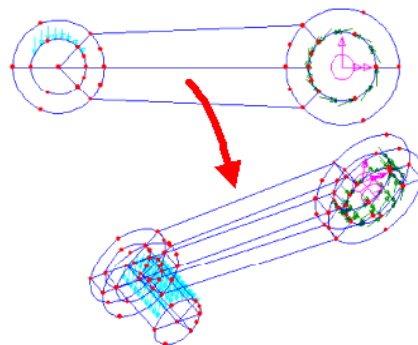
Note. Models are dynamically rotated using the 'model ball' method. With this method, a model can be imagined to be surrounded by a sphere such that a mouse-click and a drag of the cursor on the screen represents clicking on the surface of the sphere and dragging to rotate it to a new position. In doing so, it is important to note that the model rotation is restrained to rotate only around the model's vertical axis (as defined on the Vertical Axis dialog) - unless any **model viewing shortcuts** are being used at the same time. A benefit of this approach is that, no matter where on the screen you click to start the rotation of your model, if you return the mouse pointer to the same spot, (whilst you are dynamically rotating the model), the model will return to its original position and orientation on the screen.

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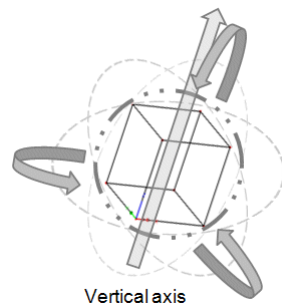
❑ **Incremental rotate** Allows the model to be rotated by an specified rotation about a chosen global axis. The specified rotation may be modified by adjusting the rotation increment on the window's properties.

- Anti-clockwise Y
- Clockwise Y
- Anti-clockwise X
- Clockwise X
- Anti-clockwise Z
- Clockwise Z
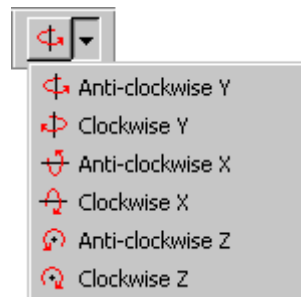
❑ **View along axis** Views the model along a chosen global axis.

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.

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

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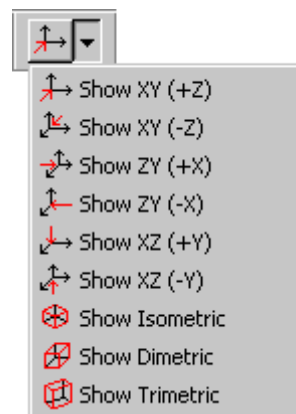
Zooming in or out

Zoom tool This tool works in two ways:

1. By dragging a box around part of the model the view will zoom into that area.

2. Clicking on part of the model will zoom in progressively for each click. Hold the **Control** key to zoom out.

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

Panning (Dragging) the Model

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

Reseting the View

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

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

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

Undo/Redo

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

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

Notes:

- The undo facility works by replaying the session file from the last save. Because of this, it is advisable to save the model frequently to speed-up the undo facility.

- Undo is only available when a model file is loaded. (i.e. undo is not available when a results file is loaded without a corresponding model file)

Page Layout Mode

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

❑ **Working Mode** is useful for model generation. In working mode annotation is scaled and moved so it is always visible.

❑ **Page Layout Mode** enables the model to be viewed as it would appear on a printed page. This makes annotation easier to position and allows pictures to be created to a specified scale.

Scaling

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

window ![button] 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 **Window Properties** accessed from the context menu. In this dialog the **scale to fit window** option should be switched off and the **scale** and **origin position** defined.

Annotating the Model

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

❑ **Window properties** contain options relating to the current window.

❑ **Startup templates** can be used to pre-load the Attributes Treeview of the interface with selected attributes for a particular analysis, set default mesh or material types, or define preferred colour schemes - to name just a few uses.

❑ **Toolbars and toolbar buttons** can be customised and user-defined toolbar buttons can added to the user-interface either to sit on a new toolbar group or alongside existing buttons in an existing group.

Window Properties

The Window Properties dialog shows options relating to the current window. Window properties may be displayed by double-clicking in a blank part of the graphics area (with no features selected), or by right-clicking a window in the 🔲 Treeview and then selecting Properties from the context menu. Window properties define basic view information.

❑ **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. Provides isometric projection. Enables the scale and origin position to be set and allows the current view settings to be saved.

❑ **View axes** Controls whether and at what position to visualise the coordinate system and provide control of the style.

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.

Colours

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** Option set by the resize button to ensure the model fits the screen area.

❑ **Scale** Option enables the model to be scaled

❑ **Origin position** Defines the origin of the model

❑ **Rotation Vector** An equivalent eye position coordinate. Entering (0, 0, 1) views the model from the Z axis, and (1, 2, 3) gives a three-dimensional view.

❑ **Rotation increment** Sets the rotation increment used for incremental rotation.

❑ **Triangle sort** Defines the triangular sort algorithm to use when shading (GDI drivers only)

❑ **Save View** Saves the current view, including the window properties, pen library, colour map and window layers. When a view is loaded into a window you have the choice of what to reload e.g. colours, layers, etc.

View Axes

❑ **Visualise coordinate system** Visualises the active coordinate system in the graphics area using the position and style specified. The active coordinate system may be set from the **Geometry** tab of the model properties dialog.

❑ **Anchor point** Anchors the coordinate axes 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.

Customise Startup Templates

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

Case Study. Creating a Startup Template

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

Case Study. Using the Startup Template

1. Run LUSAS and on the New Model dialog select the ⬜ button adjacent to the Startup template drop-down list.

2. On the Customise Startup Template dialog select the ⬜ button adjacent to the Script field. Enter **My defaults** into the Name field. Press the **Add** button to add the script and name to the table. Click **OK**.

3. On the New Model dialog, select **My defaults** from the Startup template drop-down list and click **OK**.

4. The startup template script will run setting all values to those previously chosen.

Toolbars and Toolbar buttons

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

Turning toolbar groups on and off

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

❑ Show Tooltips shows a temporary description of the toolbar button when the cursor is moved over the button.

❑ Cool Look removes the raised button style to leave a 'flat' button.

❑ Large Buttons are not implemented in LUSAS modeller

Creating a new toolbar group

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

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

Customising toolbar groups

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

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

Creating user-defined toolbar button actions

Nine user-definable toolbar buttons are provided for linking to a specified action. With the Customise tab selected these can be seen if the User entry is selected in the Categories list. Scripts can be recorded and specified to be played when a particular user-defined toolbar button is selected.

- In Modeller, use **File > Script > Start Recording** to record an action to be taken (such as the adding a Contours layer to the Treeview and the selecting a particular entity such as Force / Moment, and a component such as Mz, for example). Then use **File > Script Stop Recording** to save the script with a name such as Bending.vbs to a folder.
- Then use the **User** tab to define the action that a particular user-defined toolbar button should take when pressed. This involves inserting a text string to reference the script that was recorded. A typical entry would read: `Fileopen "C:\LUSAS144\Projects\Bending.vbs"`
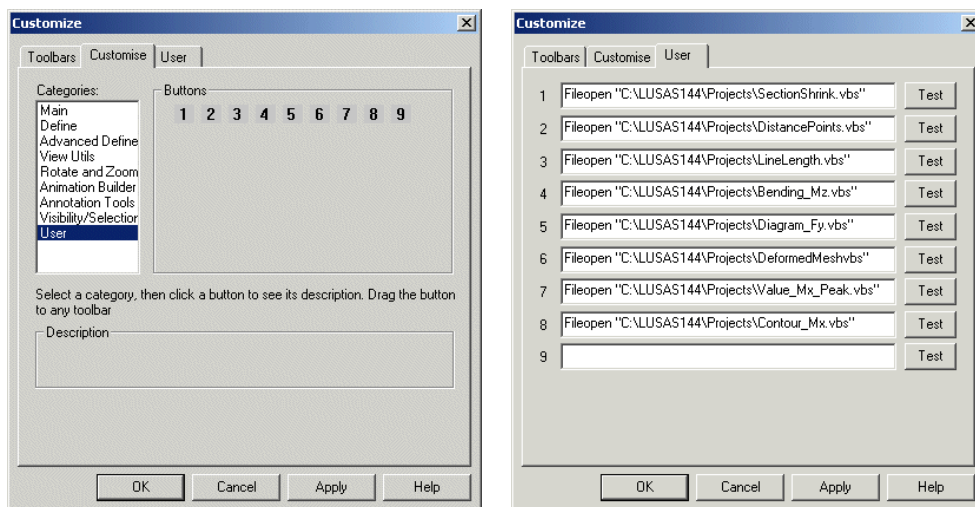
Changing the user-defined toolbar button images

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

User toolbar button bitmap image as supplied   **Example of user-defined toolbar button bitmap image**

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

Chapter 3 : File Types

LUSAS File Types

LUSAS uses a significant number of different file types for a varied range of purposes. The file types covered are summarised below.

- ❑ **Model Files** (.mdl) are created by LUSAS Modeller and are used to store all model definition information.
- ❑ **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 referred to as plot files.
- ❑ **Modeller Results Files** (.mrs) are created by LUSAS Modeller and are used to store the results cache when the model is saved. These files save assembled results and speed up the results processing of combinations. If necessary mrs files may be deleted to save disk space.
- ❑ **History Files**(.his) are created by LUSAS Solver and contain specified analysis results for access by LUSAS Modeller.
- ❑ **Script Files** (.vbs) contain a collection of LUSAS Modeller commands so that, when they are replayed, a sequence of operations may be carried out automatically. Script files can be recorded by LUSAS Modeller or edited directly using a text editor.
- ❑ **Session Files** (.ses) are created automatically every time LUSAS Modeller is run. They contain a record of all commands issued during a session.
- ❑ **Interface Files** (.dxf, .igs, .stp, .stl, .def, .nf) allow graphical structural information to be exchanged between LUSAS Modeller and external packages.

❑ **Command Files** (*.cmd) are used to import and export models from and to version 13 of LUSAS Modeller where only geometry features supported by version 13 Modeller will be exported.

❑ **Picture Files** (.pic, .bmp, .jpg, .wmf) allow 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 LUSAS picture files, screen content can be saved in BMP, JPG or WMF file formats.

Tip. 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 regarding the 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, LUSAS Modeller prompts on exit as a reminder to save changes to a model file.

❑ **New** accessed from the **File> New** menu item, prompts to close an existing model file, and creates a new model file.

❑ **Save** or **Save As** accessed from the **File** menu, saves the current model to disk at any time. **Save As** allows specification of an alternative filename.

❑ **Open** Previously saved model files may be opened by choosing the required model file using the File Open dialog. Modeller will prompt for confirmation before the currently loaded model is closed.

❑ **Close** Closes the currently open file.

Notes

- When using version 14 of Modeller, old version 13 models may be read but this may take longer than usual as the files are converted.

- 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.

Analysis Data Files

In order to perform an analysis, the model must be tabulated into a LUSAS Solver data file. The Solver data file has a **.dat** extension.

Writing the LUSAS data file is controlled using the **File> LUSAS Datafile** menu item. This produces a data file in readable ASCII text format. If necessary, the file may be

modified with a standard text editor. The format of the analysis data file is described fully in the *Solver Reference Manual*

The **File> LUSAS Datafile** menu item allows the following options to be set in order to control the analysis process from within LUSAS Modeller:

Process

This controls the parts of the model for which data is tabulated.

❑ **All** items (default)
❑ **Visible Items**

Solve Now

If this is set LUSAS Solver will run immediately after the data file is tabulated (default). If this is not set a data file is tabulated but not solved.

When the **Solve now** option is set the **Options** button is enabled. This displays a dialog which allows the following parameters to be set.

❑ **Wait for solution** - If set Modeller cannot be used while the solution is progressing (default).
❑ **Load results** - If set Modeller automatically loads the results file over the model file when the solution has successfully completed. (default).
❑ **Load output file** - If set Modeller loads the output file created during the solution process.

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 **System Variables** tab. Modified parameters will be tabulated in a SYSTEM chapter at the start of the LUSAS Solver data file.

Advanced Solution Options

The analysis is controlled from the Advanced Solution Options dialog activated using the **Advanced** button.

Datafile Type

❑ **General analysis** should be used for all analysis types except influence line analyses (default).

❑ **Influence Lines** produces a data file as above with additional files describing influence line and search area information.

Type

❑ **Structural** Carries out a structural analysis.

❑ **Thermal** Carries out a thermal/field analysis.

❑ **Coupled** - Carries out a coupled analysis.

Controlling Content of LUSAS Solver Output File

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 the **Selection Memory**. The following options allow the results written to the output file to be specified.

❑ **Element** results such as stress and strain (as controlled by the LUSAS Solver options set from the **File> Model Properties** menu item) can be written to the output file at Node and/or Gauss points and also written to a history file if required.

❑ **Node** displacements and reactions can be written to the output file and a history file can also be written if required.

❑ **Generate plot file** If selected configures the LUSAS Solver data file to create a plot file (mys). (default)

❑ **Generate restart file** If selected configures the LUSAS Solver data file to create a restart file (rst).

Solver Output Files

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

Solver Results Files

When an analysis is performed by LUSAS Solver it will create a LUSAS results file. The LUSAS results file, or plot file as it is sometimes referred to, has a **.mys** extension.

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.

The information in a Solver results file is stored in a compressed 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.

To access results from the LUSAS analysis, results files can be opened in a similar way to **model files**.

Modeller Results Files

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

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

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

History Files

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

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

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

Script Files

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

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

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

❑ 📂 **Run Script** An existing script file is replayed by choosing it from the **Open** dialog.

❑ **Start Recording** creates a script file. If a non-default file extension is specified or if the file already exists you will be prompted for confirmation before proceeding. Existing script files can be appended to if required. While recording all attempted commands are logged to the script file using the LUSAS scripting language.

❑ **Stop Recording** closes the script file.

Session and Recovery Files

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

Picture Files

About Picture Files

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

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

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

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

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

Saving Picture Files

- Pictures may be saved using the **File> Picture Save** menu item.

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

Print Files

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

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

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

Notes

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

Interface Files

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

The currently supported list of interface file formats is:

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

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

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

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

❑ **LMS CADA-X** (**.nf**) Model description and modal data exported to a file that can be read by the LMS software.

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

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

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

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

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

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

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

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

Summary of Interface File Import / Export Capability

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

Notes

- See Importing Geometry Data for details of how to import interface files.

- See Importing Mesh Data for details of how to import finite element data files created either by the prior running of an analysis in LUSAS or by importing interface files from other supported third-party software applications

- DXF, IGES and STEP files often contain much more detailed information than is required to create a finite element model, so a certain amount of model tidying should be expected after carrying out an import.

File Import

Geometry data from 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.

Mesh data from supported interface files can be imported using the **File> Import Mesh...** menu item.

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
Coalese volumes	Removes similar surfaces from adjoining volumes to simplify the model.	True
Create material groups	Create named groups for features in the data file having the same material property	False
Maximum number of groups	Maximum number of groups permitted to be created from material property types	500
Merge geometry post import	Merges geometry within the general specified merge tolerance.	True
Minimum line length	Facets containing lines of less than this specified length will be ignored.	False
Minimum angle degrees	Facets containing lines of less than this specified length will be ignored.	False
Pre-translation scale	Scaling factor applied to all entities before translation.	1.0
Radius of curvature to length ratio	Minimum allowable radius of curvature to line length ratio in surface trimming.	1%
Parsing error limit	Import terminate after specified number of error (0 indicates no limit).	2
Entity types to exclude	List of entity type numbers to ignore (if checked), entities of these type numbers will not be translated unless they define entities that are to be translated.	Ignore 106 (copious data) and 108 (plane surface)

Drawing layers to process	Allows selection of named layers when importing DXF or IGES data.	None

Importing 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. Use the **File > Import Mesh** menu item to do this. When a file is selected the **Advanced** button can be used to specify import parameters.

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. See **Mesh-only models** for more information.

File Import Mesh options (Advanced)

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

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

Exporting Model Data

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

Depending upon the export file format chosen the following export options may be available:

❑ **Current window**
❑ **All**
❑ **Visible**

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**

DXF Interface Files

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

DXF Import

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

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

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

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

Notes

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

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

- Closed surfaces are not translated by LUSAS.

- Closed polylines and three-sided polygon meshes are not translated.

DXF Export

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

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

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

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

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

Notes

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

- Only the element faces are exported when exporting volume feature mesh records

- For further information on the DXF file format, users are referred to the AutoCAD © Reference Manual.

IGES Import / Export

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

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

Notes:

- IGES data is made up of a number of discrete surfaces. These need to be merged together to create volumes which can then be meshed.

- When IGES import is used the option to crate hollow volumes is automatically invoked.

- Any active local coordinates will be ignored.

- All IGES annotation lines and font data is ignored.

- The IGES interface only supports fixed length ASCII IGES files.

- All curve and surface geometric type entities are translated into LUSAS Modeller.

Supported IGES Entities:

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

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

LMS CADA-X Files

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

❑ When exporting modal data or element matrices, a check for a results file is made. If no results are available, the `moddat` parameter is deselected. The modes for export dialog is only displayed if modal data is requested.

❑ The mode shapes input dialog will expect the mode shape numbers to be entered in the same manner as the SET MODAL MODES command.
The format is: *resultfileID:mode1;mode2;mode-a**T**mode-b**I**increment* ... etc.

❑ Additional system parameters are required to deal with problems encountered when reading the neutral file into LMS. These variables should be set as shown below in the Modeller start-up file. The variables are defined as follows:

❑ **LMSTKV** when set forces the THICKV element property keyword to accept the #NO_DEF purpose code to describe the property type instead of MEMBRANE, PLANESTRAIN, SHELL, PLATE and SHEAR. This is due to an error in the LMS parser.
LMSTKV should be set to 1 to have #NO_DEF output to the neutral file.
LMSTKV should be unset to have normal codes output to the neutral file.
The default setting is 0.

❑ **LMSDMP** forces the export routines to only output SPRING and MASS elements and properties for joints instead of SPRING, MASS and DAMPER elements and properties. This is to work around a limitation in the LMS parser which will not interpret the DAMP or DAMPER keywords.
LMSDMP should be set to 3 to have 3 properties/elements per joint output.
LMSDMP should not be set to have 2 properties/elements per joint output.
The default setting of LMSDMP is 2 (2 properties/elements per joint).

❑ **LMSPRC** allows the user to specify single or double precision real number output in the neutral file. Using this parameter can reduce the size of the neutral file. The differences between the two types of precision is as follows:

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

LMS Export

❑ **Option 290** must be set before tabulating a model to ensure element matrices are transferred from LUSAS.

❑ **Option 290** allows you to instruct LUSAS to output the element STIFFNESS and MASS matrices to the .mys plot file. The volume of data transferred can be substantial so this option is turned off by default.

❑ **Nodal Freedoms** LMS supports only six degrees of freedom: X, Y, Z, Rx, Ry and Rz. If unsupported freedoms are encountered a warning message is issued. If this section

is not present it will **not** affect the translation of the neutral file into LMS as long as no node statement contains a reference to a #Frnnn freedom label.

❑ **Node Coordinates** The node co-ordinates are written with the co-ordinate system omitted implying the use of the global co-ordinate system. The Modeller node labelling scheme is preserved during the export process.

❑ **Material Properties (mdl file)** Isotropic, 2D Anisotropic and 2D Orthotropic materials are supported. 3D material properties are exported as ANISO3D, however the values are read but not used by LMS. Joint properties are output in the element property section of the neutral file.

❑ **Material Properties (mys file)** Material properties are transferred from LUSAS to Modeller. If no material properties are detected in the LUSAS mys file, then dummy material properties are set-up and a warning is issued. Materials can be used in LMS to group common elements together. The dummy properties allow the other model description entities (element topology) to be read by the LMS parser.

❑ **Element Properties (mdl file)** The LMS element properties supported are PBAR, STIFF, PMASS, BEAMG and THICKV. The corresponding Modeller geometric properties are mapped into the expected LMS format. Supported Modeller property types are as follows: Bar/Link, Beam, Membrane/Plate/Shell. Beam and joint eccentricities are ignored. Checks for unsupported element properties are made and a warning is issued if any are found. Unsupported element properties are not output.

❑ **Element Properties (mys file)** Modeller geometric properties from the mys file are transferred and output into the LMS neutral file. The same constraints as for the mdl file apply.

❑ **Element Topology** Elements are output in element type order. Material properties must be specified for all elements, but LMS element properties for solid elements are optional. Checks for unsupported elements are made and a warning is issued with the unsupported elements not written to the neutral file. Beam elements which have end freedoms released are output to the neutral file, but the node freedoms are not transferred as they may not be valid for all connections to a node. Supported elements are shown in the table below.

❑ **Eigenvectors and Frequencies** Node displacements for all nodes specified in the model description are output to the neutral file in the global co-ordinate system. When mode shapes are read by LMS, the mode shape numbers will not necessarily be the same unless all eigenvectors are exported. Modeller mode shape numbers are preserved during an export, but are not preserved when read into the LMS software.

❑ **Element Matrices** The element matrices (stiffness and mass) are output in element type order. There are three sections required to define a matrix. These are MATSHP, MATVAL and MATDEF.

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

Supported Elements

The following elements are supported by LMS.

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

NASTRAN BDF and DAT Import

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

ABAQUS Input File Import

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

ANSYS CDB File Import

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

PATRAN Interface Files

About PATRAN

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

PATRAN Import

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

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

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

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

PATRAN Export

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

Solver DAT Import

Solver DAT files are created by LUSAS Modeller during the tabulation phase. They contain the data required by LUSAS Solver to perform an analysis. Both Geometry and Mesh data from Solver DAT files may be imported into Modeller.

Point, Line, Surface and Volume geometry data from Solver DAT 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.

STEP Import / Export

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

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

STL Import / Export

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

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

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

Notes:

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

Chapter 4 : Model Geometry

Introduction

There are four geometric feature types in LUSAS.

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

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

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

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

Notes

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

- By default, coordinates are expressed in terms of a global Cartesian axis system. Local coordinates may be used by setting a pre-defined local coordinate active. This is achieved by defining a local coordinate and choosing the **Set Active** command on the context menu. See Local Coordinate Systems.

- Geometry can be imported from other systems. **See File Import**.

Visualising Geometry

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

Show Geometry

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

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

Display Style

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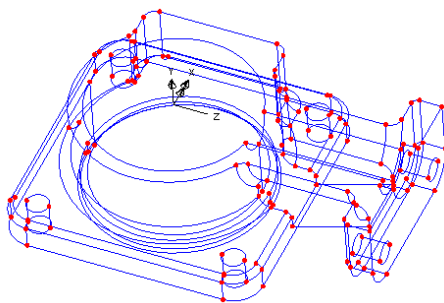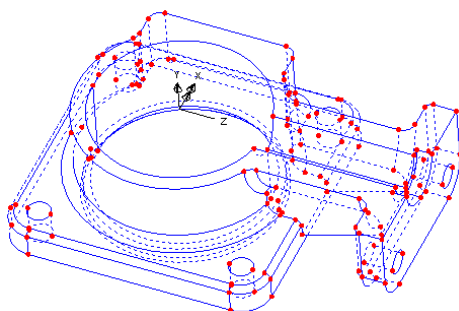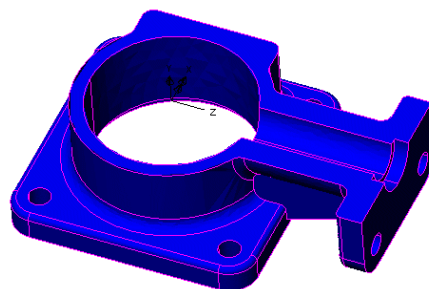

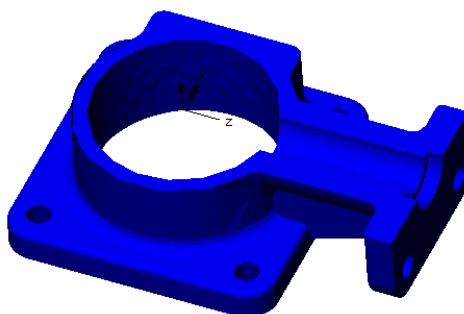**Default wireframe geometry visualisation**
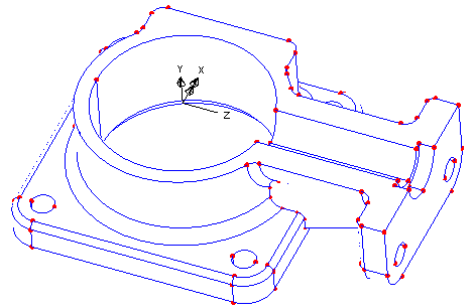
Wireframe with hidden Parts removed

Wireframe with hidden parts draw dashed

Solid fill with wireframe and hidden parts removed

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

Facet Density

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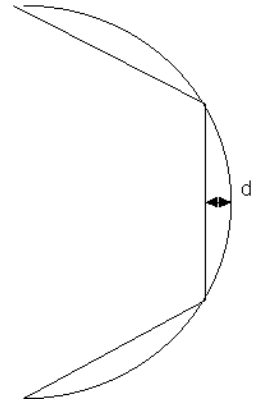
The facet density may be specified either as:

- Maximum facet length (in model units).

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

or for Lines as:

- Minimum adjacent angle (in degrees).

- Minimum number of facets for straight line, full arc line and spline line.

or for Surfaces as

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

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

Notes

- The display speed is inversely proportion to the number of facets used to define the geometry.

- The facet density can be visualised by selecting the **Facet** option on the **Geometry Layer** properties dialog. The facet density display can be restricted to only selected Lines and Surfaces using the **Facet only if selected** option.

Using Colour For Geometry

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

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

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

❑ **Type** Each geometry type has a default pen, associated with it. This option causes all geometry to be drawn in that pen. The colours may be set from the Model Properties. By default the settings are red for Points, magenta for Lines, orange for Combined lines, green for Surfaces, blue for Volumes.

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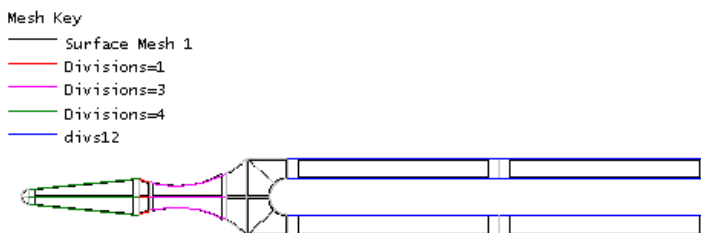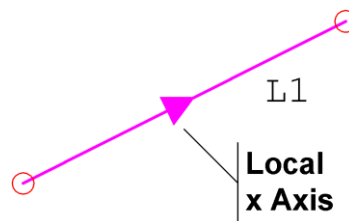

Orientation Axes

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

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

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

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

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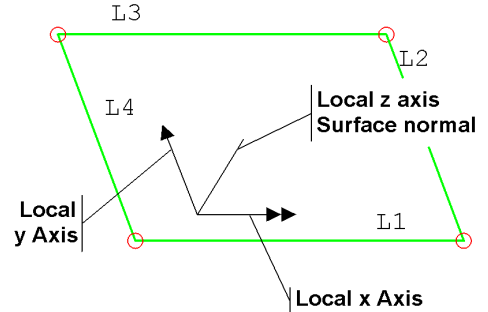
❑ **Volume** axes can be displayed. The origin or the axis is closest to the first point in the first Surface of the Volume definition.
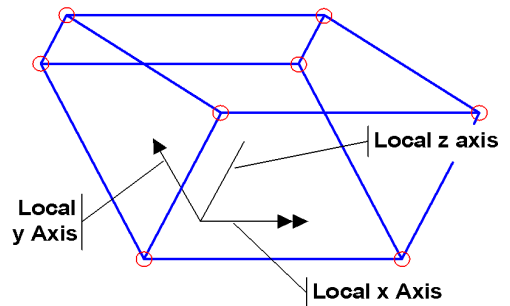
Tip: To aid axes visualisation on larger models choose the **Orientations only if selected** option. This will display axes only on selected features.

Labels

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

Labels may be added to geometry features as follows:

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

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.
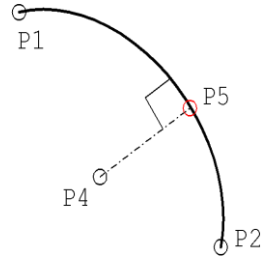
Points

Points define the vertices of the model. Point definition commands are found under the **Geometry> Points** menu item. Points can be defined in the following ways.

❑ **Coordinates** Defines a Point by entering the X, Y and Z coordinates (Z is optional). If a non-Cartesian **local coordinate system** is in use the coordinates are specified in the coordinate system of that local coordinate set. The dialog box labels will be updated to reflect the required coordinate input.

❑ **Cursor** Allows definition of a series of Points on the screen with the cursor. The Points can snap to a grid in the XY, YZ or XZ plane. The out of plane ordinate can be specified as non-zero if desired. This facility is useful for positioning Points on Lines or Surfaces which will be used for splitting that feature later.

❑ **From Mesh** Defines a Point at the position of every node of the selected mesh. **See Geometry From Mesh**. This is useful for defining a Point feature to which loads or supports can be subsequently assigned. The Point must be **equivalenced** with the underlying meshed feature in order for the Point's assigned attributes to be transferred to the underlying nodes. Subsequent re-meshing of the structure with different mesh spacing characteristics may result in movement of the underlying nodal positions.

❑ **By Intersection** Defines a Point or a number of Points at the intersections of two or more selected Lines. When the Lines selected do not physically intersect Points may be created at the nearest intersections. These nearest intersections are controlled by the following options.

    ▪ **All point pairs** creates Points at all possible intersections.

    ▪ **Nearest point pair only** creates a Point on each line where the projection of the Lines is at its nearest.
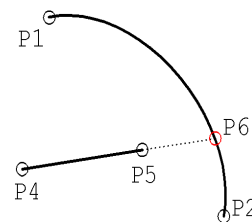
- ▪ **Nearest pair to reference position** defines Points on each line at the intersection nearest to the defined reference Point.
- ▪ **Limit distance between points** only creates Points at intersections that are within a specified distance
- ▪ **Allow extended lines** will create Points at the intersection of the extension of the selected Lines.

❑ **By Projection** Defines a Point at the (perpendicular) projection of a selected Point onto a selected Line or Surface.

❑ **By Extension** Defines a point at the extension of a selected Line. The extension may be defined as a parametric or actual length.

❑ **Make Planar** moves the selected points onto a plane defined either as an offset from an orthogonal plane, or onto a plane defined by 3 coordinates, or as a best fit to the selected points. A local coordinate may be used to define the orthogonal axes if required.

Case Study. Editing Point Properties

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

Lines

Lines define the edges of the model. They are stored in the database by referring to their lower order Point features and in some cases a map which defines their internal shape. The line types available are:
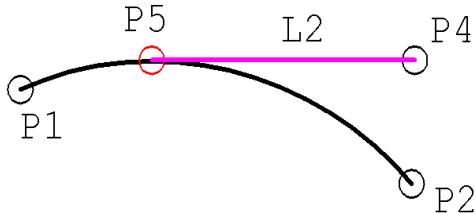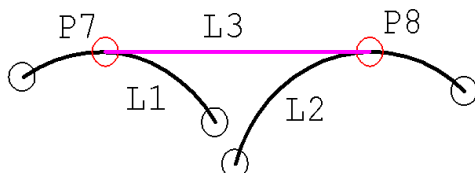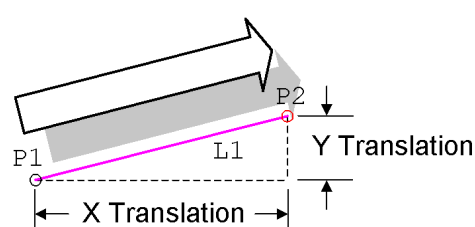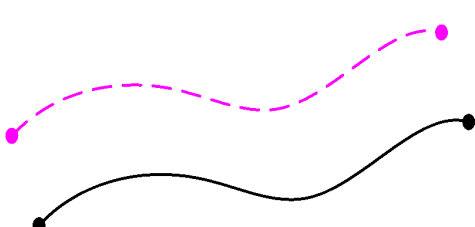
❑ **Line** defined by two Points.

❑ **Arc/Circle** defined by two Points and a Line map.

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

❑ Lines may also be created by **Splitting** lines.

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

❑ **Elliptical Arc/Ellipse**

❑ **Composition Line** defined by a Line and Surface map.

❑ **Intersection** defined by two Surface maps.

❑ **Isoparametric**

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

General Line Definition

❑ **Coordinates** Defines a Line by entering the X, Y and Z coordinates. Entering more than two coordinates will define linked Lines. If a non-Cartesian local coordinate system is in use the coordinates are specified in the coordinate system of that local coordinate.

❑ **Cursor** Allows definition of a series of straight Lines on the screen with the cursor. The Lines can snap to a grid in the XY, YZ or XZ plane. The out of plane ordinate can be specified as non-zero if desired.

❑ **Points**Defines a Line from selected Points. A dialog is displayed to specify the Line type as either Straight Line(s), Arc, or Spline.

❑ **Tangent Point to Line** Defines a Line between a selected Point and the tangent to a selected Arc. An error will occur if no tangent is possible. In this example Line 2 is created by specifying Line 1 and Point 4. Point 5 is automatically created.

❑ **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). Multiple transformations can be specified to act as one complete transformation. In this example Line 1 is created by sweeping Point 1 in a *translation* in X and Y.

❑ **By Offsetting** Defines a Line which is offset parallel to a selected Line. An additional Point may be selected to define the plane in which the new Line is to be defined. 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. The Points in selection memory define the start of each Line, the Points in selection define the end of each Line. The Points should 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.
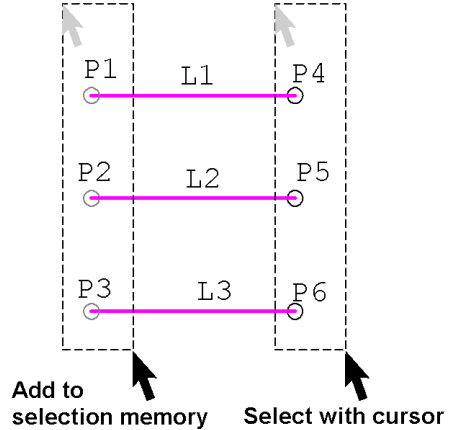
❑ **By Intersection** Lines may be defined by intersecting two or more Surfaces. Intersects all Surfaces within a single selection with all other Surfaces within that selection. If no intersection is found a warning will be issued.
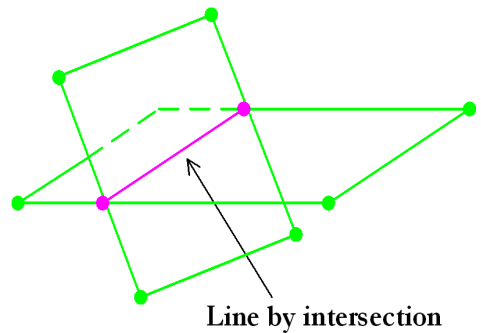
❑ **By Manifolding** (via projection) Existing Lines may be projected or laid onto an existing Surface. A Surface to be projected onto is selected, followed by the Line to be projected and the **Geometry> Line> By Manifolding** menu item is used to create the new manifolded Line. The new Line is created normal to the selected item and will lie on the map of the underlying 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> Surface menu item) prior to using the **Geometry> Line> By Manifolding** menu item to create the manifolded

Add to selection memory   Select with cursor


Line by intersection
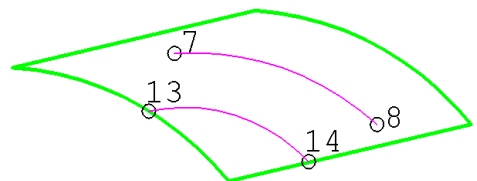
line. Points created prior to choosing
the command can be, but do not have
to be, on the boundary of the Surface.
In this example Points 7 and 8 lie
within the Surface boundary, while
Points 13 and 14 lie on the boundary.

Arc and Circle Definition

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

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

- **Start Point, Bulge Point and
  End Point** Defines an Arc or
  Circle which passes through
  three coordinate points. In this
  example Line 1 is defined by
  selecting Points 1, 2, 3 and
  specifying Point 2 as the
  'bulge' point. Next, Line 2 is
  defined by selecting P4 and
  P5, then entering the
  coordinates to define a 'bulge'
  Point, P6 (which is not
  actually created)
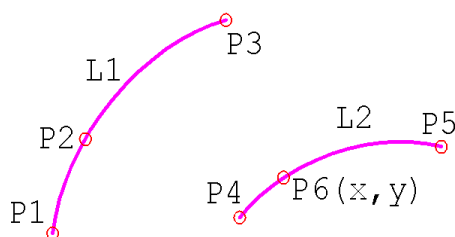
- **Start Point, Directional
  Point, End Point, and
  Radius** Defines an Arc or
  Circle between two coordinate
  points, to a specified radius. A
  third coordinate point is
  required to indicate the
  direction in which the arc or
  circle bulges. In this example
  arcs are created from Point 1
  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
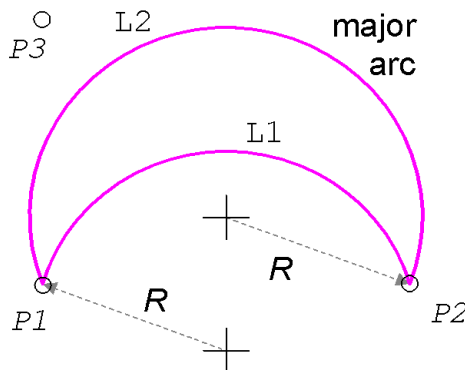
subtends an angle less than 180 degrees.

- **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).
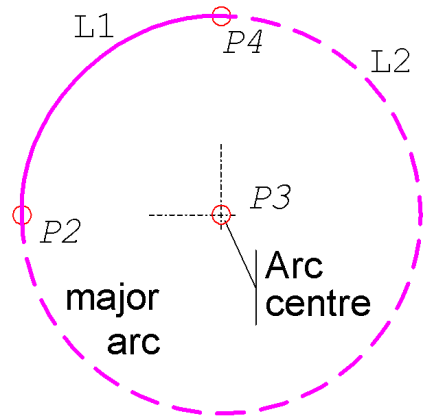
❑ **Tangent to Lines** Inserts an Arc or Circle with a specified radius, tangent to two selected Lines. The selected lines must be straight Lines or Arcs. In this example Line 3 is created by selecting Lines 1 and 2, then specifying a radius and Minor Arc. Alternatively, Line 4 is created by selecting Lines 1 and 2, then specifying a different radius and Major Arc.
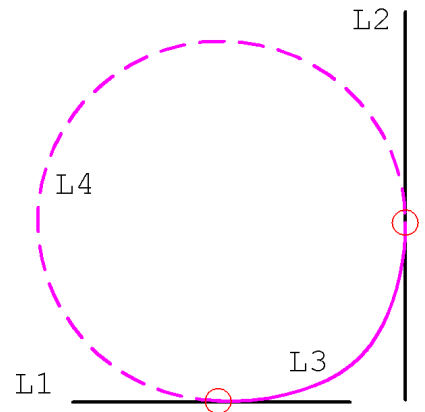
❑ **By Sweeping Points** Defines Arcs by sweeping selected Points through a specified rotation. In this example Point 1 is swept into Line 1 using a rotational transformation and choosing the Minor Arc option.
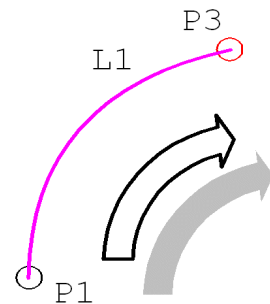
Spline Definition

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

❑ **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 t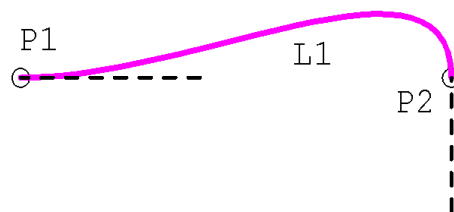angent vectors. In this example Line 1 is defined by specifying Points 1;2 and inputting start and end tangents by vectors (1, 0, 0) and (0, -1, 0) respectively. Both the direction and length of the end tangents control the spline shape. In this example, a different shape of spline would be defined, 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.
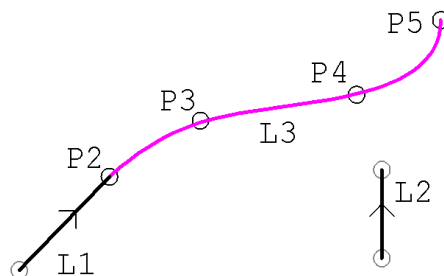
Splitting Lines

Lines may be split at a Point, or split into a number of equal or unequal divisions to form new Lines. Only straight Lines and arcs can be split using these methods. The splitting Lines command dialogs contain options to automatically split and delete the original Lines, and to replace the split Lines with Combined Lines. When splitting Lines the original Line may be deleted, but only if it did not define any Surfaces, or if such surfaces are modified to use the new Lines via the "Use in dependent Surfaces" option. 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> Splitting** menu item.

❑ **At a Point** Splits an existing Line into two new Lines at a selected Point on the Line. Arcs can only be split using a Point on the arc. Split Lines defining a Surface may be deleted and replaced by a Combined Line. 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.
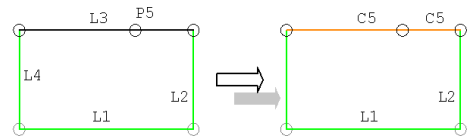
❑ **In Equal Divisions** Splits an existing Line into a specified number of equal divisions. A new Line is defined at each division. In the example shown Line 1 is split into 4 new Lines of equal length, and the original Line is deleted.
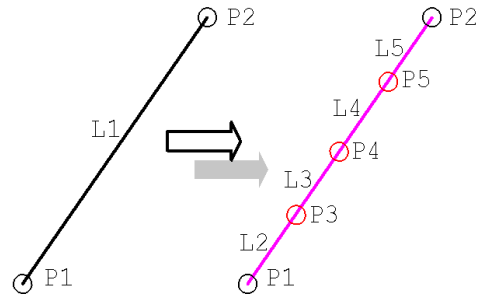

❑ **At Parametric Distances** Splits an existing Line into a number of divisions based on specification of parametric distance values along the Line. The direction of the existing line is used to calculate the splitting positions. A new line is created at each division. In this example Line 1 is split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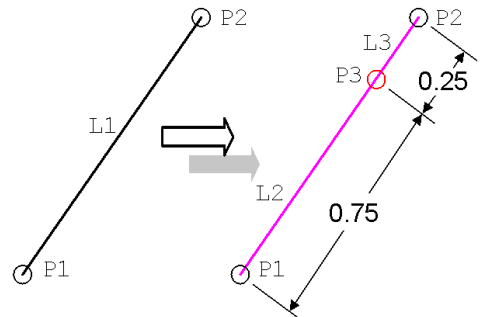
Combined Lines

A Combined Line is a Line which is composed of several continuous individual Lines. Combined Lines may be used in exactly the same was as other Lines to allow Surfaces, to be

meshed using regular mesh patterns. This is especially useful for meshing surfaces defined by more than four Lines.

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

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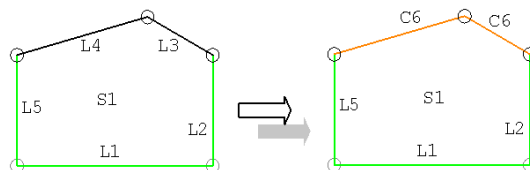
Notes

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

- Mesh attributes may not be assigned to Combined Lines, but must be assigned to the Lines defining the Combined Line.

- Using Combined Lines in the definition of Surfaces provides an additional means of producing transition meshes.

Case Study. Using Combined Lines

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

1. Define a Surface using the New Surface button ▢, and enter the coordinates (0,0,0), (100,0,0), (120,20,0), (100,100,0) and (0,100,0).

2. Define a Line mesh attribute (using **Attributes> Mesh> Line**) with of element type None with mesh divisions of 1 and assign it to the shorter Line on the left hand side of the surface.

3. Define a Line mesh attribute (using **Attributes> Mesh> Line**) with of element type None with mesh divisions of 3 and assign it to the longer Line on the left hand side of the surface.

4. Define a Surface mesh attribute using (using **Attributes> Mesh> Surface**) with Plane Stress, Quadrilateral, Linear elements and assign it to the surface. This will automatically select an irregular mesh for a five-sided Surface as shown below.

5. Now define a Combined Line by selecting the Lines on the left hand side of the model and press the [+] button from the Line menu item.

6. Since the Surface definition has been altered the Surface will be remeshed. A regular mesh will now be adopted because the line divisions on the Combined Line match those on the opposite Line.

Irregular Mesh

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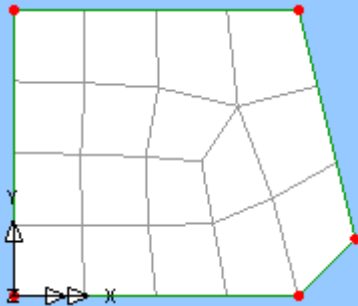
Regular Mesh Using Combined Lines

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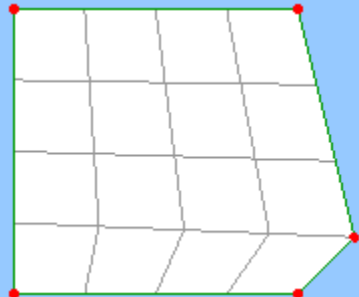
Surfaces

Surfaces define the faces of the model. The Surface types are:

❑ **Regular** defined by 3 or 4 Lines.

❑ **Irregular** defined by 5 or more Lines.

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

Surface Definition Commands

The following commands are for defining Surfaces directly:

❑ **By Coordinates** Defines a Surface by entering a list of X, Y and Z coordinates (Z is optional). If a non-Cartesian local coordinate system is in use the coordinates are specified in the coordinate system of that local coordinate. Coordinates can be entered using global (default) or local coordinate systems.

❑ **By Cursor** Allows definition of a series of flat rectangular Surfaces on the screen with the cursor. The Surfaces can snap to a grid in the XY, YZ or XZ plane. The out of plane ordinate can be specified as non-zero if desired.

❑ **By Points** Defines a Surface from three or more selected Points. A Surface can be defined from any number of Points.

❑ **By Lines** Defines a Surface from the selected Lines. When defining Surfaces in this way, any number of Lines may be specified in any order and a Surface will be formed from the greatest number of lines that form a closed loop. If a number of disconnected Lines are selected a **Lofted Surface** will be created. A lofted Surface is defined as Surface with a smooth transition between 2 or more Lines.
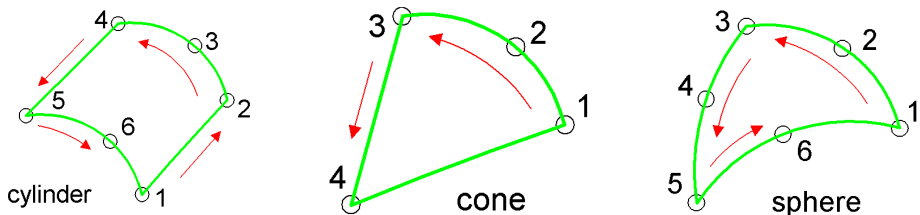
Surfaces generated from selected Lines

85
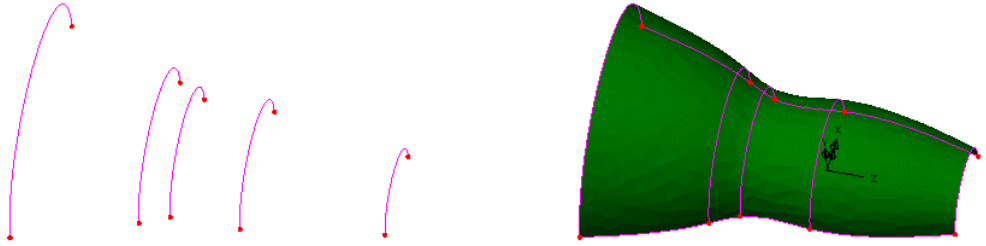
Lofted Surface generated from selected Lines

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.

Holes in Surfaces

❑ **Create** Defines a hole or number or holes in a Surface. Closed loop(s) of Lines or the Surface(s) representing the hole(s) and the Surface to be holed should be selected prior to choosing this menu option.

❑ **Move** Moves or modifies selected hole(s) within a Surface. To move a hole select the Lines and/or Points forming the perimeter of the hole. If only some Lines or Points are selected the hole will be stretched by only moving the selected features. **Note.** Moving a hole actually deletes the original hole and recreates a new hole. This means the feature numbers of the Points and Lines will not be maintained.

❑ **Copy** Creates multiple holes within a Surface.

❑ **Delete** Deletes the selected hole(s). The hole boundary lines may be deleted, retained as Lines or used to create new Surfaces.

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

Case Study. Creating a Surface with Holes

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

1. Create the outer Surface and a separate Surface for each hole.

2. Select all Surfaces.

3. Use the **Geometry> Surface> Holes> Create** menu item to create a single Surface with holes. Select the **Delete geometry defining holes** option to remove the Surfaces defining the holes. Note, when creating models with holes for section property calculation purposes the **Delete geometry defining holes** option should be de-selected to retain the original surface representing the hole(s) otherwise incorrect section properties may be calculated.

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.
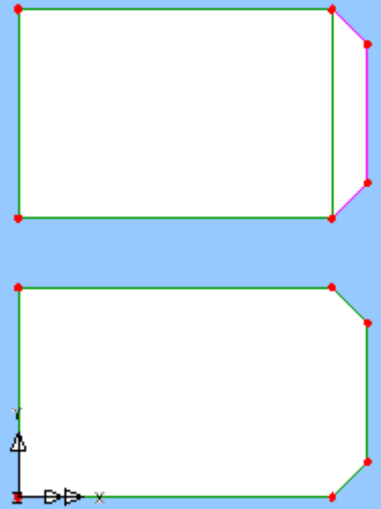
Sweeping Surfaces

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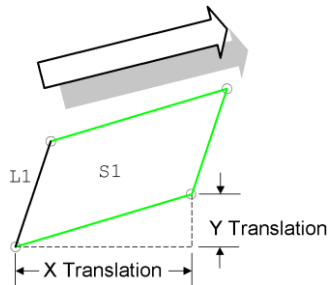
A cylindrical Surface is defined by sweeping Line 1 through a rotation:
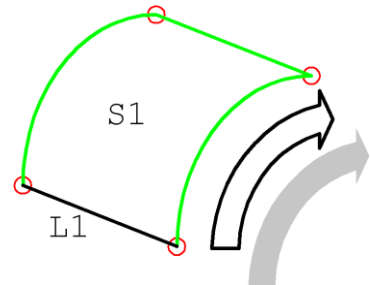
Offsetting

❑ **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.
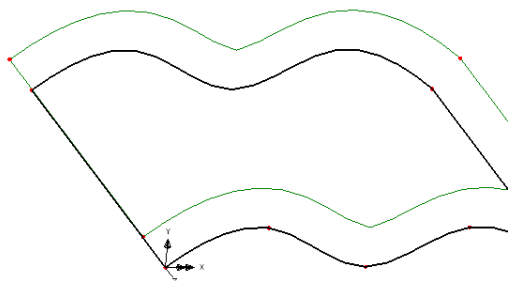
Splitting Surfaces

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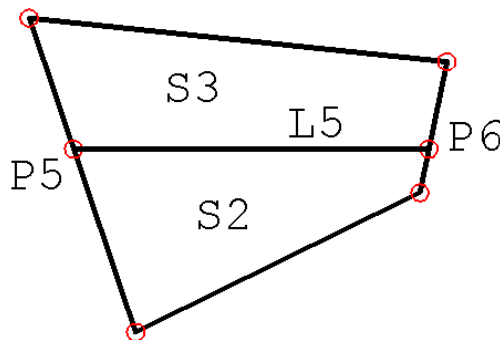
❑ **By Points** Splits a selected Surface at two boundary positions indicated by Points. A new Line will be manifolded onto the existing Surface and the Surface will be split at this Line. In the example shown, Surfaces 2 and 3 are created in the parametric space of the original Surface using the Points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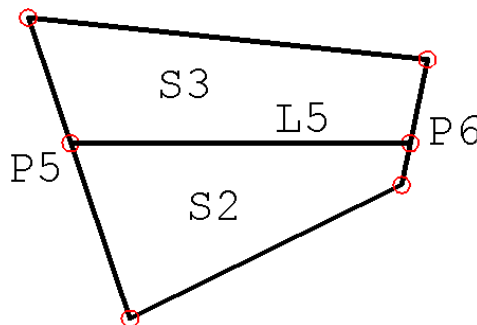
❑ **At a Point** Splits a selected Surface
at a single Point defined on the
surface inside the boundary. Four
new Surfaces are created by
manifolding straight Lines or arcs
onto the existing Surface using the
relative position of the splitting
Point. In this example, the original
Surface is split at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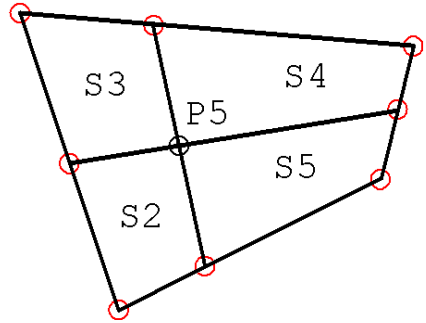
❑ **In Equal Divisions** Splits a selected
Surface into separate Surfaces at a
specified number of equal divisions.
The direction of the split is
expressed using local Surface axes.
In the example shown here, the
original Surface is split in its local x
direction into equal divisions
forming 3 new Surfaces.
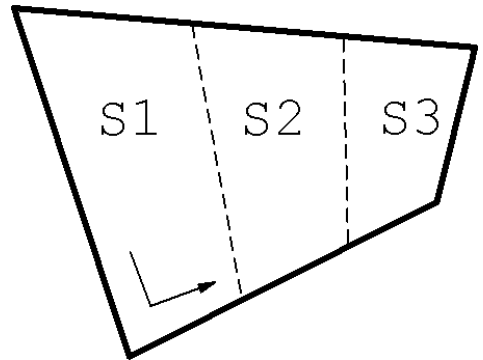
❑ **At Parametric Distances** Splits a
selected Surface into separate
Surfaces at specified parametric
divisions. For example, specifying
the parametric distances as 0.25 and
0.75 will split a Surface at the 1/4
and 3/4 position in parametric space
in the specified local direction. In
this example, the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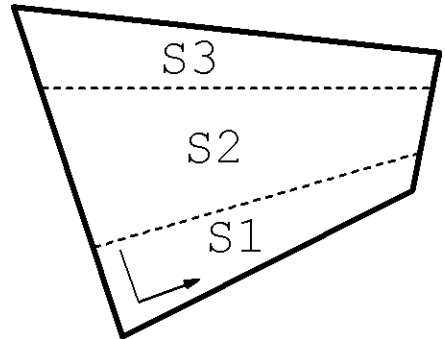
Joining

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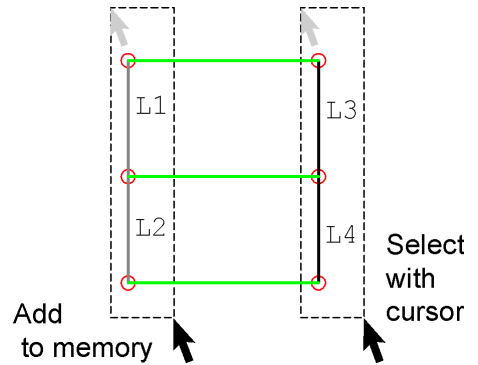
In the example shown Surfaces 1 and 2 are defined by first selecting Lines 1 and 2,
then adding them to selection memory, then selecting Lines 3 and 4 and using the
Surface by joining command.

Intersection

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

Manifolding

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

❑ **By Manifolding** Existing Surfaces may be projected or laid onto an existing Surface.
A Surface to be projected onto is placed into Selection Memory and the Surface to be
projected is then selected prior to choosing the **Geometry> Surface > By
Manifolding** menu item. The new Surface is created normal to the selected item (in
selection memory) and will lie on the map of the underlying Surface.

❑ **Manifold by Lines** Defines a
Surface positioned on an existing
underlying Surface with its
boundary specified by edges or
vertices. In this example a Surface
is created by Joining any
combination of the Points and Lines
shown. Any new Lines created will
automatically be manifolded onto
the underlying Surface.
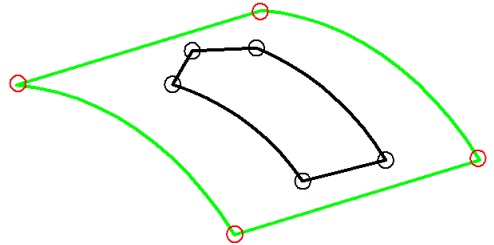
Extrusion

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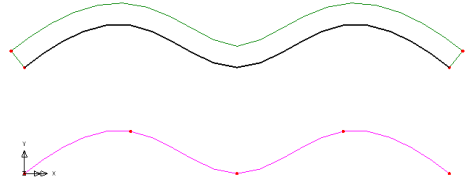
Volumes

About Volume Features

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

❑ **Solid Volumes** are defined by a number of connected Surfaces. These are recognised
by the geometry engine which allows Boolean operations to be performed. Solid
Volumes are the default form of Volume and are created when geometry is defined in
Modeller.

❑ **Hollow Volumes** are defined by a number of disconnected Surfaces. These are not
recognised by the geometry engine. Hollow Volumes are usually only of use when the
geometry has been imported from CAD.

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

❑ **Regular** volumes may be defined as **Tetrahedral** (4 sided volume with all faces
defined by triangular surfaces), **Pentahedral** (5 sided prism with top and bottom faces
defined by triangular surfaces and side faces defined by quadrilateral surfaces),
Hexahedral (6 sided cuboid with all faces defined by quadrilateral surfaces).

❑ **Swept Irregular** are defined by 2 identical irregular Surfaces joined at equivalent
positions on the boundaries by quadrilateral Surfaces. The Lines joining the two
irregular Surfaces must be either straight Lines or arcs with a common centre.

❑ **Irregular** are defined by any number and type of surface but can only be meshed with
tetrahedral elements.

Defining Volumes

The following commands are for defining Volumes directly:

❑ By using the **Shape Wizard** to create regular volumes.

❑ By selecting **Surfaces** to define a Volume from four or more selected connected
Surfaces. The Surfaces may be entered in any order. When defining Volumes in this
way any number of Surfaces may be selected to define a Volume. Duplicate and
unconnected Surfaces will be filtered out.

❑ **By Sweeping** to define a Volume by sweeping a selected Surface through a transformation (translation, rotation, mirror or scale).
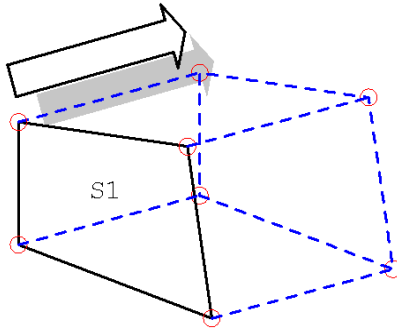
A Surface is swept through a *translation* to create a Hexahedral:
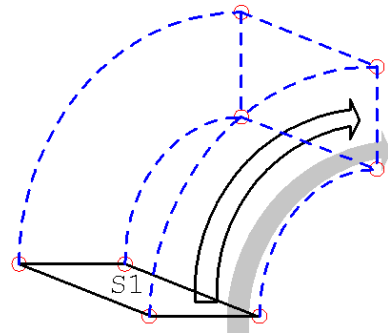
A Surface is swept through a *rotation* to create a Volume:

❑ **By Splitting** Splits a selected Volume by a selected Surface.

❑ **By Joining** Joins two selected features to form a Volume, either a Point to a Surface, or a Line to a Surface, or two Surfaces. The features are joined by straight lines.
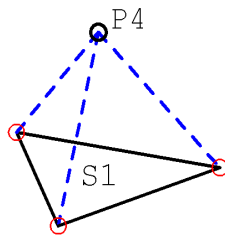
Surface 1 is joined to Point 4 to form a pentahedral.
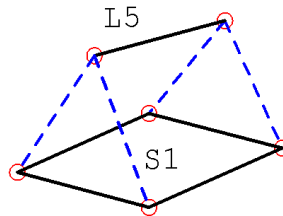
Surface 1 joined to Line 5.
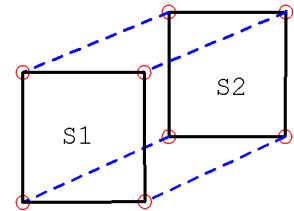
Surface 1 is joined to Surface 2 to form a hexahedral. Two triangular Surfaces joined this way would form a pentahedral.

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

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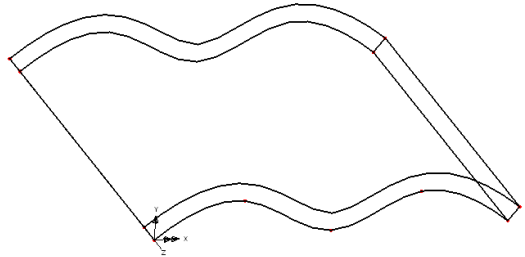
Coalescing Volumes

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

Case Study. Coalesce Volumes

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

1. Open results file and use the **File> Save As** menu item to save it as .mdl file.

2. Use **Control-A** to select the mesh and convert it to Volumes using the
   Geometry> Volume> From Mesh menu item and select the **Coalesce Volumes**
   option.

3. Create a loadcase.

Delete Holes

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

Delete Voids

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

Orientating Volume Axes

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

❑ **Axis to Surface**
❑ **Cycle Axes**
❑ **Cycle Relative**

Hollow Volumes

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

Hollow volumes may be open or closed. To understand the difference consider just one defining edge of the volume between two defining surfaces. If that edge consists of exactly one line, or exactly two lines of very similar length and shape, then those lines can be considered a matching pair. If all the defining edges of the volume can be matched in this way, the volume is said to be closed. However, if one or more edges of the volume cannot be matched in this way then the volume is said to be open.
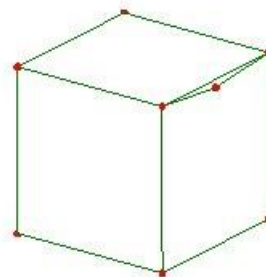
For the example shown most of the lines form matching pairs. However, one of the defining surfaces has an edge that is defined by two lines, with a point in the middle. This pair of lines cannot be matched to the corresponding single line on the adjacent surface, and so this volume is open.

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

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

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

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

Shape Wizard

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

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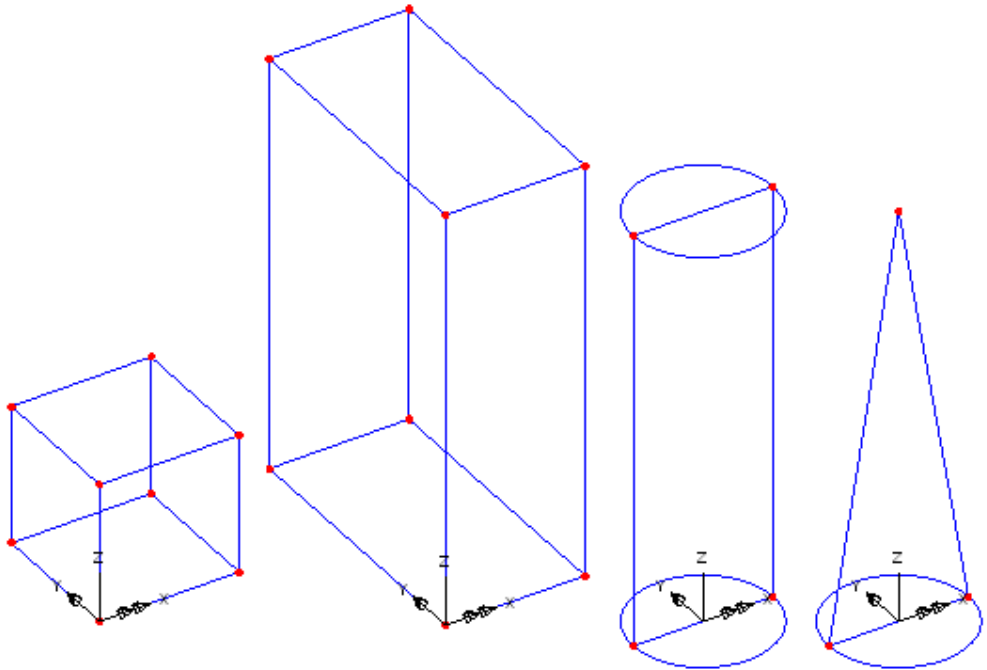
Notes.

- Specifying a negative length or height results in that dimension being defined along the negative axis direction.

- For the cylindrical shape there is an option to create the cylinder with a seam which effectively creates a cylinder using only one surface rather than the default two surfaces.

Boolean Geometry Construction

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

❑ **Union (with simplify internal geometry)** enables a Surface or Volume to be defined by union of any number of selected Surfaces or Volumes.

❑ **Union (without simplify internal geometry)** enables a number of Surfaces or Volumes to be defined by union of any number of selected Surfaces or Volumes.

❑ **Subtraction** enables a Surface or Volume to be defined by subtracting one Surface or Volume from another Surface or Volume.

❑ **Intersection** enables a Surface or Volume to be defined as the intersection of two selected Surfaces or Volumes.

❑ **Slice** Enables a selected Volume to be sliced by a plane and the resulting geometry to be deleted either side of the slice if required. The slice plane may be defined in any global or local plane which may be visualised prior to the slice operation. A Surface of any shape may be used to slice a Volume by Subtracting the Surface from the Volume.
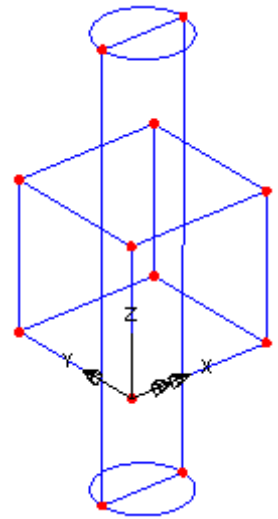
Separate (unconnected) cube and cylinder = 2 volumes
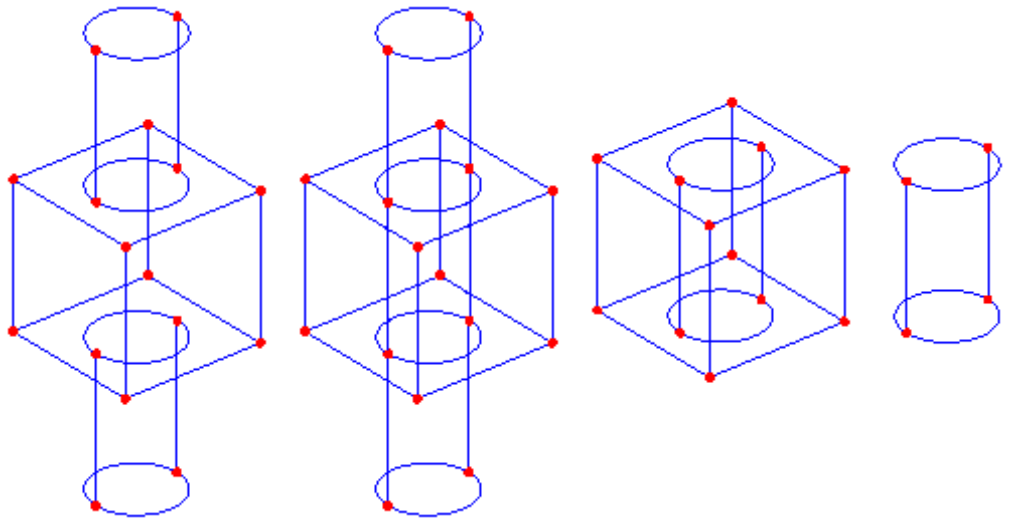
Union of cube and cylinder (simplify internal geometry) = 1 volume	Union of cube and cylinder (no simplify internal geometry) = 4 volumes	Subtraction of cylinder from cube = 1 volume	Intersection of cube and cylinder = 1 volume

Geometry From Mesh

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

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

The currently selected elements are converted using the **Geometry> Point> From Mesh, Geometry> Line> From Mesh, Geometry> Surface> From Mesh, Geometry> Volume> From Mesh** menu items.

❑ **Point** creates Points from nodes.

❑ **Line** creates Points from nodes and Lines from beam elements, surface and volume element edges.

❑ **Surface** creates Points from nodes, Lines from beam elements and Surfaces from surface elements and volume element faces.

❑ **Volume** 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 an 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.

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 and Copying Geometry

Geometry may be moved or copied to new positions 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.
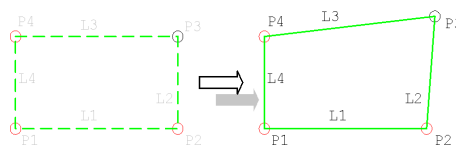
Note: When moving holes the Surface is actually deleted and recreated so the feature numbers will not be maintained.

❑ **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.
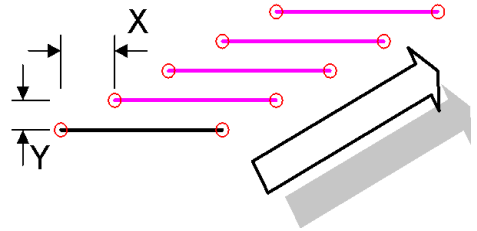
Sweeping Geometry

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

Transformations

Transformations are used in two ways:

❑ When moving, copying or sweeping geometry a transformation is specified, and may
be saved if required.

❑ For special applications such as to orient discrete point and patch loads, or to define a
reflective mirror plane for thermal analyses.

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

The following transformation types are available:

❑ **Translation** Linear translation along
a specified vector coordinate. The
vector coordinate will use the **active
coordinate set**. 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.
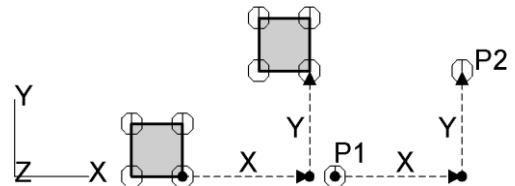
❑ **Rotation** (in a global plane) A specified angular rotation in either the XY, YZ, or ZX global plane at a specified origin.

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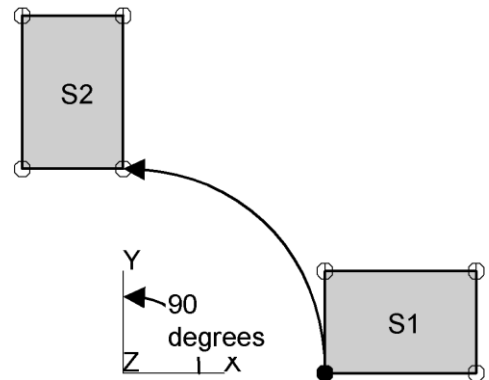
❑ **Mirror** A mirror plane may be defined by specifying three points in space to define an arbitrary plane, or two points to define a plane parallel to either the Z, Y, or X axis. Three Points added to selection memory can be used to define an arbitrary mirror plane, and two Points added to selection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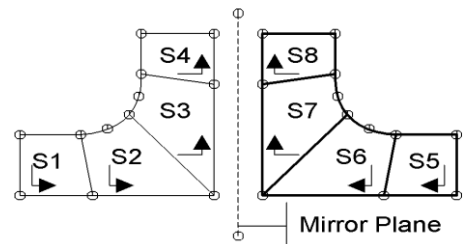
Note:.Care should be taken when mirroring Lines and Surfaces as their orientations may be reversed so some Surfaces may effectively be turned upside down. Lines may also point in the opposite direction. See **Changing Geometry Orientation** to resolve any problems.

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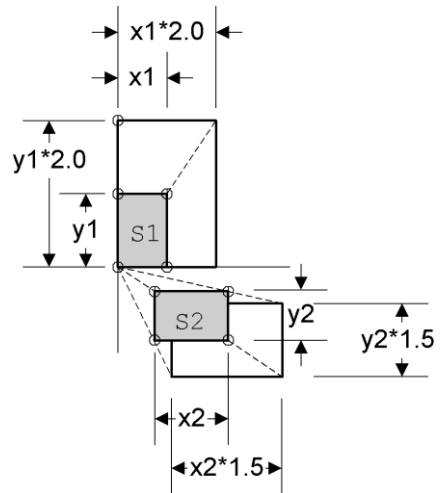
❑ **Matrix rotation** A direction cosine either specified directly, or using two or three Points added to selection memory. Points in selection memory define a plane, the rotation of the global XY plane to this new plane defines the transformation. Note. If the determinant of the matrix is not unity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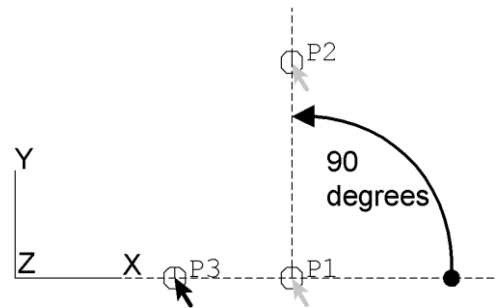
This example defines a rotation by defining a plane relative to the global Cartesian axis set by indicating three Points: at the origin (P1), along the local x axis (P2) and defining the local xy plane (P3). The resulting transformation is a rotation of 90 degrees about the global axes.

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.

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.
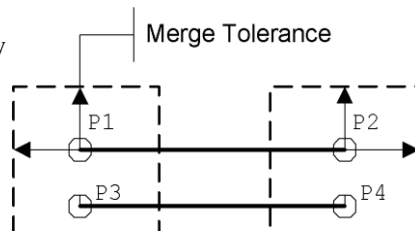

```
P1: LDG1, SUP1, EQV1 ...
P3: LDG1, SUP1, EQV1 ...
```

❑ **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 ...
```


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

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

Merge Case Study 1

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

```
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
```
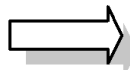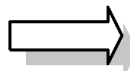
Ignore Assignments →

```
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.
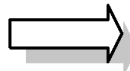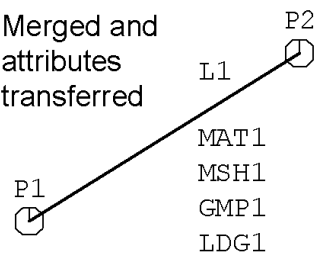
Using the Unmerge Commands

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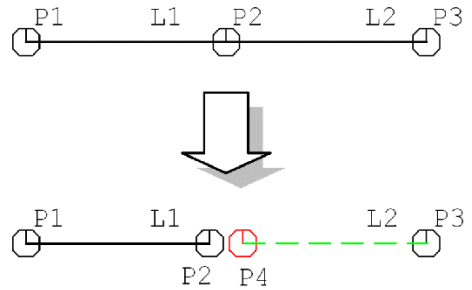
❑ **Unmerge Point in Surface** Unmerge Point 5 in Surface 2. A new Point 8 is copied from Point 5, a new Line 9 is defined and Line 6 is redefined using Point 8. New features are set to be unmergable. Point 5 and Point 8 are coincident.
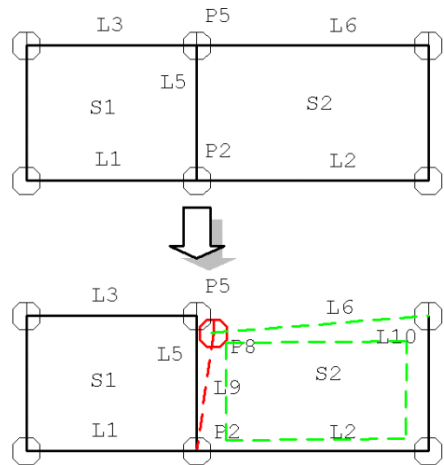
❑ **Unmerge Point in Volume**Unmerge Point 1 in Volume 1. A new Point, 2 new Lines and a new Surface are defined and affected Surfaces and the Volume are redefined. The new Point is coincident with Point 1.
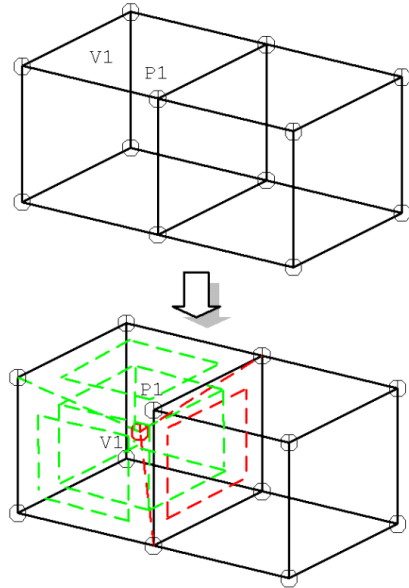
❑ **Unmerge Line in Surface**Unmerge Line 5 in Surface 2. A copy of Line 5 is defined which replaces Line 5 in the definition of Surface 2. Points are unaffected.
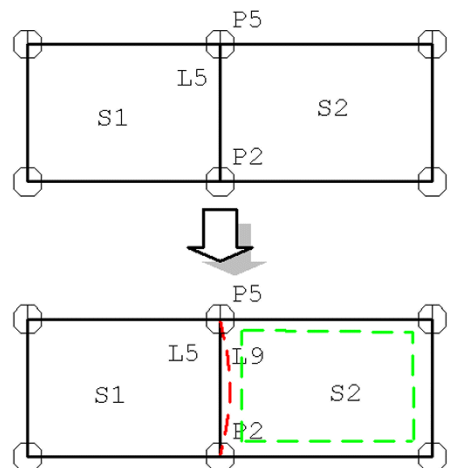
❑ **Unmerge Line in Volume** Unmerge Line 1 in Volume 2. A copy of Line 1 is defined joining the same end Points The new Line replaces Line 1 in the definition of the affected Surfaces in Volume 2
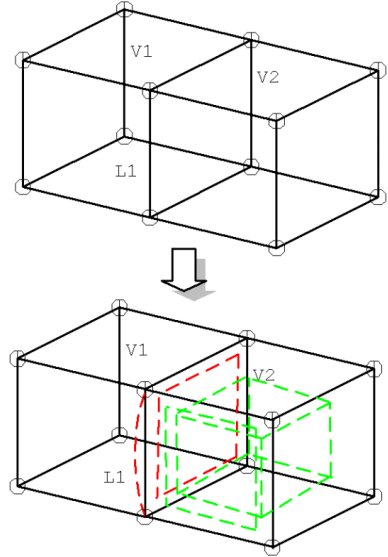
❑ **Unmerge Surface in Volume** Unmerge Surface 1 in Volume 2. A copy of Surface 1 is defined which replaces Surface 1 in the definition of Volume 2.
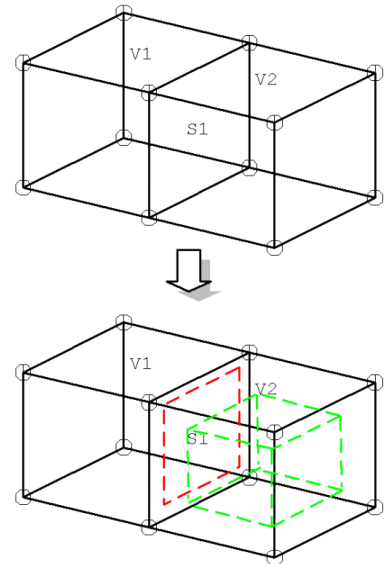
Case Study. Forcing Features to Merge

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 ![icon] 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 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.

The following commands enable the local axes of Lines, 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)**

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

This example the effect of a Line reversal. The local x axes of all elements on this Line will be reversed.
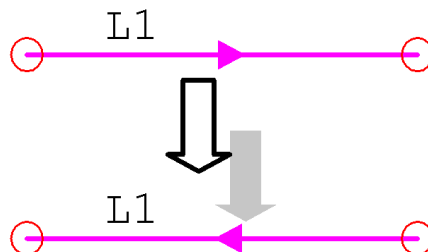
In the Surface example the local x axis remains along the first Line in the Surface definition, but the Surface normal is reversed. Elements meshed on this Surface will be inverted.
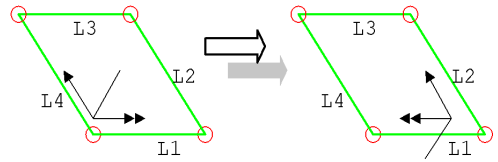
❑ **Axes to Surface** Volumes may have their local z direction set from a selected Surface. Volume axes are defined by the direction of the axes on the first Surface in their definition. This command reorders the Surfaces defining the Volume such that the first Surface in the definition is the selected Surface. The local x and y axes may then be changed using the Cycle surface command.

❑ **Cycle** Surfaces (or surface/solid elements in a mesh-only model) may have their definition order cycled, and Volumes may have the definition of the first Surface in the Volume definition cycled (and for mesh-only models, Elements may have the definition of the first Face in the Element definition cycled). In this example, the original Surface defined by Lines 1, 2, 3 and 4 is shown in grey. Cycling by changes Surface definition to Lines 2, 3, 4, 1. Cycling again defines the Surface as Lines 3, 4, 1, 2 . Surface normal directions remain consistent throughout.
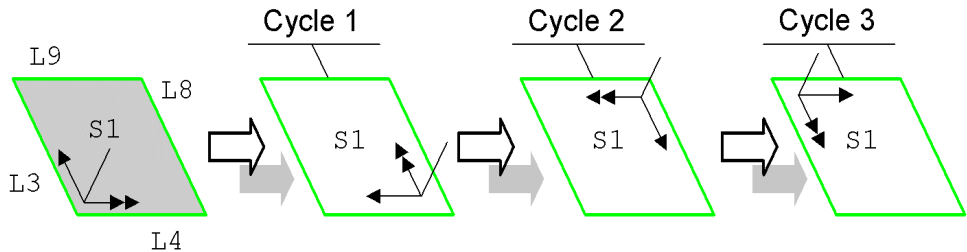
❑ **Cycle Relative** Cycles the definition order relative to the first feature of the same type in the current selection. Surfaces and Volumes (or surface/solid elements in a mesh-only model) may be reoriented in this way. In this example Surfaces 2, 3 and 4 are cycled using Surface 1 (shown greyed) as the reference orientation.
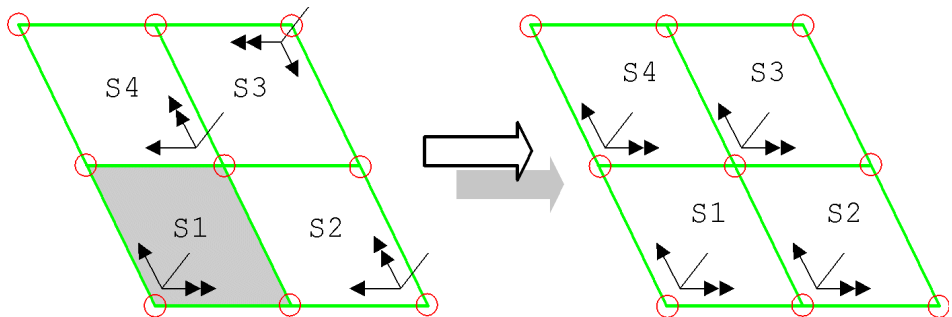
❑ **Cycle to Neighbours** (Mesh-only models) Cycles the element axes of all

---

112

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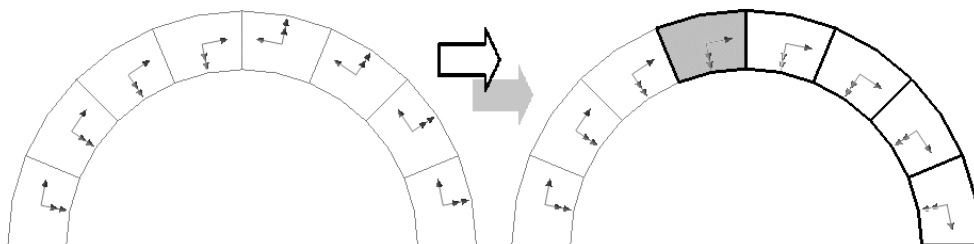
Cycle to Neighbours (correctly aligns element axes of the selected elements with initially selected neighbouring element)
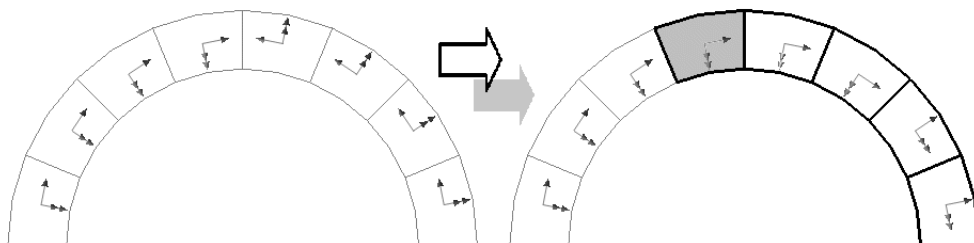
Cycle Relative (only aligns element axes of the selected elements to best angle with respect to initially selected neighbouring element)

❑ **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 determine by the orientation of the Lines defining the first Surface in the Volume the orientation of the Volume changes.

Case Study. Changing Element Orientation

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

1. Draw the orientation axes for the features which are required to be consistent (Geometry Layer Properties). It may be useful to draw just Surface normals.

2. In the case of lines, reverse any lines whose orientation is to be changed.

3. In the case of Surfaces, reverse Surface normals (local z axis) and cycle Surfaces xy axes until all axes are consistent. Surfaces can be cycled relative to a reference Surface if required.

4. In the case of Volumes, set up the Volume local z axis and cycle xy axes by cycling the first Surface in the Volume definition until all axes are consistent. Volumes can be cycled relative to a reference Volume.

CAD Interfacing

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

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

Chapter 5 : Model Attributes

Introduction

Attributes are used to describe the properties of the model. Attributes are assigned to geometry **features** (or to **mesh objects** in a mesh-only model) and are not lost when the geometry is edited, or the model is re-meshed. Attribute assignments are inherited when geometry features are copied and are retained when geometry features are moved. The attribute types are:

General Attributes

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

Specific Attributes

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

❑ **Retained Freedom** specifies the master nodes used in a Guyan reduction or superelement analysis.

❑ **Damping** defines the damping properties for use in dynamic analyses.

❑ **Birth and Death** allows elements to be added (birth) and removed (death) throughout an analysis, e.g. in a tunnelling process or a staged construction.

❑ **Equivalence** allows nodes which are close to each other but on different features to be merged into one according to defined tolerances.

❑ **Influence** parameters define the type of behaviour of the structure at and around an influence point

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

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

❑ **Crack tip** define a crack tip attribute to allow a crack tip location to be defined at a point or line in a model.

❑ **Design strength** define strength data for use in conjunction with Design Factor Plots

Manipulating Attributes

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

Attribute symbols explained

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

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

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

🔩 A coloured or greyed-out attribute 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.
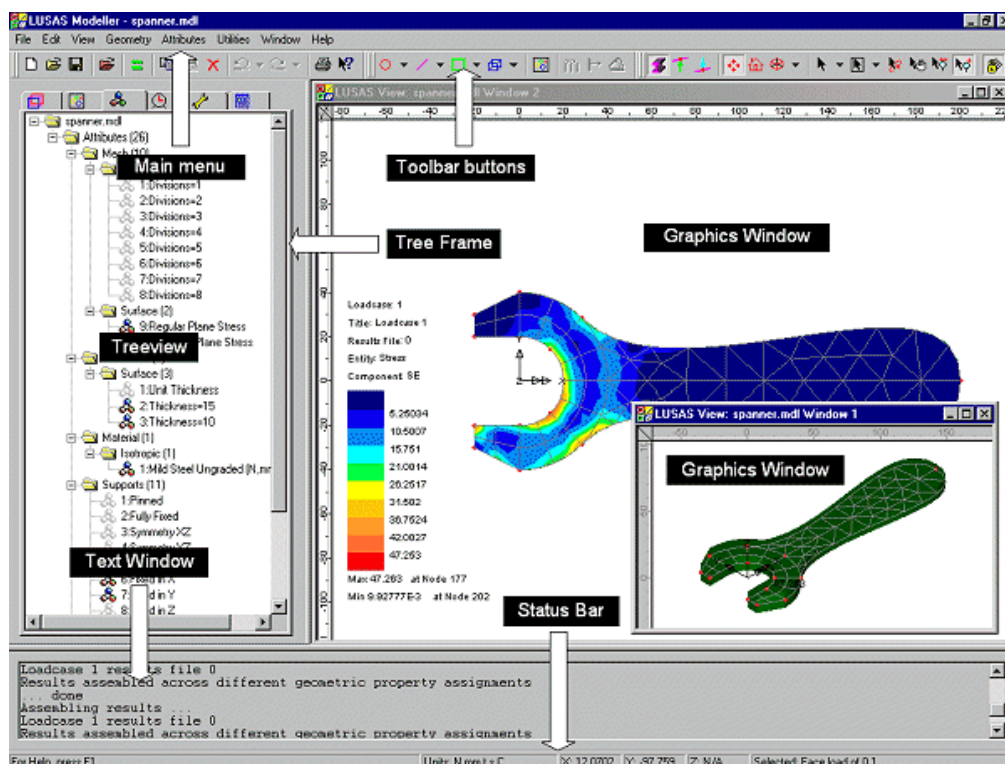
Attribute manipulation

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

- ❑ **Copy** Enables the attribute to be copied and assigned to a selected group or window using paste.
- ❑ **Rename** Attributes can be given meaningful names, for example, 'Steel' to describe a material, or 'Beam - Four Divisions' to describe a Line mesh.
- ❑ **Delete** Existing attributes may be deleted.
- ❑ **Edit Attribute** Allows the properties of the attribute to be modified. If a new name is given a new attribute is created and the original attribute is left unchanged.
- ❑ **Visible** Makes visible all features to which the selected attribute is assigned.
- ❑ **Invisible** Makes invisible all features to which the selected attribute is assigned.
- ❑ **Set As Only Visible** Sets the whole model invisible and then makes visible only those features to which the selected attribute is assigned.

❑ **Advanced Visibility** Provides fine control over the visibility of features to which the selected attribute is assigned.

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

❑ **Select Assignments** Selects the features that have the selected attribute assignment.

❑ **Deselect Assignments** Deselects the features that have the selected attribute assignment.

❑ **Visualise Assignments** Switches on and off the visualisation of features that have the selected attribute assignment in the chosen style.

❑ **Update from library** Updates any attribute data held in the model for the selected attribute if a section library item or material has been updated since first used.

❑ **Assign** Assigns the selected attribute to the selected features. The attribute will only be assigned to features for which 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.

❑ **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.

Visualising Attribute Assignments

Attribute assignments can be visualised using:

- **Attributes layer** The Attributes layer is a window layer in the ▣ 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 ▣ Treeview. Attributes can also be visualised individually by selecting an attribute name in the ⚛ 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.
- **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 (with nothing selected), click the right mouse button in the graphics area. Choose Contours from the context menu. 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 [icon] and Loading [icon] 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 [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].

Drawing Attribute Labels

Labels are a layer in the [icon] 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 [icon] Treeview , then choosing the **Set Default** entry from the context menu. This is useful for models with similar 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 [icon] in the [icon] Treeview .

Meshing a Model

Feature-based geometry 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.

Mesh attributes are defined from the **Attributes> Mesh** menu item for a particular feature type i.e. Point, Line, Surface or Volume. The mesh attributes are then assigned to the required features. Various techniques exist for meshing different types of models and are described below.

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

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.

Style

❑ **Wireframe** Displays the mesh as a wireframe using the pen specified. Only the visible mesh lines are drawn. Click on **Hidden parts** to draw the hidden mesh also (using the pen specified).

❑ **Solid** Displays the mesh as solid panels using the colour specified. Click on the coloured square to change the colour used.

❑ **Outline only** Draws only the outline of the mesh. This is also useful for spotting cracks or discontinuities in the mesh due to features not being merged or equivalenced correctly.

Mesh

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

❑ **Show normals (undeformed mesh only)** Displays the element normal for Surface elements.

❑ **Show element axes (undeformed mesh only)** 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.

❑ **Show activated only** All elements are active unless they have a Deactivate attribute assigned to them.

❑ **Show quadratic effects** Draws quadratic elements with curved edges where appropriate, otherwise straight edges are drawn.

❑ **% of elements remaining** Shrinks the elements to the percentage specified.

❑ **Colour by** Enables the mesh colour to be changed.

- Mesh colour - colours elements in default mesh colour.

- Group - colours elements by group.

- Connectivity - colours element edges by number of neighbours.

- Element Type - colours elements by element type.

- Normals - colours elements by surface normal direction.

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

Visualise

❑ **Joint elements** Marks any joint elements in the model with a symbol.

❑ **Active mesh** Marks the active elements with a symbol.

❑ **Beam end releases** Draws the beam end release for elements that use end releases.

Meshing Points

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

Meshing Lines

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

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

Line Mesh Spacing

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

❑ **Uniform Spacing**

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

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

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

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

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

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

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

Line Element Axis Orientation

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

End Releases for Beam Elements

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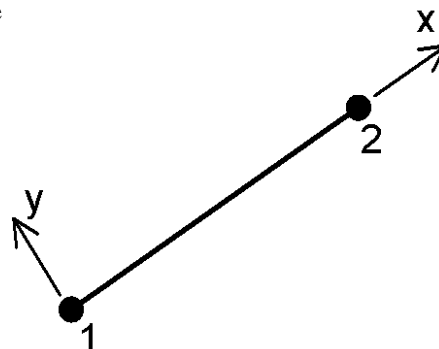
Meshing Surfaces

Regular Surface Meshing

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

❑ In order to generate a regular grid mesh pattern the number of mesh divisions on opposite sides of the Surface must match. If they do not match transition patterns will be used (if allowed in the mesh attribute definition). The examples shown here mesh triangular and quadrilateral Surfaces using both triangular and quadrilateral elements.
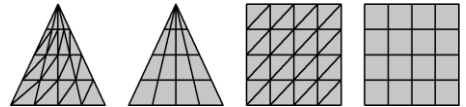
❑ The Surface mesh may be graded using mesh spacing parameters in 'None' element Line meshes assigned to the boundary Lines. In the examples shown here mesh spacing has been used to bias the elements into the apex of the triangle or one corner of the rectangle.
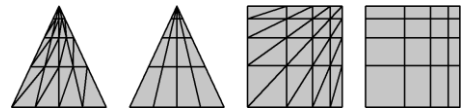
Irregular Surface Meshing

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

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

Meshing Volumes

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

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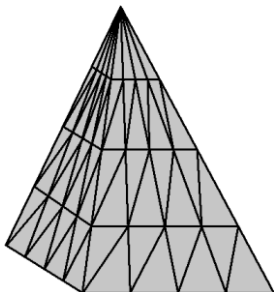
Tetrahedral Elements | **Pentahedral/Tetrahedral Elements**
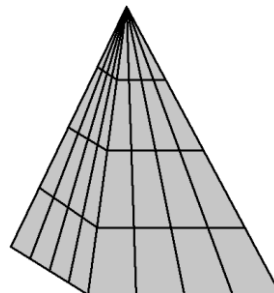
Pentahedral Volumes
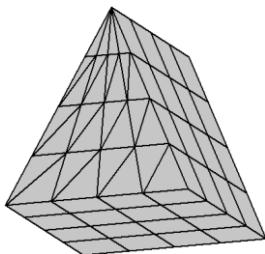
Pentahedral Elements | **Hexahedral/Pentahedral Elements**
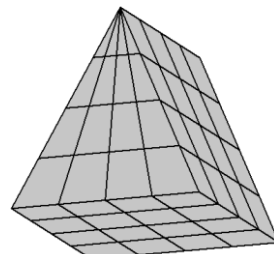
Hexahedral Volumes
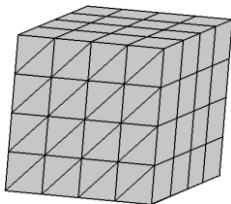
Pentahedral Elements | **Hexahedral Elements** | **Hexahedral/Pentahedral Elements**
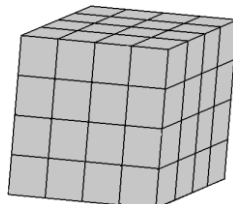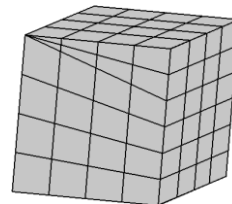
The mesh density for volumes is taken from the boundary surface mesh density.

Regular Volume Meshing

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

Extruded Irregular Mesh

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

Irregular Tetrahedral Meshing

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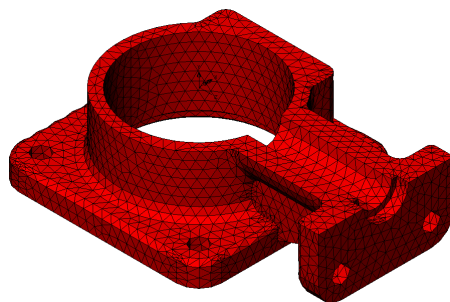
Notes

- A good initial mesh is usually obtained by specifying the element size as approximately1/50th of the diagonal model size. Specifying too small an element size will cause too many elements to be generated and may result in LUSAS using up all the available memory. Specifying too large an element size will cause the meshing algorithm to fail.

- The success of tetrahedral meshing is dependent on the quality of the Surface mesh. If the meshing algorithm fails, invoke edge collapsing or set the Volume mesh to "From defining geometry" and adjust the element size using Line and Surfaces mesh attributes of type None. If the meshing still fails try breaking the Volume into a number of smaller Volumes.

Edge Collapsing

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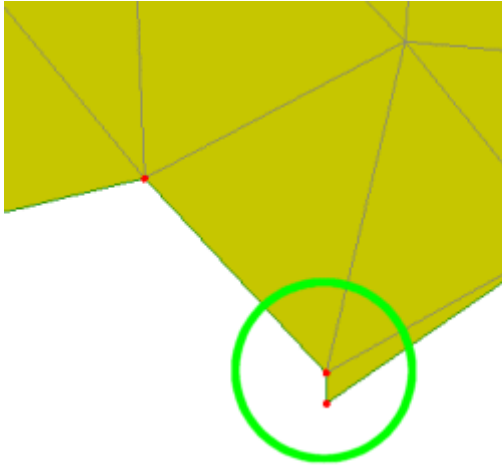
Mesh before edge collapsing
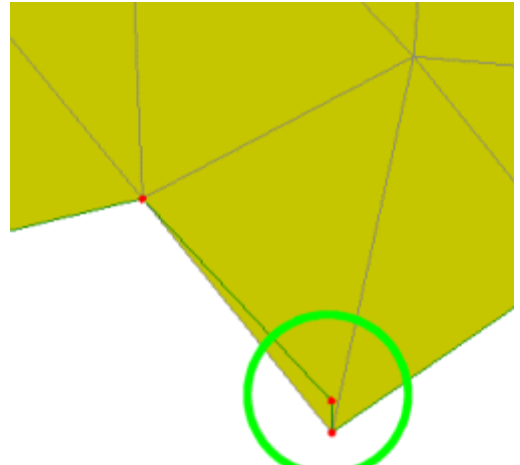
Mesh after edge collapsing (short edge removed)
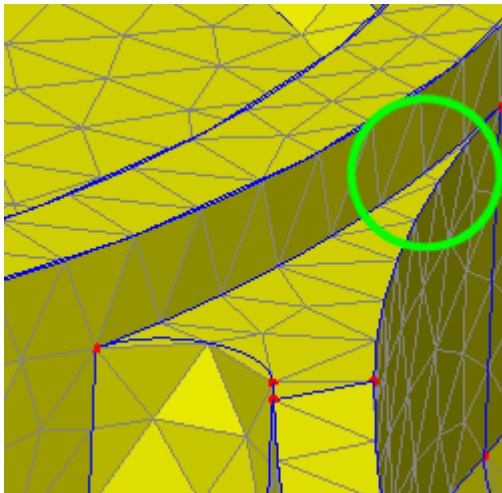
Mesh before edge collapsing
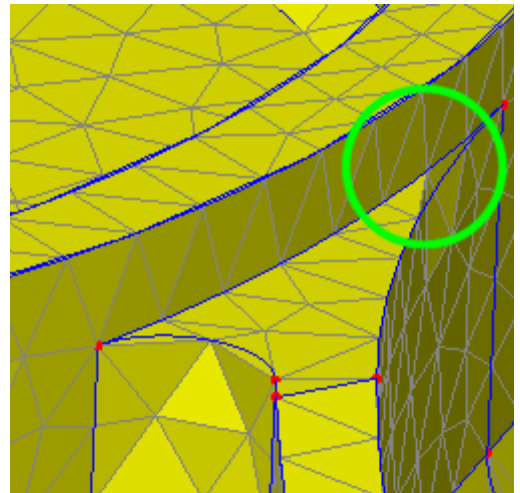
Mesh after edge collapsing

(Elements with small subtended angles removed)

Controlling the Mesh Density

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

Boundary Discretisation

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

Notes

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

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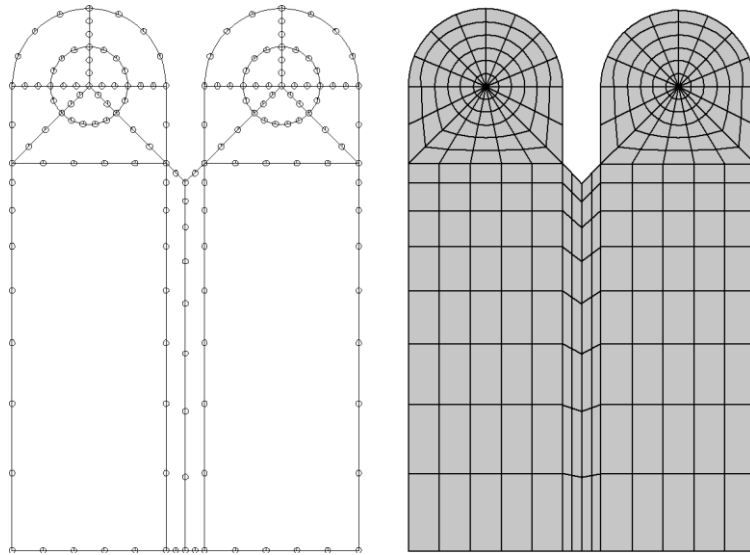
❑ **Irregular Surface Meshing The applied boundary discretisation (top) produces the irregular mesh pattern on the Surface (bottom).**
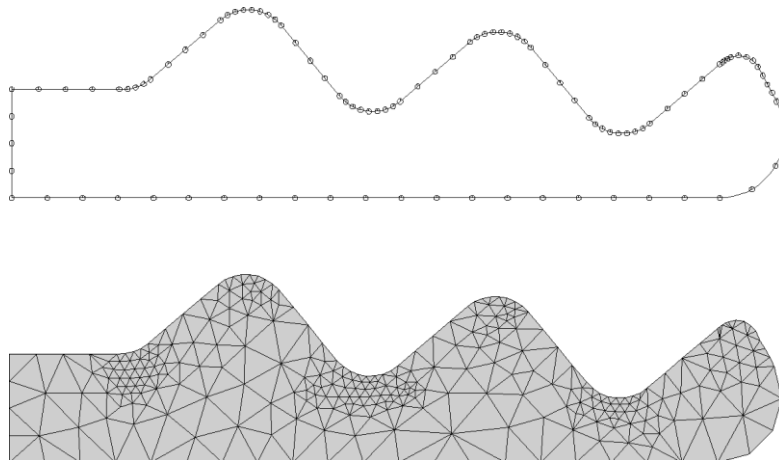
Default Number of Mesh Divisions

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

Background Grid Meshing

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

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

The background grid may be specified explicitly from Points at each vertex or generated automatically. Any mesh distortion required may be entered using the point mesh stretching parameters. If generated automatically, tetrahedral shapes will always be used.
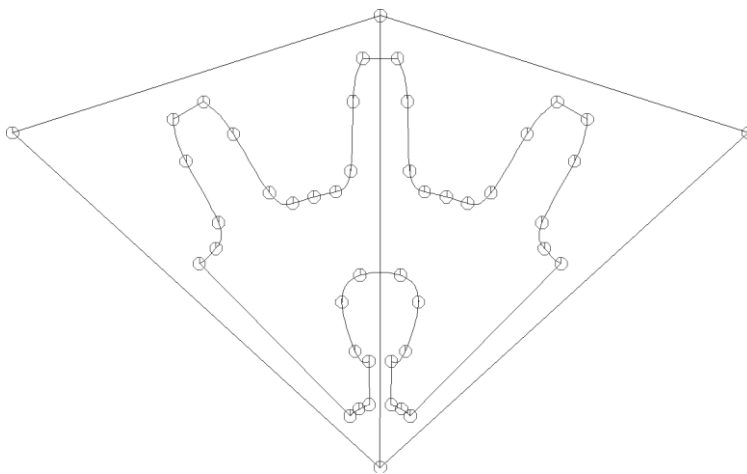
Constant Mesh Spacing	**Varied Mesh Spacing**
Same spacing parameters (Point meshes) are assigned to all Points in background grid.	Different spacing parameters (Point meshes) are assigned to the top Points (spacing=7) and the bottom Points (spacing=1) in the background grid.
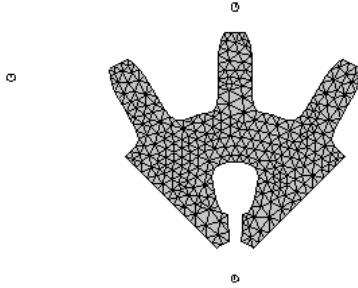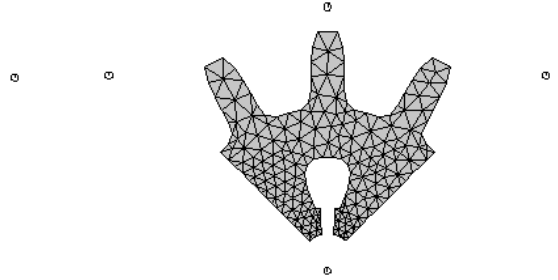
Notes

- 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 1 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.
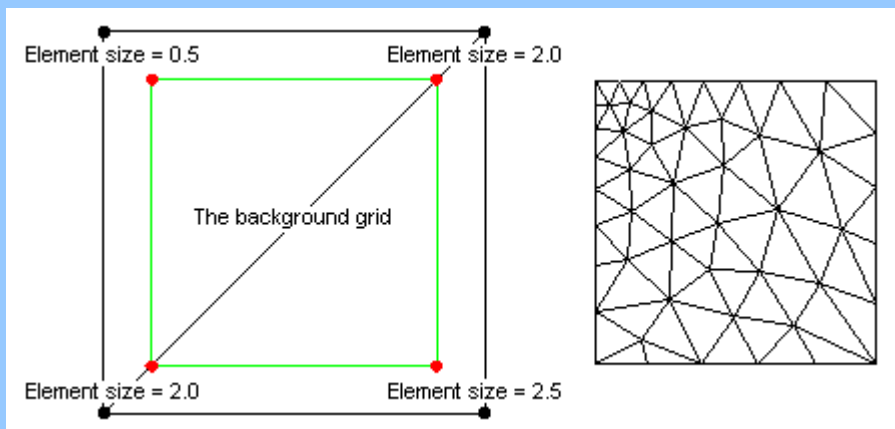
Composite Material Assignment

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.
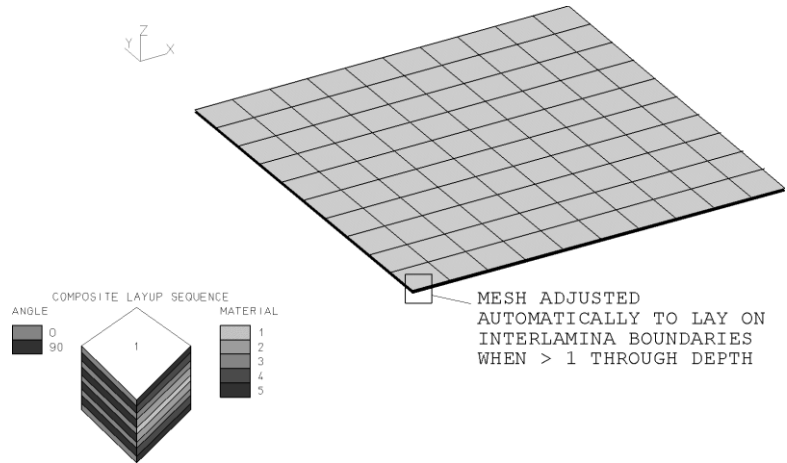
Connecting Beam and Shells

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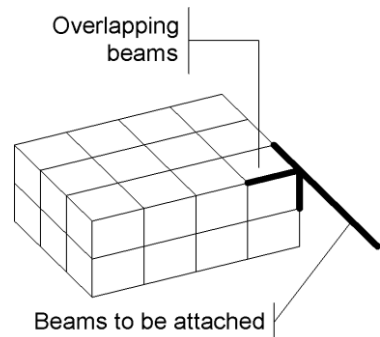
Shell Solid Connectivity

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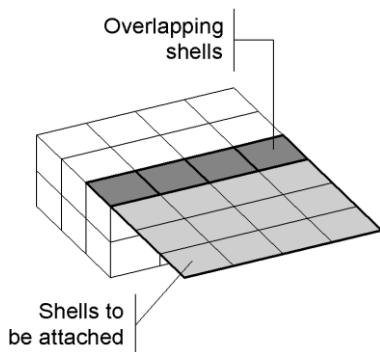
Case Study: Connecting Shells and Solids

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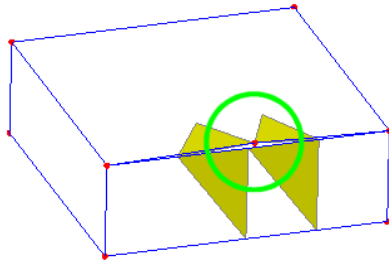
Unmerged nodes on volumes which failed to mesh.
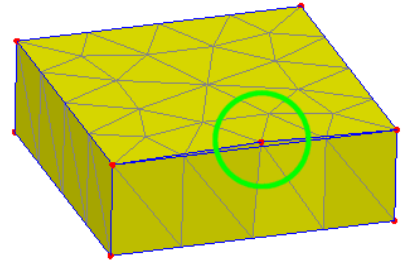
Mesh with nodes merged by adjusting node merge tolerance

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.

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.

Defining and Assigning Joints

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 or 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 to a Single Pair of Features

To model a single joint element between a pair of features (two points, two lines or two surfaces):

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 to Multiple Pairs of Features

To define joint elements between multiple pairs of features (two or more sets of points, lines or surfaces:

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.

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 determining the number of joints generated in the joint interface mesh. 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 Master feature.
- 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
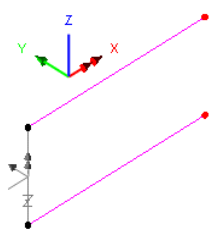
Joint between two points   Joint between two lines   Joint between two surfaces
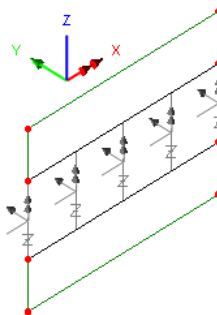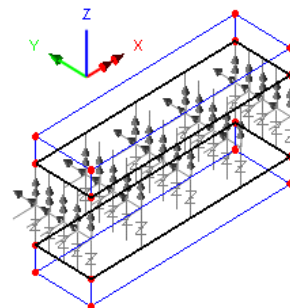
Joint Local Axis Direction

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 to define the xy-plane.
- ❑ **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. Joint elements are orientated from the Master to the Slave. To swap the Master and the Slave 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.
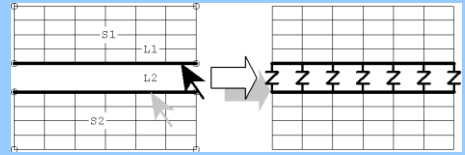
Note. The unmerge facility allows coincident features to be created from a single feature and also allows a feature to be set as Unmergable, so it will not be accidentally merged back with another coincident feature. See **Merging and Unmerging** for more details.

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.
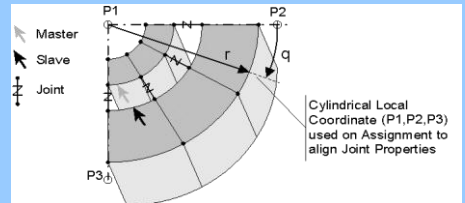
Joint Material and Geometric Properties

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 |
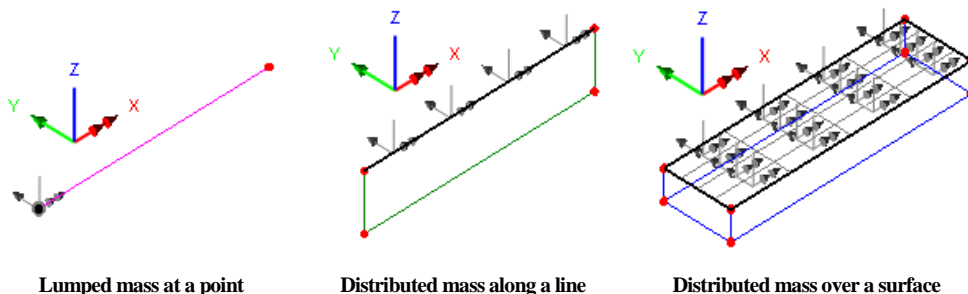
Non-Structural Mass Local Axis Direction

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)
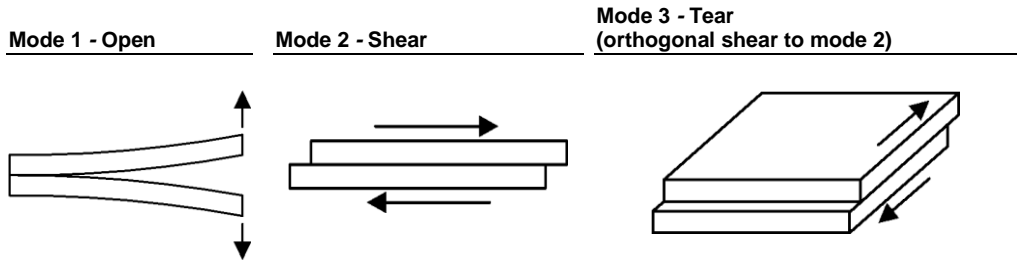
The interface elements are used to model delamination in an incremental nonlinear analysis. These elements have no geometric properties and are assumed to have no thickness.

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.
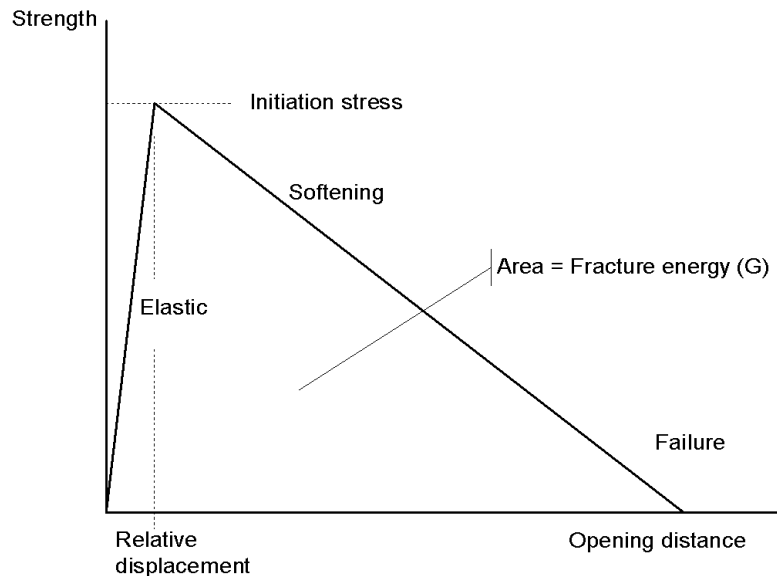
Material Parameters

❑ **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

Elements are classified into groups according to their function. The element groups are listed below.

- See also **Joint Element Meshes**, and **Interface Elements (for composite delamination)**.
- For full details of all elements refer to the *Element Reference Manual*.
- For full details of the element formulations refer to the *LUSAS Theory Manual*.

Point Element Selection

Non structural mass and Joint elements are defined at or between points.

Generic Element Types	2D	3D
Non structural mass	PM2	PM3
Joint (no rotational stiffness)	JNT3	JNT4
Joint (for beams)	JPH3	JSH4
Joint (for grillages)	JF3	
Joint (for axisymmetric solids)	JAX3	
Joint (for axisymmetric shells)	JXS3	

Line Element Selection

The following table lists the elements available for Line meshing by type and by name. The first column matches the option list in the Line mesh dialog box.

Generic Element Types	2D 2 noded	2D 3 noded	3D 2 noded	3D 3 noded
'None'	-	-	-	-
Bar	BAR2	BAR3	BRS2	BRS3
Thin beam	-	BM3	-	**BS4**
Thick beam	BEAM	-	BMS3BTS3	-
Thick beam (nonlinear)	-	-	-	-
Engineering grillage Cross-section beam	GRIL	-	-	-
Semiloof beam	-	BMX3	-	**BSX4**
Axisymmetric membrane	-	-	-	**BSL4**
Joint (no rotational stiffness)	BXM2	BXM3	JNT4	-
Joint (for beams)	JNT3	-	JSH4	-
Joint (for grillages)	JPH3	-	-	-
Joint (for axisymmetric solids)	JF3	-	-	-
Joint (for axisymmetric shells)	JAX3	-	-	-
Thermal bar	JXS3	-	BFS2	-
Axisymmetric thermal membrane	BFD2	BFD3		BFS3
Thermal link	BFX2	BFX3	LFS2	
Interface	LFD2	-	-	-
	-	**IPN6**		-

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.

- No check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, however LUSAS Solver will stop the analysis if the element is unsuitable.

- This list is a guide as to which elements to use. Not all elements are listed here. See the *Element Reference Manual* for full details of all elements.

- Elements in **bold** text are only available if your licence includes the Plus option.

Surface Element Selection

The following table lists the elements available for surface meshing by type and by name. The first column matches the option list in the Surface mesh dialog box.

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	QAX8
Thin plate	TF3	QF4	-	-
Thick plateThin shell	-	QSC4	TTF6	QTF8
Thick shell	TS3	QSI4	**TSL6**	**QSL8**
Membrane	TTS3	QTS4	**TTS6**	**QTS8**
Fourier	TSM3	SMI4	-	-
Plane field (thermal)	**TAX3F**	**QAX4F**	**TAX6F**	**QAX8**
Axisymmetric solid field	TFD3	QFD4	TFD6	QFD8
Explicit dynamic - plane stress	TXF3	QXF4	TXF3	QXF8
Explicit dynamic - plane strain	**TPM3E**	**QPM4E**	-	-
Explicit dynamic - axisymmetric	**TPN3E**	**QPN4E**	-	-
Interface	**TAX3E**	**QAX4E**	-	-
	-	-	-	**IS16**

Notes

- Elements in **bold** text are only available if your licence includes the **Plus** option.

- No check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, however LUSAS Solver will stop the analysis if the element is unsuitable.

- This list is a guide as to which elements to use. Not all elements are listed here. See the *Element Reference Manual* for full details of all elements.

Volume Element Selection

The following table lists the elements available for volume meshing by type and by name. The first column matches the option list in the Volume mesh dialog box.

Generic Element Types	Tetrahedral		Pentahedral		
	4 noded	10 noded	6 noded	12 noded	15 noded
Stress	TH4	**TH10**	PN6		**PN15**
Thermal	TF4	**TF10**	PF6		**PF15**
Explicit dynamic	**TH4E**	-	**PN6E**		
Stress composite	-	-	**PN6L**	**PN12L**	**PN15L**
Thermal composite			**PF6C**	**PF12C**	**PF15C**

	Hexahedral		
Generic Element Types	8 noded	16 noded	20 noded
Stress	HX8M		**HX20**
Thermal	HF8		**HF20**
Explicit Dynamic	**HX8E**		
Stress Composite	**HX8L**	**HX16L**	**HX20L**
Thermal Composite	**HF8C**	**HF16C**	**HF20C**

Notes

- No check is made in LUSAS Modeller as to whether the element type is valid for the analysis being performed, however LUSAS Solver will stop the analysis if the element is unsuitable.

- This list is a guide as to which elements to use. Not all elements are listed here.

- Elements in **bold** text are only available if your licence includes the Plus option.

Geometric Properties

Geometric properties which have not already been defined by the feature geometry need to be specified using geometric attributes. Geometric properties are element dependent and are defined for an element family such as bars, beams, shells, joints etc. Geometric attributes are defined for each feature type using the **Attributes> Geometric** menu item and then **assigned** to the required 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 properties such as cross-sectional area, second moments of area etc for bar/link, grillage,and thin/thick beam elements can be defined either by:

- Using the **Attributes> Geometric> Section Library** menu item to access supplied and user-created items in the **section library**, or, less commonly, by

- Using the **Attributes> Geometric> Line** menu item to enter section properties directly on the Geometric Line dialog.

- Using the **Attributes> Geometric> Tapering Section** menu item to access the geometric line tapering section dialog directly.

- Using the **Attributes> Geometric> Multiple Varying Section** menu item to create a geometric line attribute that contains details of any number of cross-sections held in section libraries 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.

The following options are available when defining geometric line properties:

❑ **Visualise** Cross-sectional shapes for standard library items and for library items created by the standard and arbitrary section property calculators will be automatically visualised. Beam sections defined by using the Attributes > Geometric > Line menu item, requiring general properties to be entered by hand, will require cross-section properties to be defined manually in order for geometric visualisation 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. 3D models normally have the Z axis set to be vertical.

❑ **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. In cases where one or both ends of a beam section have been defined using the arbitrary section calculator (and this includes section 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. Offsets (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. See Notes for details.

❑ **Cross-section...** information is used to define a series of quadrilateral shapes that define the true cross-sectional shape of the bar/grillage/beam element. It generally only needs to be defined if a beam's properties have been defined manually and it is required that the beam's shape is subsequently visualised using the fleshing option. For most other cases cross-sectional information is automatically provided or created by LUSAS. In defining the cross-section shape the coordinate of each quadrilateral must be defined in local zy cross-section coordinate pairs at each node, $z1$, $y1$, $z2$, $y2$, $z3$, $y3$, $z4$, $y4$. When defining a cross-section by this method
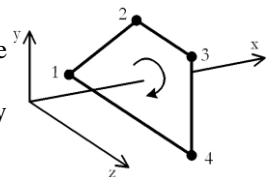
the centroid of the section must reside at 0,0. For Cross Section Beam elements (for advanced use only) the number of integration points (also known as gauss points) can also be set.

❑ **Fibre locations** define positions on 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 local (user) or server (all users) libraries using the arbitrary section property calculator have fibre definitions calculated automatically.

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

Ap - Plastic area (= elastic area)
Zyyp - Plastic modulus for bending about y axis
Zzzp - Plastic modulus for bending about z axis
Zyp - Plastic modulus for torsion about y axis
Zzp - Plastic modulus for torsion about z axis
Sp - Plastic area for shear (Sp=0)

The actual parameters required depend on the chosen beam element. See the *Element Reference Manual* for further details.

Once defined, the geometric section properties are added to the 🔵 Treeview using the **OK** or **Apply** button. The geometric properties can then be **assigned** to the required Line(s) in the model.

Offsets

- Thick beam elements accommodate Offsets which are measured **from** the bending plane **to** the nodal line in the local element direction. When offsets are defined the beam section properties are input relative to the beam axis.

- When sections are defined at either ends of a tapering beam the offset of one section to the other to achieve the vertical and horizontal alignment setting
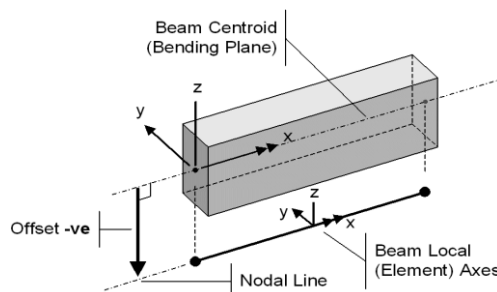
specified is automatically calculated. Subsequent entering of an offset value for the 'master' end in the Value field will automatically offset and update the value for the other 'follower' end by an equivalent amount to ensure the beam ends are moved equally from their nodal positions. If a vertical or horizontal alignment offset is stated in the Alignment panel of the dialog that value will only affect the 'follower' end.

- For thin beam elements eccentricity may be incorporated within the geometric properties. In this case the properties are input relative to the nodal line. This type of eccentricity is used when elements share the nodal line as can occur in the analysis of stiffened shells.

Notes

- The orientation of the beam axes and hence the orientation of the visualised section is governed by the vertical global axis stated on the Direction Definition dialog.

- The orientation of the beam axes and hence the orientation of the visualised section is governed by the vertical global axis stated on the Direction Definition dialog.

- If a thick beam's properties with offsets defined is assigned to a thin beam the 1st and 2nd moments of area will be updated to accommodate the offset as an eccentricity.

- For beams 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 beams 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 beams 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.

- More complex tapering can be achieved using the **Multiple Varying Section** facility.

- Models created prior to version 14.2 will not have any fibre locations data stored for each beam. However, the relevant fibre location data can be added automatically for these models by double-clicking on each Geometric line entry in the 🔗 Treeview and re-selecting the same section size from the appropriate sections library.

- Double-clicking on a geometric line attribute name in the 🔗 Treeview allows editing of beam section information.

Geometric Surface Properties

Geometric attributes are defined for surfaces using the **Attributes> Geometric> Surface** menu item.

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

Eccentricity

❑ Optionally an **Eccentricity** can be specified for certain element types. Eccentricity is measured **from** the bending plane **to** the nodal line in the local element z direction.
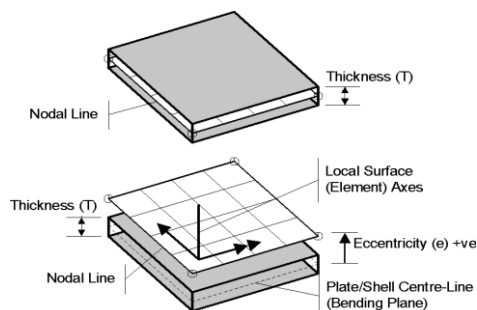
Geometric Joint Properties

Geometric attributes are defined for joints using the **Attributes> Geometric> Joint** menu item. For certain joint elements eccentricity (in the local z direction) is an optional geometric property that can be defined. See the *Solver Reference Manual* for details.

Setting Geometric Attributes for Default Use

A geometric attribute may be designated as the default assignment using **Set Default** on the context menu. When done, default attributes are automatically assigned to new geometry as it is defined.

Visualising Geometric Properties (Fleshing)

Beams of standard or arbitrary cross-section (that are held in section libraries), surface thicknesses and offsets can be visualised on the geometry model using the fleshing button

or from the Attributes Layer 🔗 properties Geometric tab. From the geometric settings dialog or from the context menu of the fleshing button the cross-section may be exaggerated

in size and shrunk in from the ends of the assigned Line to aid visualisation at the connections. When processing results the deformed cross-section shape may also be visualised by selecting the **Deform** option on the geometric properties dialog.
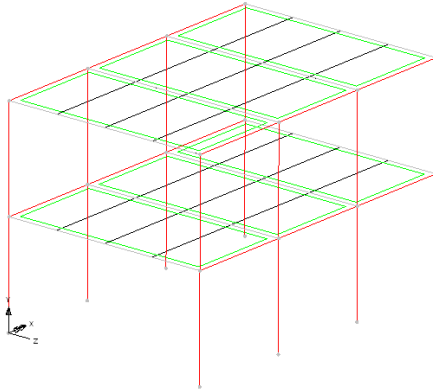
Visualisation of Attributes Without Fleshing   **Visualisation of Attributes With Fleshing**
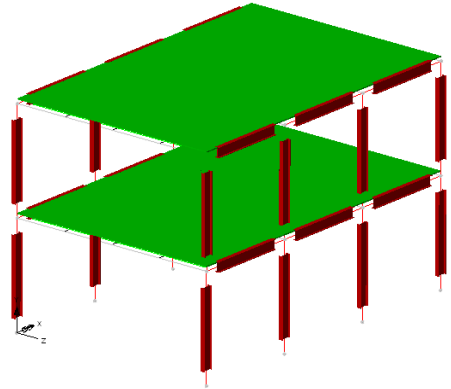
Notes

- Geometric properties can be varied along a line using a tapered section definition or by using the multiple varying section facility.

- Geometric properties can be varied over a surface by using a variation. See *Variations* for more details.

- Geometric attributes are not required for plane strain, axisymmetric solid or 3D solid elements.

- The geometric attributes are specified in a generic form for all elements and only the properties required for the intended element need be specified. For example eccentricity is reported as a error if assigned to semi-loof shell elements which do not use it in their formulation.

- For more details on the properties required for a specific elements refer to the *Element Reference Manual*

Section Library

The section library is available from the **Attributes> Geometric> Section Library** menu item.

Standard section libraries are currently available for the following:

❑ **Australia steel sections**
❑ **Canada steel sections**

❑ **China steel sections**

❑ **EU steel sections**

❑ **KS steel sections - Korean Rail Sections**

❑ **UK steel sections**

❑ **US steel sections**

In addition, user-created section properties can be saved in the following libraries:

❑ **User (local) - for use inside the current project only**

❑ **User (server) - for use across all projects**

Sections selected from a library are added to the 🦠 Treeview. From there they can be assigned to selected line features on a model. For more details on the use of section library items see the **Geometric Properties** section.

Adding Additional Sections to the Section Library

In addition to sections provided in the geometric beam section library, other sections can be added to the library by using Section Property Calculator facilities. These are accessed from the **Utilities > Section Property Calculator** menu. Facilities exist to calculate the properties of standard sections, precast beam sections (with or without a top slab), simple and complex box sections, and user-defined arbitrary sections that are created in LUSAS Modeller.

Multiple Varying Sections

The multiple varying section dialog is accessed using the **Attributes > Geometric > Multiple Varying Section** menu item. It enables pre-defined cross-sections to be specified at distances 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.

❑ **Usage** Selects the element type for which the varying section properties will be defined.

❑ **Specify shape interpolation** Allows selection of an interpolation type. If unchecked, a smoothed option is used.

❑ **Distance interpretation** settings specify the method of spacing the sections.

Section selection

In the section selection table the sections that will be used to generate a varying cross section along a line or path of lines are each added to the table and an interpolation method and a distance from a starting point is specified for each. User-defined sections need to be saved to the local or server libraries prior to using this facility.

❑ **Section** Clicking on the launch dialog button in this cell allows a pre-defined section to be chosen from the section library.

❑ **Shape Interpolation** Clicking on the drop-list button in this cell permits the definition of a smoothed, linear, quadratic or a function-based interpolation setting. The shape interpolation setting defines the shape between adjacent pairs of defined sections. This is only available for second and subsequent entries in the table.

❑ **Distance** specifies a value for a chosen distance type.

Distance type

❑ **Scaled to fit each line individually** Values must be entered in the Distance cells of the table to represent the locations along a line that the sections will apply. Values are entered either as proportional distances along a line (for example entering 0, 10 and 20 would specify a section at either end and at a mid-point of any line that was selected and assigned this geometric line attribute). Distances are mapped to the actual line length so entering 0, 0.5, and 1, in three separate cells would produce the same result. Note that a section does not necessarily have to be defined to start at a distance of 0, so entering 0.5, 10.5, and 20.5, in three separate cells would produce the same result. This latter example would be of particular use in creating preliminary models for eventual staged construction uses.

❑ **Along reference path** For this option, the actual distances must be entered at which each section will be positioned along a reference path. For bridge engineering use this equates to entering 'chainage' values. 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. If a model is defined local to an origin of 0,0,0, the reference path origin (the 'Value of distance at start of path' in the Path Definition dialog) can be defined to start at, say, 100, and then entering 100, 105 and 110 with reference to the path assignment would position sections at the start of, and incrementally along, the lines selected.

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.

Alignment

- ❑ **Align all sections to section (number)** - align all sections with respect to the specified 'master' section number. Use in conjunction with the section visualisation panel.
- ❑ **Vertical and horizontal alignments** - these govern how tops, centres, bottoms and sides of adjacent beam sections are set out relative to the 'master' section. An option to enter user-defined individual offsets is also provided and enables more advanced alignment to be achieved. Use the vertical alignment options in conjunction with the section visualisation panel.

Section offsets 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 offset of each 'follower' section to achieve the desired vertical and horizontal alignment with respect to the 'master' is automatically calculated for each section. The offset values seen for 'follower' sections are made up of a value corresponding to the automatically calculated offset required to achieve correct alignment to the master section plus any additional user-defined individual offset that may have been defined for the 'master' section.

Subsequent updating of an offset value for a 'master' section (by clicking the launch dialog button ⬇ in the Section cell of the table and entering an offset value on the Enter section dialog) when **Top to top**, **Centre to centre**, or **Side to side** alignment options are selected will automatically update the offset 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.

Note that for thick beam elements the offsets are 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 offset 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 offset for the 'master' section need be modified because all other sections will have their offset values updated automatically to be moved by the same amount.
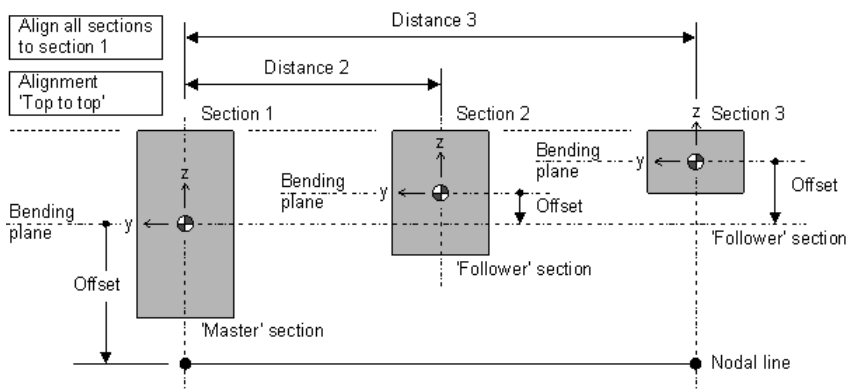
Whilst not commonly used, user-defined individual offsets can be entered for selected sections by selecting the **Individual offsets** menu item from either the Vertical or Horizontal Alignment drop-down menu, then accessing a section's properties by clicking the launch dialog button ⬛ in this cell and entering an offset value on the Enter section dialog. When individual offsets are specified on a 'follower' section any connection with the 'master' section is broken and any offsets specified on 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, or if sections are of completely different shapes) the engineering properties at locations along the multiple varying section can be calculated in two different 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.

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.

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. If section properties of this geometric line are manually edited the automatically added part of the attribute name in the 🔗 Treeview is removed. The automatically created part of the name is uneditable if a rename is carried out.

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.

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. Longitudinal section visualisation only takes place once all required data has been entered and 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 is, and can only be, carried-out when the OK button is clicked.

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
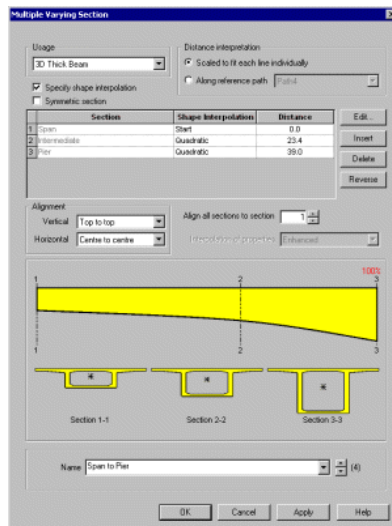
| Varying section distances defined for assignment of the line attribute to a single line. | Varying section distances defined for assignment to multiple lines with reference to a path. |
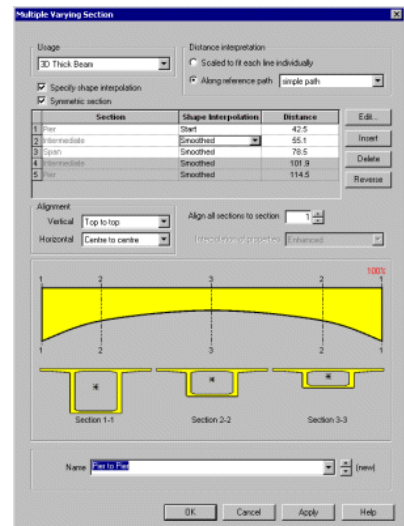
The values entered in the distance cell of the multiple varying section dialog depend upon the intended assignment. Examples of each type follow.

Scaled to fit each line individually

In their simplest form, multiple varying sections can be defined for assigning to single selected lines on a model. Values are entered that will be mapped to the actual line length when the multiple varying section line attribute is assigned to a line or lines on a model. For example, entering 0, 0.333, and 1, or 0, 1 and 3 in three separate cells would specify a section at either end and at a third-point of a selected line that was selected and assigned this geometric line attribute. Note that a section does not necessarily have to be defined to start at a distance of 0, so entering 0.5, 1.5, and 3.5, in three separate cells would produce the same result. This latter example would be of particular use in creating preliminary models for eventual staged construction uses.
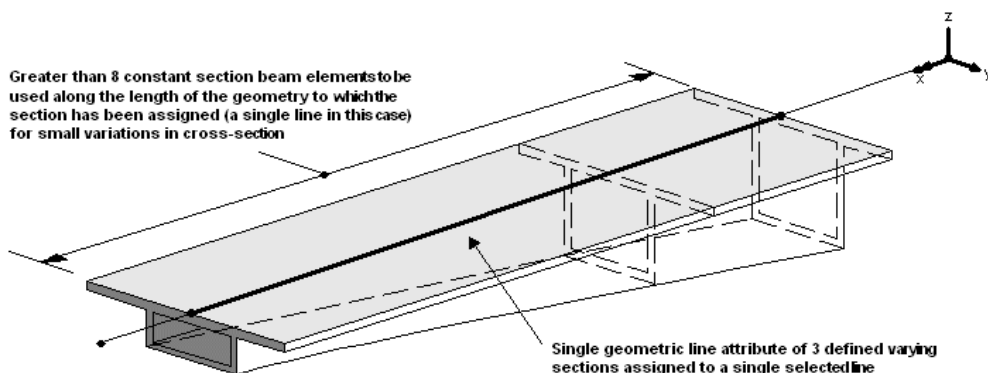
Single line beam assigned a single multiple varying section line attribute
(for clarity beam line has been visualised at top of section)

Along reference path

For assignment of multiple varying sections to multiple lines a reference path method is used. For this option, values are entered which 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.
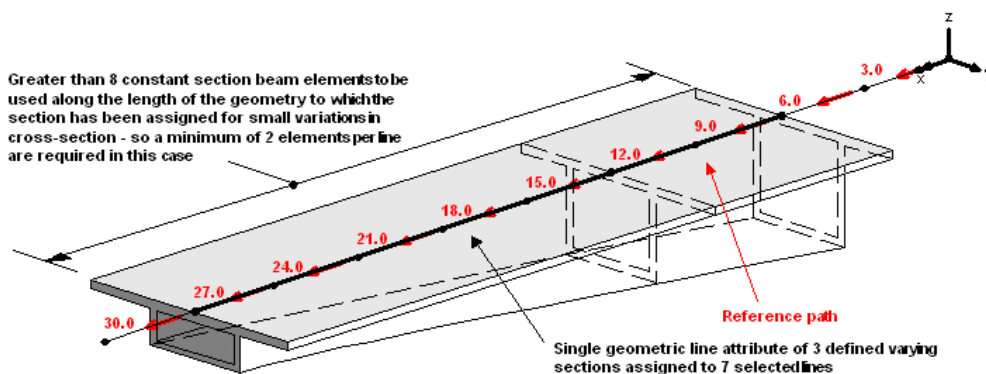
Multiple line beams assigned a single multiple varying section line attribute with reference to a path
(for clarity beam line has been visualised at top of section)

Line mesh density

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.

Using specified distances from a start of a path

Specified distances from a start of a path are used when it is desired to define a whole series of similar sections for either a complete structure or for a particular length of a structure and assign them all to a series of lines on a model with reference to a path. This most powerful of methods can produce models very quickly.

As an example, if a model is defined local to an origin of 0,0,0, the reference path origin can be defined to start at, say, 100, and then by entering distances of 106, 112 and 127 with reference to a path assignment, sections would be positioned at those distances along the lines selected.
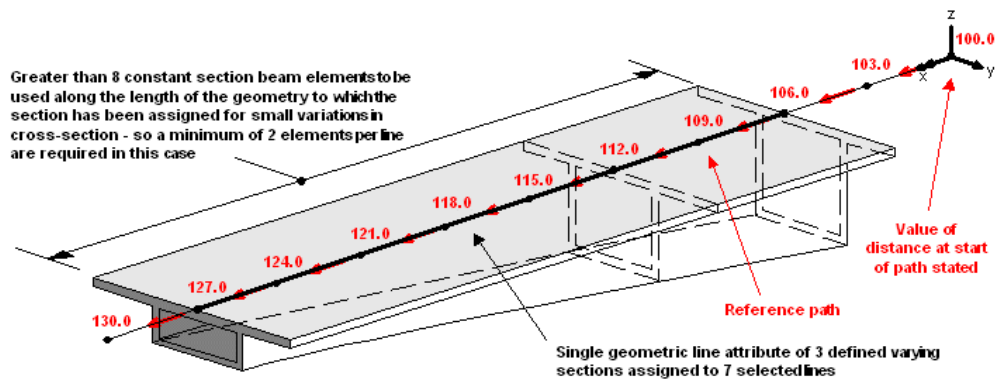
Multiple line beams assigned a single multiple varying section line attribute with reference to a path with starting distance specified
(for clarity beam line has been visualised at top of section)

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.

Modelling examples

Use of the multiple varying section facility enables simple prototype and assessment models of bridges formed of tapered box sections to be created in a very straightforward manner. When combined with the reference path facility more detailed modelling can be done.
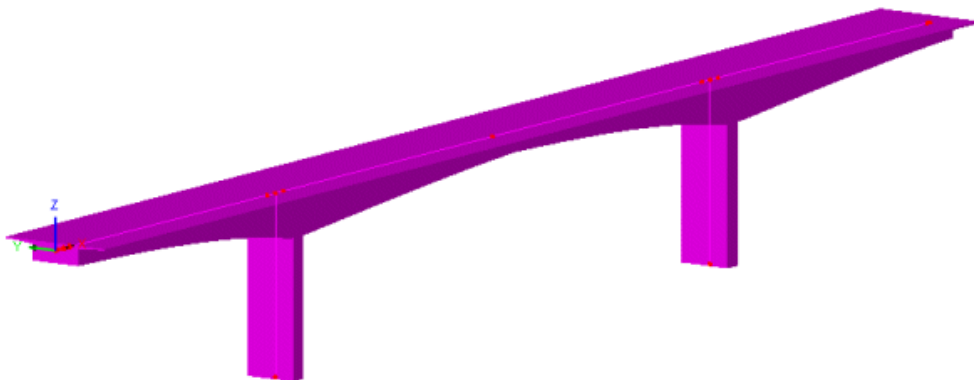
Varying section line attributes assigned to single lines on a model

Use of the multiple varying section with the reference path facility enables detailed models of bridges formed of tapered box sections to be created in a very straightforward manner. If necessary, one geometric multiple varying section line attribute can 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 can be made to override any previously assigned 'temporary' assignments of voided sections that may have been previously made. An example of this follows.
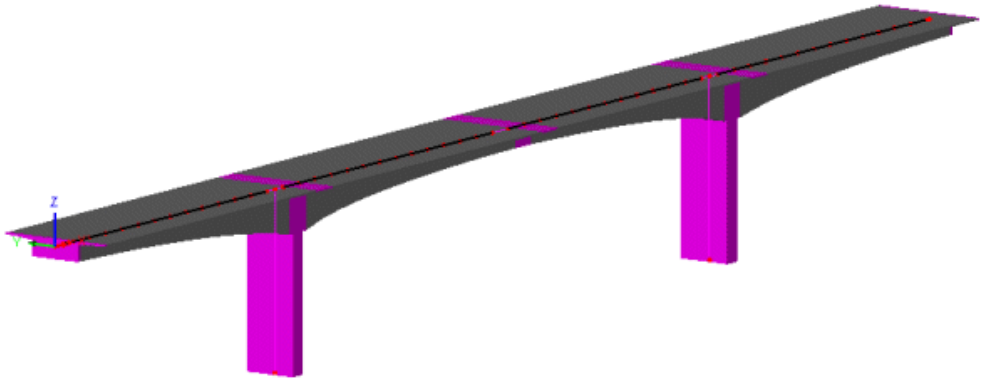
Varying section line attributes (shown selected) assigned to multiple lines on a model with reference to a path

Material Properties

Every part of a **finite element** model must be assigned a material property attribute. Material attributes are defined from the **Attributes> Materials** menu item and then assigned to the required geometry feature (or **mesh object** in a mesh-only model). Note that not all elements accept all material property types. Refer to the *Element Reference Manual* for full details of valid element/material combinations.

Linear and Nonlinear Material Properties

- ❑ **Isotropic/Orthotropic** Defines **linear** elastic or **nonlinear** material properties with options for plasticity, hardening, creep, damage (continuum or composite, viscosity and two-phase materials.
- ❑ **Anisotropic** Different material properties are specified in arbitrary (non-orthotropic) directions by direct specification of the modulus matrix.
- ❑ **Rigidities** Allows direct specification of the material rigidity matrix. In-plane and bending rigidities are defined from prior explicit integration through the element thickness.
- ❑ **Thermal** Applicable to thermal (field) elements only. Whenever thermal elements have been used in a model thermal material properties should be defined and assigned to the relevant parts of the model.
- ❑ **Joint** Linear and nonlinear joint material models for contact and impact analyses using joint elements.

Specialised Material Properties

- ❑ **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.
- ❑ **Generic Polymer** Defines a material model consisting of a number of parallel Maxwell units, an Eyring dashpot and a non-linear spring which can incorporate damage to represent the behaviour of polymer like materials..
- ❑ **CEB-FIP (1990) Concrete Shrinkage and Creep** Defines a material to predict the mean behaviour of a concrete section due to creep effects
- ❑ **Elasto Plastic Interface** Defines a material to represent the friction-contact relationship within planes of weakness between two discrete bodies
- ❑ **Delamination** models for use with the composite delamination interface elements.
- ❑ **Mass** material models for specifying mass in structure using non-structural mass elements.
- ❑ **Nonlinear User** A user defined constitutive model defining the stress from strain.
- ❑ **Resultant User** A user defined constitutive model defining the stress resultants from strain.

Editing of Material Properties

Editing of pre-defined material data (such as that provided in the material library) allows users to view both the original material definition input data, as well as modify the values used. Editing of user-defined material properties only permits viewing and editing of the values used.

Material properties added to the Attributes Treeview have context menu entries named Edit Definition and Edit Attribute.

- Selecting the **Edit Definition...** menu entry or double clicking the attribute displays the original definition dialog with all the original input data intact.
- Selecting the **Edit Attribute...** menu entry displays values that can be modified. For materials added from the material library, these values may be changed but this breaks the link to the original definition dialog and a warning message will be displayed.

Notes

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

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

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

Material Library

The more commonly used structural material properties are defined in the material library which is located under the **Attributes> Material> Material Library** menu item.

The units will default to those chosen on model startup but may be changed if desired. Pick the material required from the drop down list and click **OK** or **Apply** to add the material properties to the 🔬 Treeview. The material properties may then be assigned to the model in the usual way.

Composite Library

The more commonly used structural composite material properties are defined in the composite library which is located under the **Attributes> Material> Composite Library** menu item.

The units will default to those chosen on model startup but may be changed if desired. Pick the composite material required from the drop down list and click **OK** or **Apply** to add the properties to the 🔬 Treeview. The composite material properties may then be used to define a composite stack or be assigned to the model in the usual way.

Isotropic/Orthotropic Material

Isotropic and **orthotropic** material attributes can be used to specify the following material properties.

❑ **Elastic** is used to specify linear elastic material properties including Young's modulus, Poisson's ratio, mass density. Orthotropic material orientation is specified as global, relative to a local coordinate system or relative to the feature local axis system. Optional thermal expansion and dynamic constants can be specified. Note that not all elements accept all the orthotropic models. Refer to the *Element Reference Manual* for full details of valid element/material combinations. Orthotropic models are **Plane stress**, **Plane strain**, **Thick**, **Axisymmetric**, **Solid**.

❑ **Thermal** is used to specify properties for general thermal and heat of hydration analysis. For general thermal analysis phase change state, thermal conductivity, specific heat coefficient, and enthalpy values can be set. For concrete heat of hydration analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, additional thermal options such as exotherm type, cement type and timescale units can be specified.

❑ **Plastic** Used to model ductile yielding of nonlinear elasto-plastic materials such as metals, concrete, soils/rocks/sand.

❑ **Hardening** Used to model a nonlinear hardening curve data. Hardening is defined as part of the plastic properties. Isotropic, Kinematic and Granular sub-types are available. Isotropic hardening can be input in three ways.

❑ **Creep** Used to model the inelastic behaviour that occurs when the relationship between stress and strain is time dependent.

❑ **Damage** Used to model the initiation and growth of cavities and micro-cracks.

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

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

❑ **Two-phase** Required when performing an analysis in which two-phase elements are used to define the drained and undrained state for soil.

Plastic Material Models - *Isotropic*

The following are Isotropic models available from the **Attributes> Material> Isotropic** menu item by choosing the **Plastic** check box on the material attribute dialog.

❑ **Stress Potential**(von Mises) 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. The modified von Mises model allows pressure dependent plasticity to be defined.

❑ **Optimised von Mises**(Model 75) Represents ductile behaviour of materials which exhibit little volumetric strain (for example, metals). Especially for explicit dynamics.

❑ **Tresca** (Model 61) Represents ductile behaviour of materials which exhibit little volumetric strain (for example, metals). Incorporates isotropic hardening.

❑ **Mohr-Coulomb** (Model 65) The non- associated Mohr Coulomb model represents dilatant frictional materials which exhibit increasing shear strength with increasing confining stress (for example, granular materials such as rock and soils). The model incorporates isotropic hardening and dilatancy.

❑ **Drucker-Prager** (Model 64) Represents ductile behaviour of materials which exhibit volumetric plastic strain (for example, granular materials such as concrete, rock and soils). Incorporates isotropic hardening.

❑ **Concrete** (Model 94) A two and three-dimensional concrete material model that accounts for non-linear behaviour in both tension and compression. It is able to model both cracking and crushing behaviour.

❑ **Stress Resultant** (Model 29) May be used for **certain beams and shells**. The model is formulated directly with the beam or shell stress resultants plus geometric properties, therefore it is computationally cheaper.

Plastic Material Models - *Orthotropic*

❑ **Stress Potential** **(Hill and Hoffman models)** These models are available from the **Attributes> Material> Orthotropic** menu item by choosing the **Plastic** check box on the material attribute dialog.

The stress potential model defines nonlinear material properties applicable to a general multi-axial stress state requiring the specification of yield stresses in each direction of the stress space. Incorporates hardening, yield stress and Heat fraction. **Hoffman** is a pressure dependent material model allowing for different properties in tension and compression.

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.

Thermal Material

Thermal material properties are used to define the thermal behaviour of a material when using Thermal (Field) elements. The thermal properties describe the way in which heat flows. Heat may be transferred through conduction, convection or radiation. For linear steady state heat transfer problems only the conductivity needs to be specified.

For materials in which the conductivity is constant in all directions isotropic material input should be used. When the conductivity varies in different directions orthotropic material input should be used. The direction of orthotropy is defined relative to any local coordinate systems.

For transient thermal analysis the specific heat capacity is also required. It should be noted that within LUSAS a specific heat coefficient is used. The specific heat coefficient is computed by multiplying the specific heat capacity by the density.

If phase change is to be modelled the enthalpy must be specified. Two phase changes models are available . When carrying out a phase change it is recommended that lumped specific heat (OPTION 105) is used. This is specified in the **Model Properties> Solution> Element Options** dialog by choosing the lumped mass option.

For heat of hydration analysis, where internal heat is generated by the chemical reaction between cement and water as concrete hardens, additional thermal options such as exotherm type, cement type and timescale units can be set.

LUSAS Solver can model temperature dependent properties but this needs to be defined in the Solver datafile. See the *Solver Reference Manual* for further details.

Stress Potential (von-Mises, Hill, Hoffman)

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.

Material Properties

❑ **Yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.

❑ **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.

Hardening Properties

There are three methods for defining nonlinear hardening. Hardening curves can be defined in terms of either the hardening gradient, the plastic strain or the total strain as follows:

❑ **Hardening gradient vs. Effective plastic strain** Requires specification of gradient and limiting strain values for successive straight line approximations to the stress vs. effective plastic strain curve.

In this case hardening gradient data will be input as (C1, ep1), (C2, ep2) for each straight line segment. LUSAS extrapolates the curve past the last specified point.
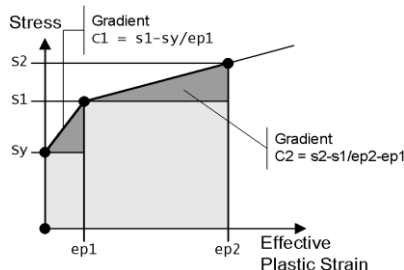
❑ **Uniaxial yield stress vs. Effective plastic strain** Requires input of coordinates at the ends of straight line approximations to the uniaxial yield stress vs. effective plastic strain curve.

For the curve shown here the plastic properties will contain the yield stress (sy) and the hardening data will be input as (s1, ep1), (s2, ep2), etc. LUSAS extrapolates the curve past the last specified point.
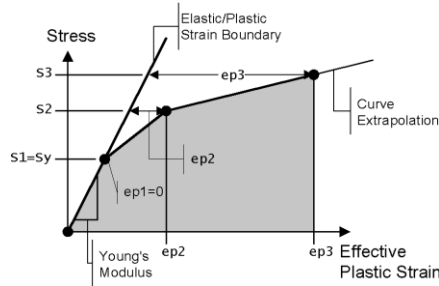
❑ **Uniaxial yield stress vs. Total Strain** Requires input of coordinates at the ends of straight line approximations to the stress strain curve.

Linear properties specify the slope of the stress strain curve up to yield in terms of a Young's modulus. Plastic properties specify the yield stress (sy) and the hardening data is input as a series of coordinates, for example (s1, e1), (s2, e2), etc. LUSAS extrapolates the curve past the last specified point.
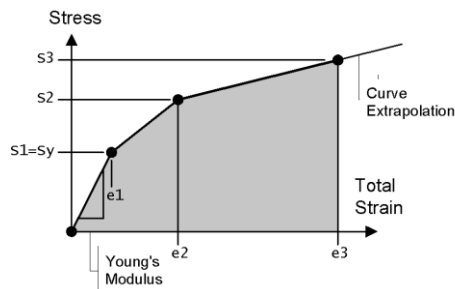
Optimised von Mises (Model 75)

This model represents ductile behaviour of materials that exhibit little volumetric strain (for example, metals). It is especially suitable for explicit dynamics.

Material Parameters

❑ **Yield stress** The level of stress at which a material is said to start unrecoverable or plastic behaviour.

❑ **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat

produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.

Hardening (von Mises)

❑ **Kinematic hardening** Plasticity hardening formulation associated with translation, as opposed to expansion, of the yield surface.

In the optimised implicit model the direction of plastic flow is evaluated from the stress return path. The implicit method allows the proper definition of a tangent stiffness matrix which maintains the quadratic convergence of the Newton-Raphson iteration scheme otherwise lost with the explicit method. This allows larger load steps to be taken with faster convergence. For most applications, the implicit method should be preferred to the explicit method.

The model incorporates linear isotropic and kinematic hardening.
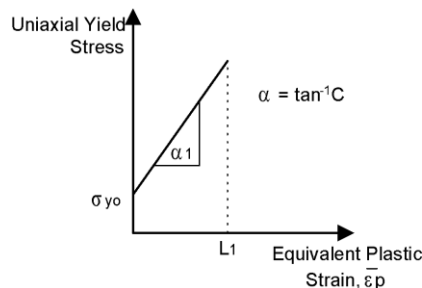
Nonlinear Hardening Curve for the von Mises Yield Model (Model 75)

Tresca (Model 61)

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.
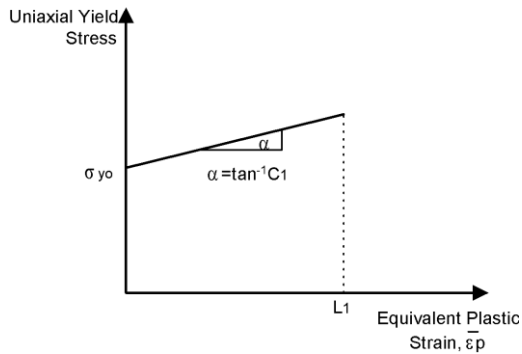
Hardening Curve Definition for the Tresca Yield Model

Non Associated Mohr-Coulomb (Model 65)

The non-associated Mohr-Coulomb elasto-plastic model may be used to represent dilatant frictional materials that exhibit increasing shear strength with increasing confining stress (for example, granular soils or rocks). The model incorporates isotropic hardening and dilatancy.

Material Properties

❑ **Initial Cohesion** defining the degree of granular bond and a measure of the shear strength.
❑ **Friction Angle** defining angle of shearing resistance
❑ **Dilation Angle** defining magnitude of plastic volume strains.
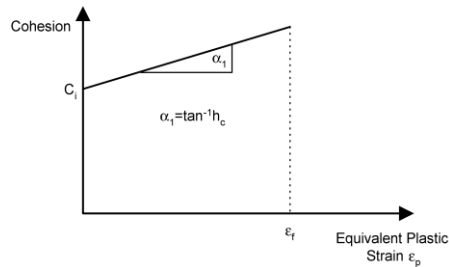
Cohesion Definition for the Non-Associated Mohr-Coulomb Model (Model 65)
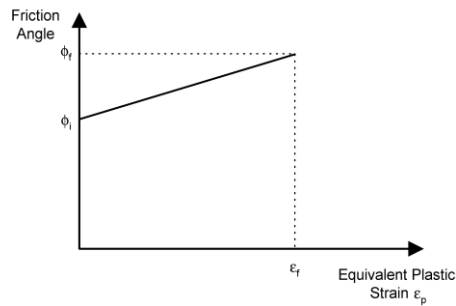
Friction Angle Definition for the Non-Associated Mohr-Coulomb Model (Model 65)

Drucker-Prager (Model 64)

The Drucker-Prager elasto-plastic model (see figures below) may be used to represent the ductile behaviour of materials which exhibit volumetric plastic strain (for example, granular materials such as concrete, rock and soils). The model incorporates isotropic hardening.

Material Properties

❑ **Initial Cohesion** A material property of granular materials, such as soils or rocks, describing the degree of granular bond and a measure of the shear strength. Setting the initial cohesion to zero is not recommended as this could cause numerical instability under certain loading conditions.

❑ **Initial Friction angle** A material property of granular materials, such as cohesive soils and rocks.

❑ **Heat fraction** The fraction of plastic work that is converted into heat energy. Only applicable to temperature dependent materials and coupled analyses where the heat produced due to the rate of generation of plastic work is of interest. The value should be between 0 and 1.

❑ **Slope of Yield Stress** The slope of the uniaxial yield stress against equivalent plastic strain.

❑ **Plastic strain** The limit of equivalent plastic strain up to which the hardening curve is valid.
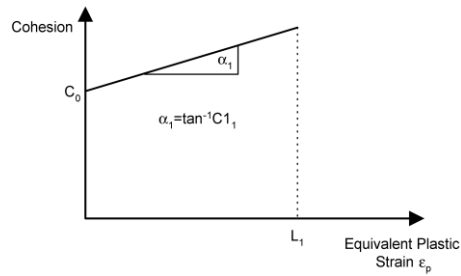
Cohesion Definition for the Drucker-Prager Yield Model (Model 64)
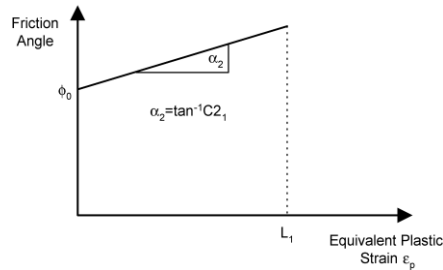
Friction Angle Definition for the Drucker-Prager Yield Model (Model 64)

Multi Crack Concrete (Model 94)

The multi-crack concrete model is a plastic-damage-contact model in which damage planes form according to a principal stress criterion and then develop as embedded rough contact planes. The basic softening curve used in the model may be controlled via a fixed softening curve or 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.

Material Properties

- ❑ Uniaxial compressive strength (f_c) e.g. 40 N/mm^2.
- ❑ Uniaxial tensile strength (f_t) e.g. 3 N/mm^2.
- ❑ Strain at peak uniaxial compression (ε_c) e.g. 0.0022.
- ❑ Strain at effective end of softening curve for distributed fracture (ε_o) e.g. 0.035 or 0 if $G_f > 0$.
- ❑ Fracture energy per unit area (G_f) e.g. 0.1 or 0 if $e_0 > 0$.
- ❑ Biaxial to uniaxial peak principal stress ratio (β_r) e.g. 1.15.

- ❑ Initial relative position of yield surface (Z_o) e.g. 0.6.
- ❑ Dilatancy factor giving plastic potential slope relative to that of yield surface (ψ). e.g. -0.1
- ❑ Constant in interlock state function (m_g) e.g. 0.425
- ❑ Contact multiplier on ε_o for 1st opening stage (m_{hi}) e.g. 0.5.
- ❑ Final contact multiplier on ε_o (m_{ful}) e.g. 5.
- ❑ Shear intercept to tensile strength ratio for local damage surface ($r\sigma$) e.g. 1.25.
- ❑ Slope of friction asymptote for local damage surface (μ) e.g. 1.
- ❑ Angular limit between crack planes (radians) e.g.1.

Explanations for some of these suggested values are stated in the notes that follow.

Notes

- The model can be used with 2D and 3D continuum elements, 2D and 3D explicit dynamics elements, solid composite elements and semiloof or thick shell elements.

- If no data for the strain at peak compressive stress is available it can be estimated see *Solver Reference Manual* for details. As a guide, a reasonable value for most concretes is 0.0022.

- It is important that the initial Young's modulus, E, is consistent with the strain at peak compressive stress, e_c. A reasonable check is to ensure that $E > 1.2\, f_c/\varepsilon_c$

- For concrete that contains reinforcement, distributed fracture will be the dominant fracture state. In this case a value for the strain at the end of the tensile softening curve, ε_o, should be entered and G_f set to zero. If no data is available then a value for ε_o of 0.0035 is reasonable to use for most concretes.

- For unreinforced concrete the strains will tend to localise in crack zones, leading to localised fracture. The value for ε_o must be set to 0.0 and the fracture energy per unit area, G_f, given a positive value. G_f varies with aggregate size but not so much with concrete strength. Typical values for various maximum coarse aggregate sizes are:

  16 mm aggregate: G_f = 0.1 N/mm;
  20 mm aggregate: G_f = 0.13 N/mm;
  32 mm aggregate: G_f = 0.16 N/mm;

Damage Evolution Function (Softening Curve)
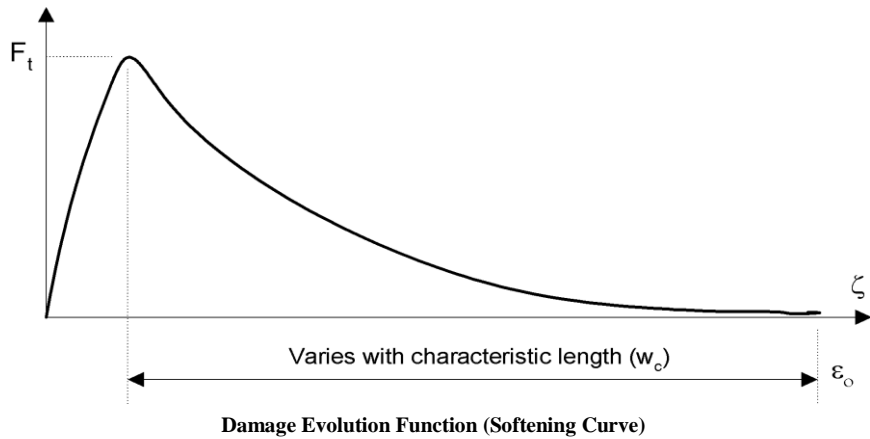
- If the effective end of the softening curve parameter, $\varepsilon 0$, is set to zero, it will be calculated from $e0 \sim 5Gf / Wc\ ft$ where Wc is a characteristic length for the element; if a finite value is given for $e0$, Gf will be ignored.

- The initial position of the yield surface is governed by the value of Z_0. For most situations in which the degree of triaxial confinement is relatively low, a value of between 0.5 and 0.6 is considered appropriate for Z_0 however, for higher confinements a lower value of 0.25 is better.

- The 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 were 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.

- The constant m_g can be obtained from experimental data from tests in which shear is applied to an open crack. The default value for m_g is taken as 0.425 but it is considered that a reasonable range for m_g for normal strength concrete is between 0.3 and 0.6. However, it was found that a low value of 0.3 could lead to second cracks forming at shallow angles to the first, due to the development of relatively large shear forces.

- 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 $e0$) 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.

- A plane of degradation (POD) is formed when the principal stress reaches the fracture stress (f_t); the POD is formed normal to the major principal axis. Thereafter, it is assumed that damage on the plane can occur with both shear and normal strains.
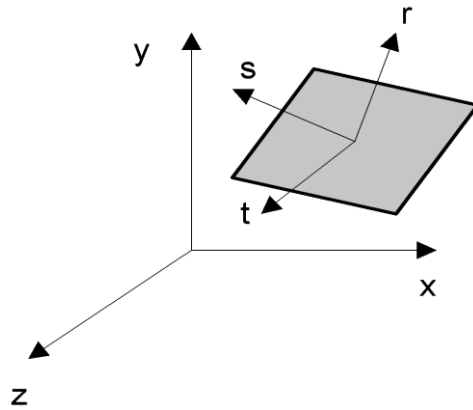
POD Local and Global Coordinate Systems
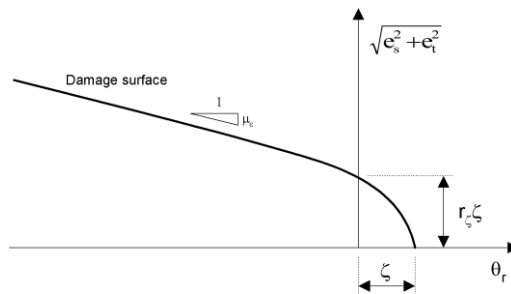
Local Damage Surface

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.

- Fine integration and the non-symmetric solver are always set by default with this material model.

- It is recommended that the following LUSAS options are used with this model:

    252 Suppress pivot warnings.

    62 Allow negative pivots.

See the *Solver Reference Manual* and *Theory Manual* for further details.

Stress Resultant (Model 29)

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.

Creep

Creep is the inelastic behaviour that occurs when the relationship between stress and strain is time dependent. The creep response is usually a function of the stress, strain, time and temperature history. Unlike time independent plasticity where a limited set of yield criteria may be applied to many materials, the creep response differs greatly for different materials.

Creep Properties

Power, exponential and eight parameter uniaxial creep laws are available and a time hardening form is available for each. The power creep law is also available in a strain hardening form. Fully 3D creep strains are computed using the differential of the von Mises or Hill stress potential. A user-definable creep interface is also available which allows a programmable uniaxial creep law.

Stress Potential

The definition of creep properties requires that the shape of the yield surface is defined. The stresses defining the yield surface are specified using the Stress Potential material model.

If a Stress Potential model is used in the Plastic definition then this will override the Creep stress potential and will apply to both the plastic properties and the creep properties. The Creep stress potential is only required when defining linear materials. If a stress potential type is not specified then von Mises is set as default.

None of the stresses defining the stress potential may be set to zero. For example, in a plane stress analysis, the out of plane direct stress must be given a value which physically represents the model to be analysed.

User Supplied Creep Properties

The user creep facility allows user supplied creep laws to be used. This facility provides completely general access to the LUSAS Solver property data input and provides controlled access to the pre- and post-solution constitutive processing and nonlinear state variable output.

Notes

- The user-supplied routine must return the increment in creep strain. Further, if implicit integration is to be used then the variation of the creep strain increment with respect to the equivalent stress, and also with respect to the creep strain increment, must be defined.

- If the function involves time dependent state variables they must be integrated in the user-supplied routine.

- If both plasticity and creep are defined for a material, the creep strains will be processed during the plastic strain update. Stresses in the user routine may therefore exceed the yield stress.

- User-supplied creep laws may be used as part of a composite element material assembly.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Damage

Damage is assumed to occur in a material by the initiation and growth of cavities and micro-cracks. The damage models allow parameters to be defined which control the initiation of damage and post-damage behaviour. Damage models are available for continuum and composite elements.

Continuum Damage Models

In LUSAS a scalar damage variable is used in the degradation of the elastic modulus matrix. This means that the effect of damage is considered to be non-directional or isotropic. Two LUSAS damage models are available (Simo and Oliver) together with a facility for a user-supplied model.

A damage analysis can be carried out using any of the elastic material models and the following nonlinear models:

❑ **von Mises**
❑ **Hill**
❑ **Hoffman**

Note. **Creep material properties** may be included in a damage analysis. See the *Solver Reference Manual* for further details.

Composite Matrix Failure Model

The composite matrix failure model simulates matrix failure using the Hashin damage criteria. The model can only be used with composite solid elements. It is defined under the **Damage** tab on the Material Orthotropic attribute dialog.

Material Properties

❑ **Ply tensile strength in fibre direction**

❑ **Ply compressive strength in fibre direction**

❑ **Ply shear strength measured from a cross ply laminate**

❑ **Ply transverse tensile strength (normal to fibre direction)**

❑ **Ply transverse compressive strength**

See the *Solver Reference Manual* for further details.

Viscoelastic

Viscoelasticity can be coupled with the linear elastic and non-linear plasticity, (isotropic or orthotropic), creep and damage models available in LUSAS. The model restricts the viscoelastic effects to the deviatoric component of the material response. This enables the viscoelastic material behaviour to be represented by a shear modulus G_v and a decay constant β. Viscoelasticity imposed in this way acts like a spring-damper in parallel with the elastic-plastic, damage and creep response. Coupling of the viscoelastic and the existing nonlinear material behaviour enables hysteresis effects to be modelled.

User Supplied Visco Elastic Properties

The user supplied viscoelastic properties facility enables routines for implementing a user supplied viscoelastic model to be invoked. This facility provides completely general access to the LUSAS Solver property data input via this data section and provides controlled access to the pre- and post-solution constitutive processing and nonlinear state variable output via these user supplied routines.

Notes

- When viscoelastic properties are coupled with a nonlinear material model it is assumed that the resulting viscoelastic stresses play no part in causing the material to yield and no part in any damage or creep calculations. Consequently the viscoelastic stresses are stored separately and deducted from the total stress vector at each iteration prior to any plasticity, creep or damage computations. Note that this applies to both implicit and explicit integration of the creep equations.

- Nonlinear Control must always be specified when viscoelastic properties are assigned. In addition Dynamic or Viscous Control must also be specified to provide a time step increment for use in the viscoelastic constitutive equations. If no time control is used the viscoelastic properties will be ignored.

See *Solver Reference Manual* for further details.

Shrinkage Properties

The cure of concrete and thermoset resins is accompanied by isotropic shrinkage which in the case of concrete depends on time, temperature and other environmental factors whilst for thermoset resins the shrinkage is normally described with respect to the degree of cure.

The shrinkage implementation in LUSAS allows an irreversible reduction in volume with time to be modelled. The shrinkage of concrete is accommodated using the equations defined in the design code CEB-FIP90 and also using a more general routine in which shrinkage is defined using a piecewise linear curve. In the general case, shrinkage can be defined as a function of time or degree of cure. A user facility is also available if required.

See the *Solver Reference Manual* for further details.

Two-Phase

Two-phase material properties are required when performing an analysis in which two-phase elements are used to define a drained and undrained state for soil.

Notes

- Usually, the value of Bulk modulus of solid phase is quite large compared to Bulk modulus of fluid phase and not readily available to the user. If Bulk modulus of solid phase is input as 0, LUSAS assumes an incompressible solid phase. Bulk modulus of fluid phase is more obtainable, e.g. for water Bulk modulus of fluid phase = 2200 MPa [N1].

- Two-phase material properties can only be assigned to two phase elements.

- When performing a linear consolidation analysis transient control must be specified.

- Two-phase material properties may be combined with any other material properties together with creep, damage and viscoelastic properties if required.

See *Solver Reference Manual* for further details.

Rubber

Rubber materials maintain a linear relationship between stress and strain up to very large strains (typically 0.1 - 0.2). The behaviour after the proportional limit is exceeded depends on the type of rubber (see diagram below). Some kinds of soft rubber continue to stretch enormously without failure. The material eventually offers increasing resistance to the load, however, and the stress-strain curve turns markedly upward prior to failure. Rubber is, therefore, an exceptional material in that it remains elastic far beyond the proportional limit.
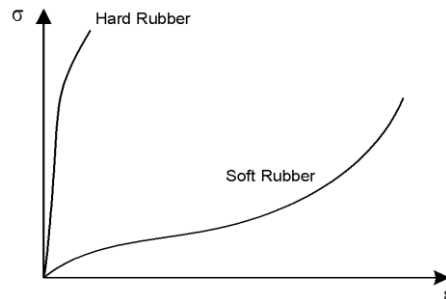
Rubber materials are also practically incompressible, that is, they retain their original volume under deformation. This is equivalent to specifying a Poisson's ratio approaching 0.5.

The strain measure used in LUSAS to model rubber deformation is termed a **stretch** and is measured in general terms as:

$$\lambda = \text{dnew/dold}$$

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/V0.

Pressure - Logarithm of Relative Volume Curve
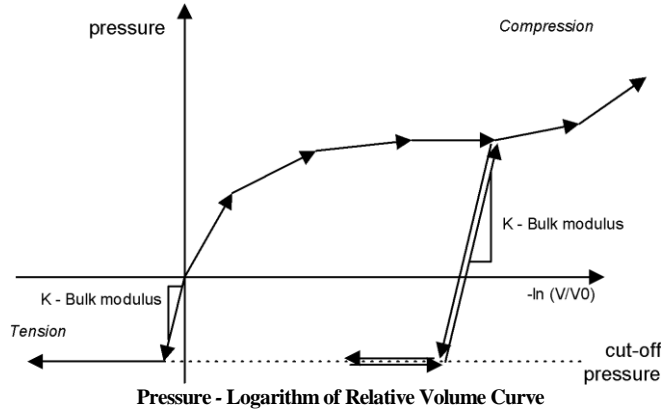
From this figure, it can also be seen that the material model permits two different unloading characteristics volumetrically.

❑ **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.
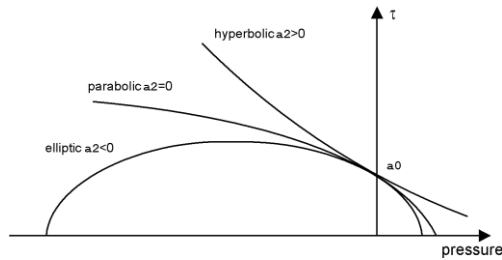
Yield Surface Representation For Different Pressure Dependent Yield Stress Values

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: $K = \dfrac{E}{3(1-2\nu)}$

- **Shear modulus** The relationship between the elastic modulus values in shear, G, and tension, E, assuming small strain conditions, is given by: $G = \dfrac{E}{2(1+\nu)}$

- **Heat fraction coefficient** Represents the fraction of plastic work which is converted to heat and takes a value between 0 and 1.

- **Cut-off pressure** Should be negative (i.e. a tensile value).

- **Pressure dependent yield stresses** (`a0`, `a1`, `a2`) (Should be positive). The yield surface defined is quadratic with respect to the pressure variable. Therefore it can take on different conical forms (see 2nd figure), either elliptic (`a2<0`), parabolic (`a2=0`) or hyperbolic (`a2>0`). The parabolic form is comparable to the modified von Mises material model while the elliptic form can be considered to be a simplification of critical state soil and clay material behaviour.

- The **volumetric crushing** indicator effectively defines the unloading behaviour of the material. If there is no volumetric crushing, the same pressure-logarithm of relative volume curve is used in loading and unloading and if volumetric crushing takes place the alternative unloading/reloading curve is used (see 1st figure).

- **Log relative volume** Natural logarithm (loge, not log10) of relative volume coordinate for ith point on the pressure-logarithm of relative volume curve (see 1st figure)

- The **pressure-logarithm of relative volume** curve is defined in the compression regime hence logarithms of relative volume must all be zero or negative and the pressure coordinates must all be zero or positive.

Generic Polymer with Damage (Model 89)

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 materials. 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.

Material Properties

- ❑ **Eyring damper activation energy.**
- ❑ **Eyring damper activation volume.**
- ❑ **Mass density.**
- ❑ **Linear Spring stiffness.**
- ❑ **Bulk Modulus.**
- ❑ **Maxwell element spring constant.**
- ❑ **Maxwell element Newtonian dashpot viscous parameter.**
- ❑ **Damage properties.**

See the *Theory Manual* for further details.

Concrete Creep and Shrinkage CEB-FIP (Model 86)

Concrete material properties to CEB-FIP Model Code 90 are defined from the **Attributes> Material> Specialised** menu item. This 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.

Material Properties

The following material parameters are required:

- ❑ **Young's Modulus at 28 days**
- ❑ **Poisson's Ratio**
- ❑ **Mass density**
- ❑ **Coefficient of thermal expansion**
- ❑ **Mean compressive strength at 28 days**
- ❑ **Cement type**
- ❑ **Relative humidity**
- ❑ **Nominal size**

Notes

- The CEB-FIP Code states that the modulus of elasticity at 28 days may be estimated from

$$E_{ci} = 2.15x \ 10^4 x \ [f_{cm} \ / \ 10]^{1/3}$$

- The cement type can be **Slow Hardening** , **Normal or Rapid Hardening** or **Rapid Hardening High Strength**. **Normal or Rapid Hardening** is the default option.

- The **Nominal Size** is computed as 2A/u where A 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. Nominal Size should be compatible with the units chosen for the model as an automatic data conversion will be performed based on the units in use.

- 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. It must be noted that it may be difficult to establish an appropriate value of "nominal size" for anything other than beam elements.

- The CEB-FIP creep and shrinkage model must be run in a transient nonlinear analysis in which the time step and total response time are specified in *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.

See the *Solver Reference Manual* and the *Theory Manual* for further details.

Elasto-Plastic Interface (Model 26, 27)

An elasto-plastic model for representing the friction-contact relationship between two discrete bodies. The model is embedded in the plane membrane, plane strain and solid elements to reproduce the nonlinear response of a system containing planes of weakness using Mohr Coulomb type laws.

See the *Theory Manual* for further details.

Delamination Interface (Model 25)

Delamination properties are assigned to interface elements to model delamination between elements. The model behaves linearly until the threshold strength is exceeded. Linear strain softening behaviour then occurs until the fracture energy is exceeded. Once the fracture energy is exceeded further straining occurs without resistance.

Fracture Modes

A 2D model has two fracture modes, a 3D model has three. The fracture modes are:

❑ **Mode 1** Opening

❑ **Mode 2** Shearing

❑ **Mode 3** Tearing (Shear orthogonal to mode 2)

Material Parameters

❑ **Fracture energy** Measured values for each fracture mode depending on the material being used, i.e. carbon fibre, glass fibre.

❑ **Initiation Stress** The tension threshold /interface strength is the stress at which delamination is initiated. This should be a good estimate of the actual delamination tensile strength but, for many problems, the precise value has little effect on the computed response. If convergence difficulties arise it may be necessary to reduce the threshold values to obtain a solution.

❑ **Relative displacement** The maximum relative displacement is used to define the stiffness of the interface before failure. Provided it is sufficiently small to simulate an initially very stiff interface it will have little effect.

❑ **Coupled** The model used for coupling the failure modes (Coupled, Uncoupled Reversible, Uncoupled Origin).

See the *Theory Manual* for more details.

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.

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

Used to specify user material parameters for the user defined nonlinear material model.

See the *Solver Reference Manual* for further details.

Joint Properties

Joint material models are used in conjunction with joint elements to define the material properties for linear and nonlinear joint models. See **Joint and Interface Elements** for information about using 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** full joint properties of spring stiffness, mass, coefficient of linear expansion and damping factor.

Nonlinear Joint Models

❑ **Elasto-Plastic uniform tension and compression** with isotropic hardening. Equal tension and compression yield conditions are assumed.

❑ **Elasto-Plastic General** with isotropic hardening. Unequal tension and compression yield conditions are assumed.

❑ **Smooth Contact** with an initial gap. See notes below.

❑ **Frictional Contact** with an initial gap. See notes below.

❑ **Nonlinear user-defined** joint model.

Seismic Isolator Joint Models

❑ **Viscous dampers** - Kelvin and Four Parameter Solid modules available.

❑ **Lead Rubber Bearings** with plastic yield and biaxial hysteric behaviour.

❑ **Friction Pendulum System** with pressure and velocity dependent friction coefficient and biaxial hysteretic behaviour.

Elasto-Plastic Joint Models

Smooth Contact

+ : tension
- : compression

F

+κ strain hardening stiffness

+Yield force

K - elastic spring stiffness

$\varepsilon = \delta 2 - \delta 1$

-Yield force

-κ
strain hardening stiffness

F

κ - lift-off stiffness

Lift-off force

Gap

$\varepsilon = \delta 2 - \delta 1$
$\psi = \theta 2 - \theta 1$

K - contact spring stiffness
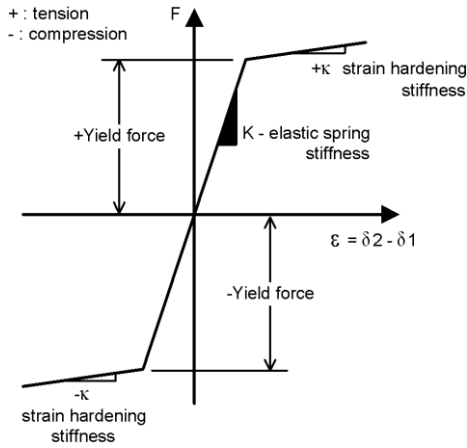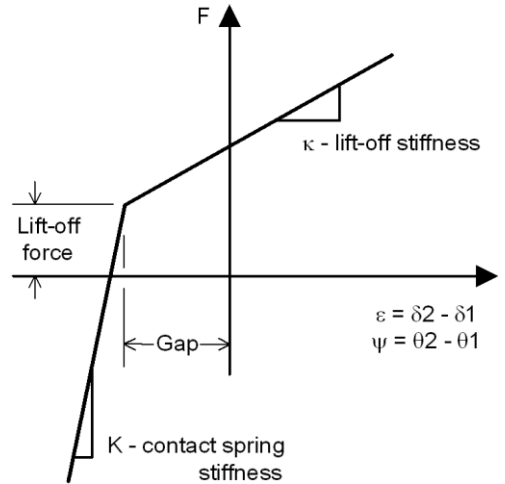
Frictional Contact 1

Frictional Contact 2

Fx

Gap

$\varepsilon xx = \delta x2 - \delta x1$

K - contact spring stiffness

Fy or Fz

Fo

Kcy or Kcz - contact spring stiffness

$\gamma xy = \delta y2 - \delta y1$
or
$\gamma xz = \delta z2 - \delta z1$

-Fo
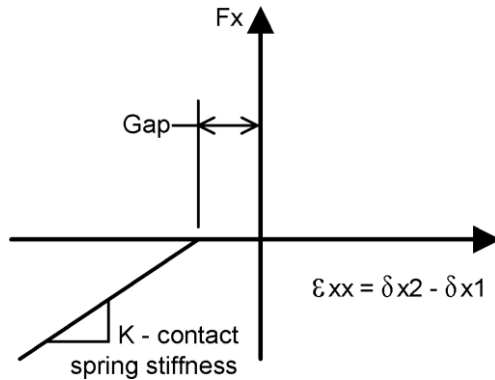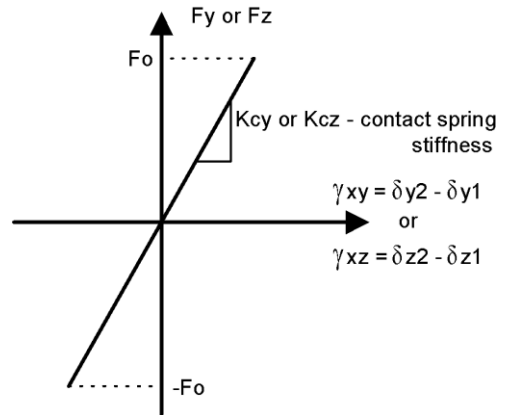
Frictional Contact 3

Fo

μ - coeff. of friction

Fx
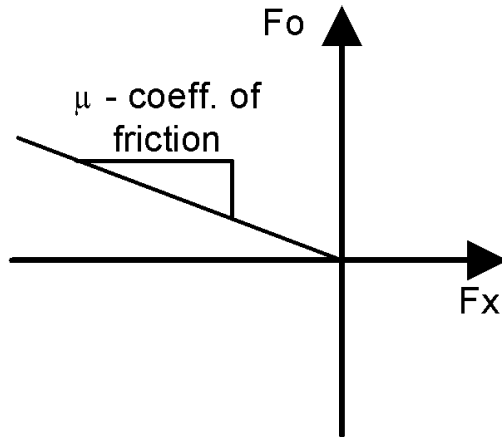
Viscoelastic Damper Joint Model

K_3 C

$\bar{F}$ K_1 K_2 F
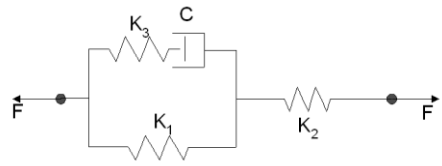
Lead Rubber Bearing Joint

Force

κ - post-yield stiffness

F_{yld}

Displacement

K - elastic spring stiffness
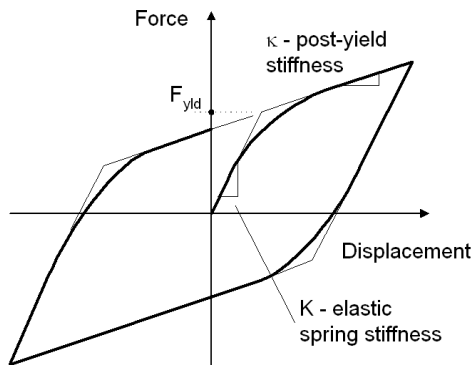
Friction Pendulum System Joint

Force

μN

Displacement
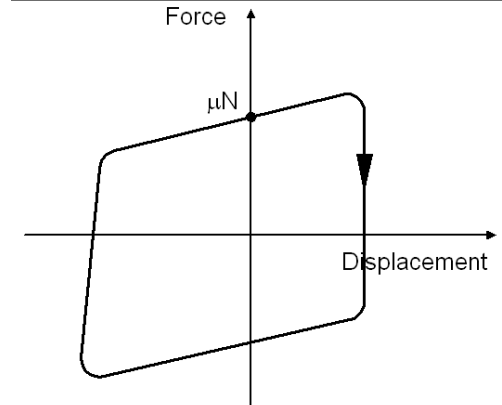
Notes

- For a full description of the joint material input parameters required for these joint models please refer to the *Solver Reference Manual*.

- When defining joint properties for single joint elements the total stiffness or yield force should be defined. When using interface joint meshing the stiffness and yield force defined in the joint properties should be defined per unit length when using interface joints assigned to lines or per unit area when using interface joints assigned to surfaces.

- Initial gaps are measured in units of length for translational freedoms and in radians for rotational freedoms.

- **Smooth Contact**. If an initial gap is used in a spring, then the positive local axis for this spring must go from node 1 to 2. If nodes 1 and 2 are coincident the relative displacement of the nodes in a local direction (d2- d1) must be negative to close an initial gap in that direction.

- **Frictional Contact** If an initial gap is used in a spring, then the positive local x axis for this spring must go from node 1 to 2. If nodes 1 and 2 are coincident the relative displacement of the nodes in the local x direction ($\delta x2 - \delta x1$) must be negative to close an initial gap.

- Both **Smooth Contact** and **Frictional Contact** joints can be used for lift-off or hook contact by using appropriate stiffnesses, gap and yield force.

Support Conditions

General

Support conditions describe the way in which the model is restrained. A support attribute contains information about the restraints to be applied to each degree of freedom. There are three valid support conditions:

- ❏ **Free (F)** the degree of freedom is completely free to move. This is the default.
- ❏ **Fixed (R)** the degree of freedom is completely restrained from movement.
- ❏ **Spring Stiffness (S)** the degree of freedom is subjected to a specified spring stiffness. 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**. Alternatively, **per unit length** or **per unit area** values can be applied.

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).

Using Support Conditions

Support attributes are defined from the **Attributes** menu and **assigned** in the same way as other attributes, by dragging a defined attribute from the 🔗 Treeview onto previously

selected geometric features (or to **mesh objects** in a mesh-only model). 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 is specified when carrying out the assignment.
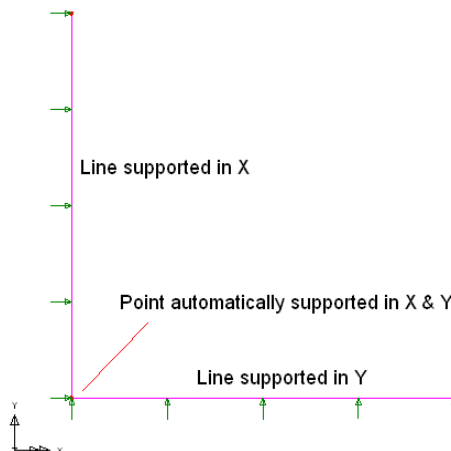
Notes

- On the common feature where support assignments meet, the support condition applied is additive.

- Support assignments on lower order features override those on higher order features.

- Support conditions cannot be changed for different loadcases in a linear analysis.

- Support conditions may only be reassigned on a new increment of a nonlinear problem or a new time step of a transient problem. All supports assignments from previous increments or time steps which are not reassigned will remain unchanged.

- Ensure that nodes are not free to rotate when attached to beam elements with free ends. For example, node 1 in the diagram shown must be restrained against rotation as well as displacement otherwise the element will be free to rotate as a rigid body.

- Support conditions may be omitted for eigenvalue analyses provided a shift is used in the **eigenvalue control**.

- 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.
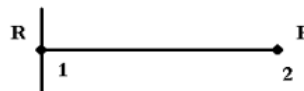
Tip. Supports which act in tension, but not in compression, may be modelled using **joints or interface meshes**.

Visualising Support Conditions

For a support condition to be visualised the model must have been meshed. 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.

❑ **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.

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.

- Support visualisation can be toggled on and off using the support visualisation button .

Example: Translational Fixed and Spring Supports

This 3D structure is restrained from any lateral movement at the base of all legs. The same points are also sprung vertically to represent a non-rigid base support.
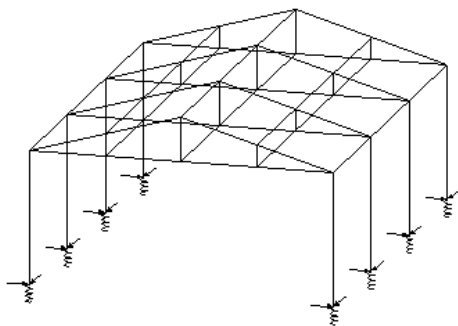
Example: Translational Supports

This 2D structure is restrained horizontally and vertically at the left edge with a single restraint in both in-plane translational directions. This rigidly fixes the body along the edge shown while allowing the rest of the model to move.
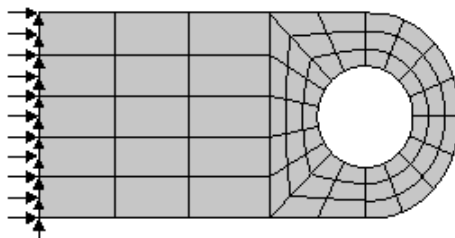
Example: Symmetry

Only the right half of this structure is modelled using shell elements but the full structure is represented by assuming symmetry at the centre-line. Symmetry assumes the same behaviour for both sides of the model therefore a translational restraint is applied to stop movement across the symmetry boundary and a rotational restraint is applied to force zero rotation at the boundary.
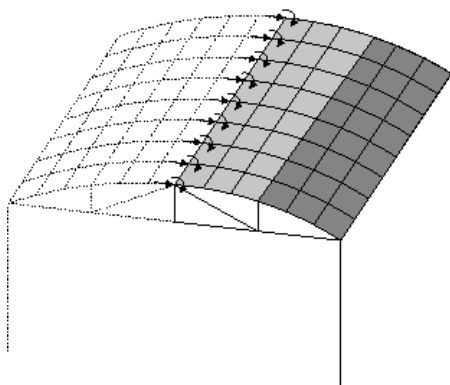
Example: Symmetry

This quarter plate membrane model uses symmetry restraints to effectively model the whole plate. Supports are positioned in order to prevent any movement at lines of symmetry.
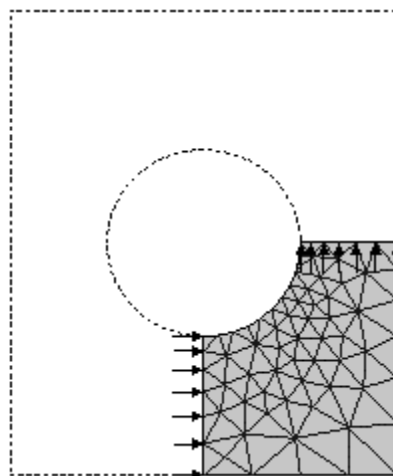
Loading Attributes

Loading attributes describe the external influences to which the model is subjected. Structural and Thermal loading options are provided on separate menu options according to the user interface in use. A summary of the loading types is given here:

❑ **Structural** Concentrated, body force, distributed, face, temperature, stress/strain, and beam loads.

❑ **Prescribed** used to specify initial displacements, velocity or acceleration at a node. (Note that you can also specify prescribed temperatures).

❑ **Discrete** 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** loads to describe the temperature or heat input to a thermal analysis.

Structural, Prescribed and Thermal loads are feature based loads that are assigned to the model geometry and are effective over the whole of the feature to which they are assigned. Discrete loads are feature independent. Further control over how discrete loads are applied is available by using a **Search area**.

Assigning Loading

Loads are **assigned** in the same way as other attributes, by dragging a defined attribute from the 🍃 Treeview onto previously selected features (or to **mesh objects** in a mesh-only model). When a load is assigned, a loadcase and a factor may 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. Additional options are available when applying discrete loads see **Assigning Discrete Loads**.

Some loads act in global directions, others in local element directions. The defined loading value will be assigned as a constant value to all of the nodes/elements in the feature unless a **variation** is applied. Variations can be applied to all feature load types except for Beam Distributed loads that have a variation built into the definition.

Tip. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

Notes

- Load visualisation can be toggled on and off using the [icon] load visualisation button.

- Load factors of assigned loads can be changed by selecting the Change load factor menu item accessed from the loading name context menu.

- In nonlinear and transient analysis feature based loads can be factored using **load curves**.

- Consult the *Element Reference Manual* in order to check that the required loading is available for that particular element

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 previously defined variation. For information on which load types can be applied to which element types, see the *Element Reference Manual.*

Concentrated Load (CL)

A **Concentrated Load** defines concentrated force and moment loads in global (or transformed) directions.

A Concentrated Load is applied per node of the underlying feature onto which the load attribute is assigned. A Concentrated Load is therefore normally only used to assign a load to a point as the total applied load would otherwise be dependent on the mesh density of each feature assigned to.

Concentrated loads are defined relative to the nodal coordinate system. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

Concentrated loads can be applied in cylindrical coordinates for Fourier elements by setting the option on the **Attributes** tab of the **Model Properties** dialog.
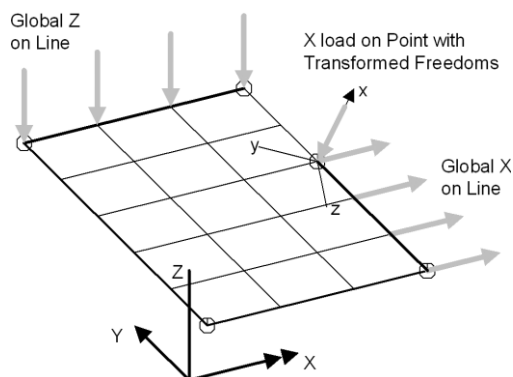
Body Force (CBF)

A **Body Force** defines an acceleration or force per unit volume loading in global directions. A typical example of body force loading is self weight, which requires the specification of gravitational acceleration and mass density (in the material properties).

By default, Body Forces define accelerations, but an option on the **Attributes** tab of the **Model Properties** dialog, can be set so that Body Forces define a force per unit volume.
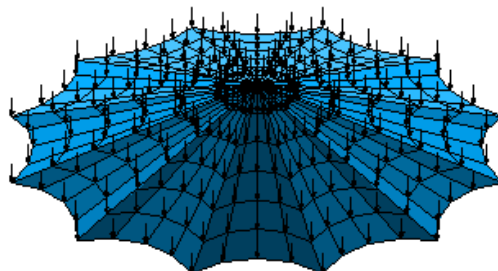
Note that gravity loading can be defined either by directly specifying a constant body force load or by defining its existence as a property of a structural loadcase.

Global Distributed Load (CL)

Defines concentrated force or moment loads in global (or transformed nodal) axis directions. Concentrated force loads are applied to all nodes underlying the feature onto which the load attribute is assigned. Nodal freedoms can be transformed using local coordinate sets. The following sub-types are supported:

❑ **Total** applies nodal load values calculated according to contributions from surrounding elements and to element nodal weighting values, e.g. loads are weighted with ratios 1:4:1 at nodes along the edge of a quadratic shell in such a way as to make the shell strain equally.

❑ **Line (per unit length)** applies nodal loads using the specified values per unit length loads. Must be assigned to Lines.

❑ **Surface (per unit area)** applies nodal loads using the specified values per unit area loads. Must be assigned to Surfaces.

Face Load (FLD)

Defines face traction values and normal loading applied in local element face directions. Face loads are applied to the edges of plane elements or the faces of solid elements. This type of loading is applicable to 2D and 3D continuum elements, and certain shell, membrane and thermal elements.

In the example shown, a local y direction structural face load is assigned to the Surface boundary Lines. Note the direction of the axes of the local element faces.
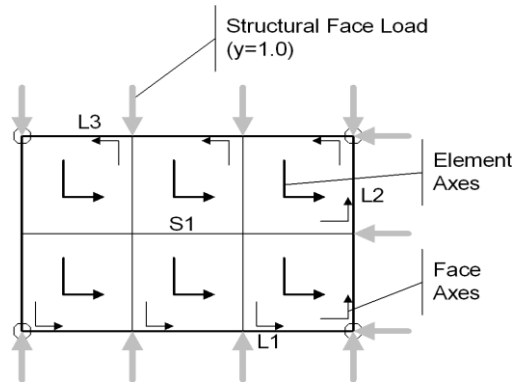
Where a loaded Line or Surface feature is common to two or more higher order features, it is possible to specify to which higher order feature elements the load is assigned.

See the *Element Reference Manual* for details of element face directions.

Local Distributed Load (UDL)

Defines a load per unit length or area for line or surface elements in the local element directions. Typically, local distributed loading is applied to beam elements and shell faces. An example of a local distributed load is internal pressure loading. For beam elements, when the element type permits, uniformly Distributed Load will be written to the LUSAS data file as Beam Element Loading (ELDS).

Temperature Load (TEMP, TMPE)

Nodal and **Element** temperatures define the LUSAS Solver TEMP and TMPE load types respectively. These loads apply temperature differences on a nodal and element basis. Temperature gradients in X, Y and Z directions may also be input. This load type can be used in conjunction with temperature dependent material properties to activate a different set of properties at a specified point in the analysis. The thermal expansion coefficient is normally set to zero in this case.

Notes

- Nodal temperatures apply to all elements connected to that node, except joints, in which temperature loading is invoked using Option 119.

- Elemental temperature is only applied to the node of the element specified.

- For step by step problems, the (initial) temperature values need only be specified on the first load step.

- The Temperature load may be used to provide a temperature field for computing initial material properties in a nonlinear analysis. To initialise the temperature field in a nonlinear field analysis, the temperature loading must be applied using a manual loading increment.

Case Study. Temperature Gradient Through Slab Thickness

Nodal and element temperature values accept gradient values for some element types. This gradient applies a differential thermal load across the top and bottom surfaces of a Surface element. The effect of this gradient is to cause bending in the structure. See the *Element Reference Manual* for temperature load input variations on an element basis.
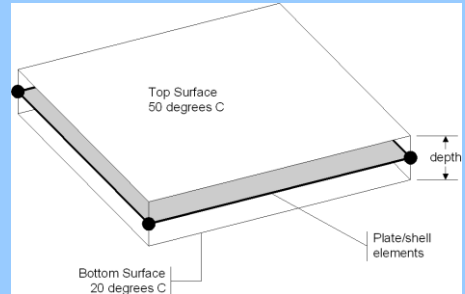
In this example (which assumes no slab eccentricity) a 0.5m thick concrete slab is at 20 degrees Celcius. The top surface is subjected to a temperature of 50 degrees Celsius and the bottom surface remains at 20 degrees Celsius.

To model this enter the following on the structural temperature loading dialog:

- The final slab mid-surface temperature of **(50+20)/2** should be entered in the **Final temperature** field
- The temperature gradient through the slab of dT/dZ should be entered as **(50-20)/0.5** in the **Final Z temperature gradient** field
- The initial slab mid-surface temperature of **20** is entered in the **Initial temperature** field

When the analysis is run, LUSAS will multiply the temperature gradient by the thermal coefficient of expansion specified in the material property attribute to calculate the thermal bending strain.

This method assumes a linear temperature distribution through the depth of the slab. If a known nonlinear variation is required, solid elements must be used with a **variation** defining the nonlinear through-thickness behaviour.

Stress and Strain (SSI, SSR)

The input values that are required in order to define stress and strain loading for particular elements can be seen by selecting the either the element description or by entering the element name in the Stress and Strain loading dialog.

❑ **Initial** defines an element initial stress/strain state in local directions. Initial stresses and strains are applied as the first loadcase and subsequently included into the incremental solution scheme for nonlinear problems. Initial stresses and strains are only applicable to numerically integrated elements.

❑ **Residual** defines element residual stress levels in local directions. This can only be done for elements with a nonlinear capability. Residual stresses (unlike initial

stresses) are assumed to be in equilibrium with the undeformed geometry and are not treated as a loadcase as such. They are considered as a starting position for stress for a nonlinear analysis. Failure to ensure that the residual stresses are in equilibrium will result in an incorrect solution. There is no concept of residual strains and therefore when the residual button is chosen a reduced number of components are presented.

Refer to the individual element descriptions in the *Element Reference Manual* for full details of the initial stress and strain, and residual stress components.

LUSAS Modeller will automatically write an appropriate initial stress and strain, or residual stress type to the datafile when a solve is requested. See the *Solver Manual* for more detailed information regarding the tabulation of initial stresses and strains, and residual stresses in LUSAS datafiles.

Internal Beam Loads (ELDS)

The **Internal Beam Point** load is a point load applied to lines in the local or global direction. The distance may be defined as either parametric (0 to 1) or actual distance. The distance from the start of the line to the point load is defined along with the point load values. Several point loads may be defined in one load attribute if required.
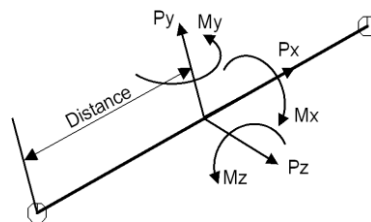
The **Internal Beam Distributed** load is a distributed load applied to lines in the local, global or projected direction. For local and global loading the distance maybe defined as either parametric (0 to 1) or actual distance. Only actual distance is permissible for projected loading. The distance 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. Several distributed loads may be defined in one load attribute if required.
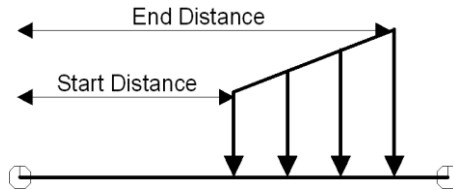
See the *Element Reference Manual* for details of internal beam loading (ELDS) and elements which support this loading type.

Initial Velocity / Initial Acceleration (VELO/ACCE)

In dynamic analyses, velocities or accelerations at a nodal variable can be defined. These values can be used to specify an initial starting condition or they may be prescribed for the whole analysis. If values are to be prescribed throughout the analysis load curves must be used and the appropriate freedoms must be restrained.

Prescribed Loads

For information on which load types can be applied to which element types, see the *LUSAS Element Library*.

Prescribed Displacement

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 automatically be **restrained**.

This 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.
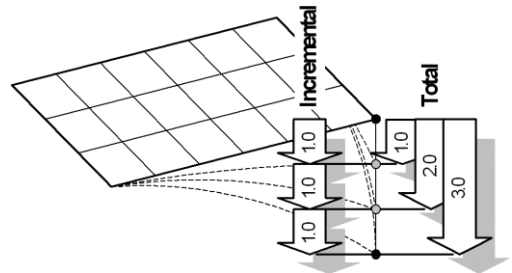
Notes

- 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.

- 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.

Prescribed Velocity and Acceleration

In dynamic analyses, velocities and accelerations may be defined for any nodal variable. These values can be used to specify an initial starting condition or prescribed for the whole analysis.

- ❑ A prescribed, or initial, **Velocity** defines a velocity loading in global (or transformed) directions.
- ❑ A prescribed, or initial, **Acceleration** defines an acceleration loading in global (or transformed) directions. If acceleration loads are required, the density must be specified in the material properties. Initial accelerations are only valid for implicit dynamic analyses.

Notes

- If the values are to be prescribed throughout the analysis load curves must be used, see **Load Curve Definition**.

- Initial velocities and accelerations should only be applied to the first loadcase.

- In general, load curves should be used to prescribe velocities and accelerations in an analysis. However, initial values may be defined without using load curves if no other load type is controlled by a load curve.

- If velocities and accelerations are prescribed for the same variable at the same point in time in an analysis, the acceleration will overwrite the velocity and a warning will be output. An exception to this rule occurs for implicit dynamic analyses where an initial velocity and acceleration may be used to define an initial condition for the same variable.

- If initial conditions are to be applied, refer to **Transient Dynamic Analysis** for details on how to compute the data input required for the appropriate integration scheme.

Discrete Loads

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, and may be effective over full or partial areas of the model. Discrete loads may be projected over an area, onto Lines or into Volumes. Examples of discrete loads that are created automatically by LUSAS include those created for vehicle and lane loading, and equivalent nodal loading defined as a result of using a Prestress Wizard. Separate discrete loads may be applied to a model as a set or load train using the Compound load option. To identify critical vehicle loading patterns on bridges **vehicle load optimisation** is available for Bridge and Civil & Structural 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 may be spread or skewed across several features. LUSAS automatically calculates the nodal distribution of forces that is equivalent to the total patch load. This example shows a typical set of point loads assigned to a grillage model. A single point, a group of 6 and a group of 16 point loads are shown.
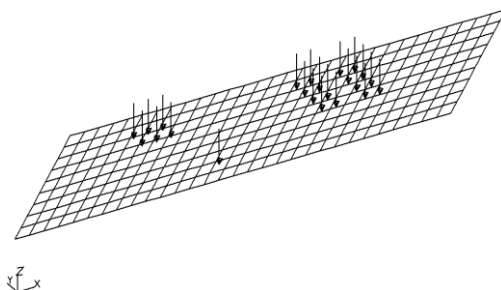
The coordinates of the vertices defining the patch are relative to the Point to which the patch load is assigned, i.e. a load definition is defined in a local coordinate system, the origin of which is given by the coordinates of the Point to which the load is assigned. The Point does

not have to lie on the Surface to which the load will be applied as the patch load is projected in a specified direction.

Using Search Areas with Discrete Loads

A discrete load is distributed onto the elements over which the load lies. A **Search Area** is a way of controlling the load distribution onto these elements. If no search area is specified when assigning the load, then all of the underlying elements will be eligible for load distribution.

Notes

- While projecting the loads into the search area a check is made for multiple intersections of the load and the search area. Multiple intersections indicate an ambiguity in the location of the load. This ambiguity may be resolved with a more specific search area.

- The distribution of load to the nodes follows the shape functions of the particular element. In quadratic elements, this distribution can appear at first unlikely. For example, a unit positive load at the centre of an 8-noded quadratic element, results in negative 0.25 loads at the corners and positive 0.5 loads at the mid-side nodes.

- Search areas are automatically created and used by the **prestress wizards** to define the target to be loaded.

Discrete Load Types

A discrete load consists of coordinates defining the local x, y and z position and a load intensity. Any Points selected when the Discrete loading dialog is initiated are entered as coordinates. Discrete load types available are **Point load**, **Patch load** and **Compound load.**

Point Load

Defines a general set of discrete 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.
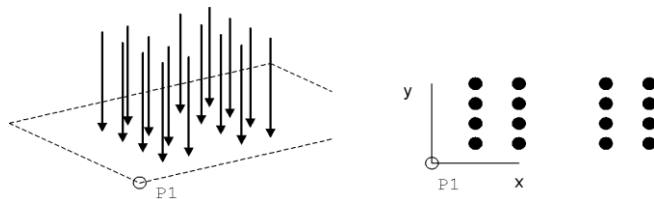
Patch Load

The number of coordinates given dictates the shape of the patch.

- 2 specified coordinates indicates a straight line load.
- 3 specified coordinates indicates a curved line load.
- 4 specified coordinates indicates a straight sided quadrilateral
- 8 specified coordinates indicates a curved sided quadrilateral

The following examples show patch loads assigned to Point 1. Once assigned, the load origin is located at Point 1.

Example 1.

A straight line load defined using 2 coordinates.
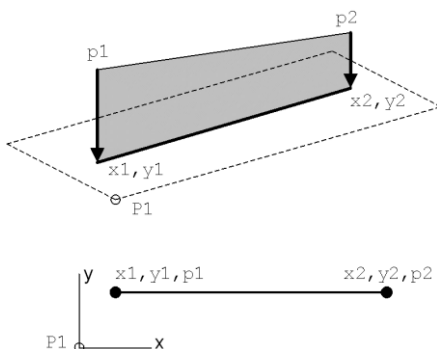
Example 2.

A curved line load is defined using 3 coordinates.
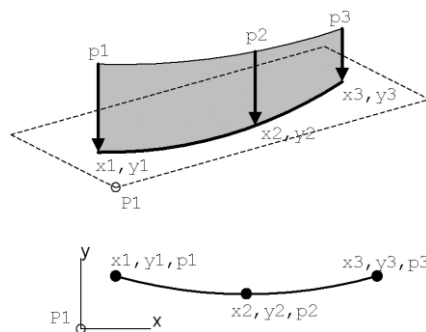
Example 3.

A straight sided quadrilateral defined using 4 coordinates.
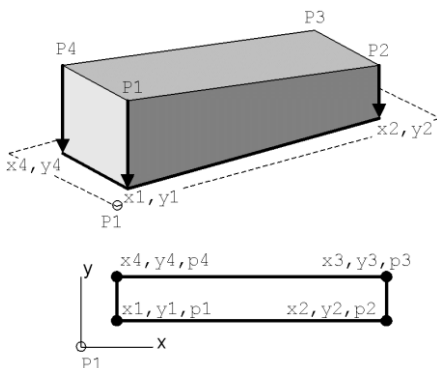
Example 4.

A curved sided quadrilateral defined using 8 coordinates.
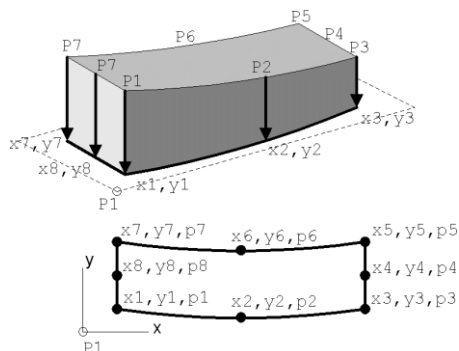
Note that the mid-side nodes for a curved line load and for a curved sided quadrilateral load must lie with +/- 10% of the overall distance between the corner nodes.

Compound load

Compound loads may be created to simplify the definition and assignment of more complex loads. Compound loads form a set, or load train, of previously defined discrete loads that are subsequently assigned to a model as one loading. A compound discrete load may be defined from any combination of existing point, patch and compound discrete loads. For example a patch load representing a truck may be included in a compound load twice and by specifying the distance between the trucks a simple load train is created. Additionally, the same load may also be used any number of times to define the same compound load. 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.

Defining Discrete Point and Patch Loads

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 load types include 8 node, 4 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 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.
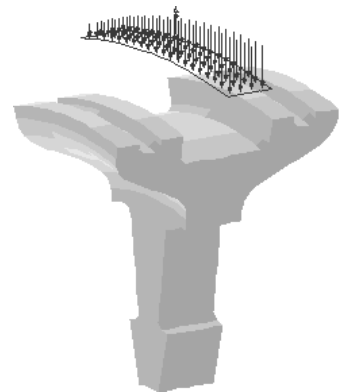
❑ **(Untransformed) Load Direction** defines the direction of the loads in the patch before any transformation is carried out at the assignment stage. Options are: **X**, **Y**, **Z** or **Surface normal**.

In this example, 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.

This example shows a typical 3D patch load where the patch is defined in space and projected onto the mode
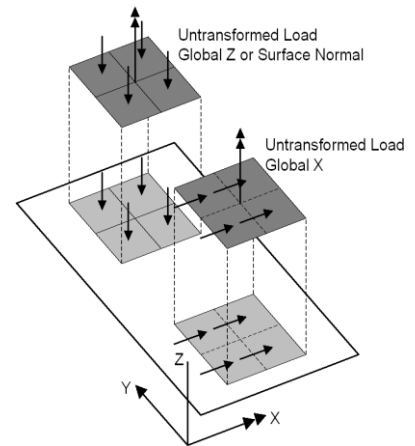
❑ **Patch load divisions** specifies the numbers of divisions in the local x and y directions of the patch being assigned. The divisions are used to split the applied patch into individual component loads before they are in turn used to calculate equivalent nodal loads on the model. By default, the patch 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 defiend 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.
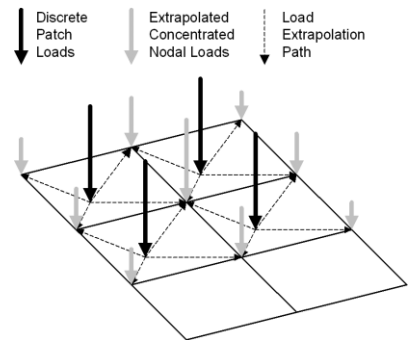
Assigning Discrete Loads

Discrete loads are independent of features therefore their application can be more flexible. The load assignment parameters are explained below:

❑ **Patch Transformation** Changes the patch orientation. 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.
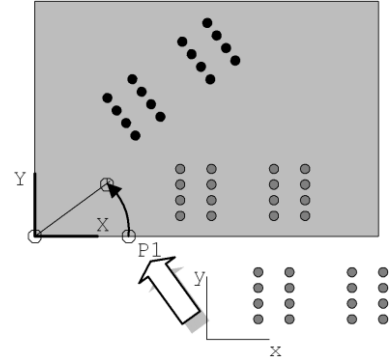
To rotate a patch about its centre, define the patch with its local origin at its centre

❑ **Load transformation** Changes the load orientation from the (untransformed) direction given in the load definition. The transformation applies to the direction of the individual load components rather that to the patch as a whole. For example, it can be used to model braking loads on a 3D model that have horizontal and vertical components by specifying a transformation that will rotate the loads out of the vertical direction and into an inclined plane in the direction of vehicle travel.

❑ **Search area** A **search area** restricts loading to a specified portion of the model. If a search area is not specified, the load is projected onto the whole model. For 2D models it is usually acceptable to default to the whole model, but for 3D models where multiple intersections of the load projection onto the model may occur it is safer to restrict the loading to the required face using a search area. In either case the time taken to assemble the loads is significantly improved by using a search area to restrict the number of elements tested for intersection with the load. Search areas are automatically created and used by the **prestress wizards** to define the target to be loaded.

In addition, the discrete load can be specified to:

❑ **Project onto line** This option is used to project discrete loads onto 2D line beam structures and frame models. Discrete loading is applied to the beam as corresponding forces and moments along the beam.

❑ **Project over area** This option is used to project discrete loads over an area. The area may be defined by a grid of beam elements (a grillage), a plate or shell structure (slabs), or the face of a solid model.

❑ **Project into volume** This option is used to project discrete loads into volumes (solid models) and is primarily for use with tendon loading.

Loads that extend beyond a search 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**.

General loadcase information that can be entered includes:

❑ **Loadcase** specifies in which loadcase the loading is to be applied. Loadcases can themselves be manipulated. See **Loadcase Management** for more details.
❑ **Load factor** specifies a factor by which the loading is multiplied before the equivalent nodal loads are calculated.

Editing of Discrete Loading Data

Editing of pre-defined discrete loading data (such as that used for supplied vehicle loads) allows users to view both the original vehicle definition input data, as well as the actual loading applied (the vehicle load converted into a discrete load format), for any and all vehicles within LUSAS. Editing of user-defined discrete loading data only permits viewing and editing of the discrete loading data.

So, for the case of creating a vehicle load from a pre-defined vehicle, the resulting attribute in the Attributes Treeview has context menu entries named Edit Definition... and Edit Attribute... These menus can be seen by right clicking on the attribute.

- Selecting the **Edit Definition...** menu entry or double clicking the attribute displays the original definition dialog with all the original input data intact. The user can change these inputs that may be either loading parameters such as width, length and intensity etc or even the type of vehicle, at any time. For each modification, the name of the attribute and the equivalent discrete load values are modified. Although the name of the attribute is altered, the attribute itself is merely modified and so the assignment links between the bridge structure and the load will not be lost.
- Selecting the **Edit Attribute...**menu entry displays the equivalent discrete loads. These values may be changed but this breaks the link to the original definition dialog and a warning message will be displayed.

Editing of automatically generated discrete loading data (such as that created by the use of the **prestress wizards**) is not permitted.

Editing patch load divisions

- When the first discrete load type is added to the Attributes Treeview, a 🐞 *Patch divisions* object is also created. Double-clicking on this object displays a dialog which allows the type and default number of patch divisions used on discrete patch loading to be edited.

Case Study. Hydrostatic Loading

In this example, a discrete patch load will be used to apply a hydrostatic load to the side-wall of an underground box culvert.

1. The box culvert wall is defined using a Surface in the global XZ plane with corners at coordinates (0,0,0), (5,0,0), (5,0,3) and (0,0,3). Define a Surface using the **New Surface** button at the specified corner positions.

2. Rotate the view using the **Dynamic Rotate** button until the Surface can be visualised in 3D.

3. Using **Attributes> Mesh> Surface**, define a mesh using **Thick Shell**, **Quadrilateral**, **Linear** elements. Specify the spacing as **15** divisions in the local x and **9** divisions in the local y directions. Since only one Surface is present in the model, the divisions for the mesh can be entered directly onto the Surface mesh dialog.

4. With the cursor in normal mode, **assign** the mesh to the Surface by dragging the attribute from the Treeview onto the selected Surface.

5. To define a patch load that is coincident with the side-wall Surface, first select the four Points defining the Surface in the order they were defined. Choose the **Attributes> Loading** menu item and pick the **Discrete Patch** option. Note that LUSAS has selected a **4 node patch** and filled the Point coordinates into the dialog.

6. The load direction coincides with the global Y axis direction so select **Y** from the **Untransformed Load Direction**. Specify patch corner load intensity values of -3, -3, -1, -1 respectively.

7. The patch definition uses a coordinate system that is coincident to the global Cartesian axis system, so the load can be assigned to the Point at the origin (Point 1). **Assign** the load to Point 1 (0,0,0) leaving all dialog entries as default and press **OK** to assign the load. Note that the patch is drawn as discrete point loads. This is because the patch load is automatically split into point loads.

8. The number of discrete loads in each direction is dependent on the numbers of divisions entered in the Assign Loading dialog. In this case, the default number of divisions (10) is insufficient as there are insufficient loads to apply at least one load per element along the culvert. To improve the load application accuracy, deassign the load from the Point, and reassign using **15** divisions in the local X direction. Leave the Y divisions field blank. Note that LUSAS has automatically used the aspect ratio of the patch load to calculate a suitable number of divisions in local Y.

Hydrostatic Patch Load	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.
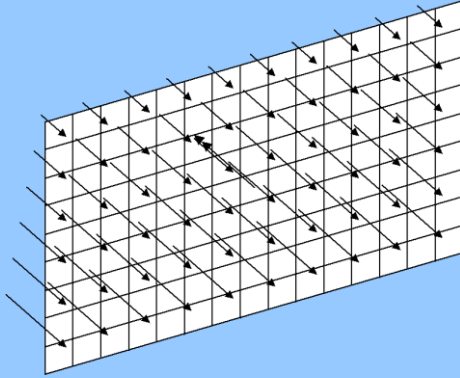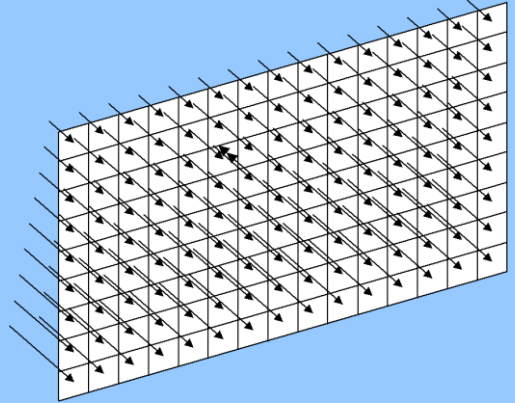
Search Areas

Search areas may be used to restrict the area of application of **discrete loads** (point and patch). This is useful for several reasons:

❑ **Improved Control of Load Application.** The search area will effectively limit the area over which the load is applied so that the effect of loads on certain features may be removed from the analysis. For 3D models it is possible that a chosen projected direction will cross a model in several locations. A search area is therefore used to limit the application of load to one of these multiple intersections. Restricting the area of application of discrete loads allows the same load attributes to be used to apply loads to different parts of the model.

❑ **Speed Improvement** the speed of calculation of equivalent nodal loads will be increased by cutting down the number of features considered in the calculation.

Search areas are automatically created and used by the **prestress wizards** to define the target to be loaded.

In the example shown, a multiple span grillage structure is defined with Span 1 as the search area. A discrete Patch load, indicated by the grey shaded region in the upper diagram, is applied across the whole structure, Span 1 and 2. The area of the structure coinciding with both the Search Area and the patch load will take the load as shown in the lower diagram.

Tip. Search areas should be used if the model is three dimensional and discrete loads are applied, as, for example, for box-section or cellular construction decks.
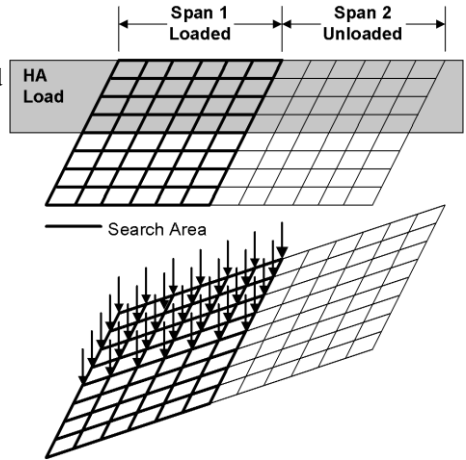
Defining and Assigning Search Areas

Search areas are defined from the **Attributes** menu then **assigned** to the required Lines or Surfaces. Control of loads lying **outside** the search area is available when the load is assigned, see **Assigning Discrete Loads**. If a search area is not specified when the load is assigned, all of the highest order features, excluding volumes, in the model will be used as a default search area. Valid search area configurations are shown below.

Note: The default maximum number of elements that can be used with search areas per grillage bay (each four-sided framing of a section of slab) is 30. In the unlikely event that a higher number is required this can be changed by setting a user-defined option in Modeller. Contact LUSAS technical support if you wish to do this.

Rules for Creating Search Areas for Grillages

The following general guidelines should be noted when assigning a search area to a grillage model.

Overhanging elements defined such that only one side of a cell is missing are included in the search area, as shown.

In these cases LUSAS automatically 'closes' the cell.
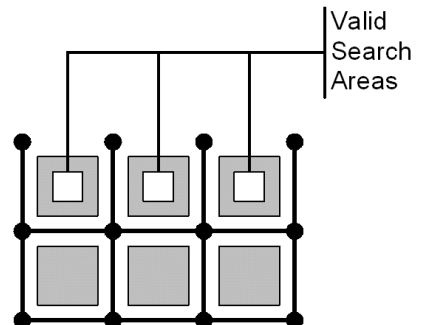
Elements cannot be included in the search area when they overhang from the same node, as shown.

In this case, a dummy Line can be added manually between the 2 overhanging points to close the bay to make the search area valid. When closing the bay in this way note that a single null-line mesh should be used having one mesh division.
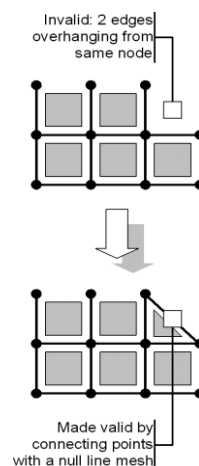
Cells of more than four edges are automatically subdivided into triangles, but overhanging elements are only included if divided by no more than one edge.

In this case, a dummy Line can be added manually between the 2 overhanging points to close the bay to make the search area valid. When closing the bay in this way note that a single null-line mesh should be used having one mesh division.

For the invalid region shown, a dummy Line can be added manually between the 2 overhanging points to close the bay and make the search area valid. When closing the bay in this way note that a single null-line element, having one mesh division should be assigned. If done, the resulting cell of five edges will be subdivided into 4 triangles.
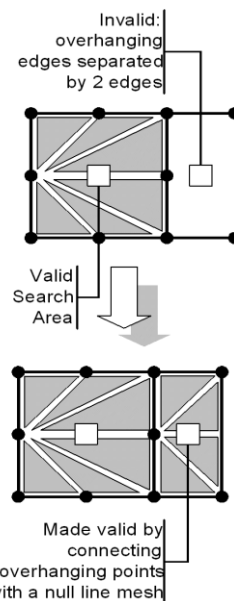
There is no limit to the number of edges that may hang over the main body if the overhanging members are only separated by one edge (right).

In these cases LUSAS automatically 'closes' the cell and either sub-divides the resulting cells into triangles or uses a quadrilateral as appropriate.
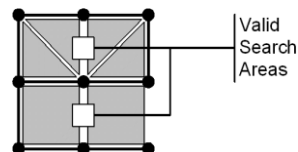
Processing Loads Outside a Search Area

For **point** and **patch** loads any load outside a search area can either be excluded from the search area, or be projected to be included into the search area.

Discrete loads on lines

When discrete loads of patch or point loading are assigned onto Lines with an assigned Search Area the individual load components are projected onto the line, normal to the local x axis of the Line, and their effective loading is calculated and applied to the line as forces and moments along the line.

Options available when assigning onto Lines are:

❑ **Exclude All Load beyond the end of the Line (default)** patch load components beyond the end of line will be disregarded and all load components within the search area will be applied to the line with an appropriate force and moment to represent the positions of the loads.

❑ **Include Full Load** all load components within the search area will be applied to theline with an appropriate force and moment to represent the positions of the loads. Patch load components beyond the end of line will be applied to the point at the end of the line with an appropriate force and moment to represent the actual position of the loads.

Note that when a search area is assigned to a line the search area extends for the length of the line and for an infinite distance perpendicular to the line direction. See the diagram that follows or details.
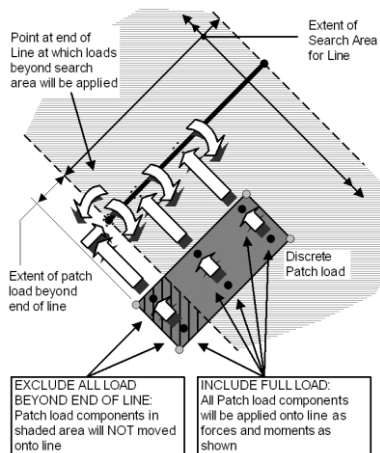
Patch Load onto Line

Patch loads not lying on a line but within an assigned search area will be applied to the line as effective forces and moments. Loads outside the search area can be either included or excluded. If included, the applied moments will be computed by using the actual location of the defined loads

Discrete loads over areas

When discrete patch loads are assigned over an area the projection path(s) is/are defined by the **local x and y axes** of the loading patch. Each patch load component is 'moved' along a specified local x or y direction and added to the first loading positions found inside the patch in that projected direction. See the diagram that follows which illustrate the various options.

Options available when assigning onto areas are:

❏ **Exclude All Load (default)**
❏ **Include Local X Projected Load**
❏ **Include Local Y Projected Load**
❏ **Include Local X and Y Projected Loads**
❏ **Include Non-Projected Load**
❏ **Include Full Local X Load**
❏ **Include Full Local Y Load**
❏ **Include Full Load**

Notes

- Loads will not be moved to the edge of the search area if the entire patch load lies outside the search area.

- Loads inside the search area are not moved.

- Discrete patch loads assigned over areas are not work equivalent as the discrete points are simply lumped at the nearest node.

- Patch loads outside the search area are lumped onto the nearest edge of the search area.

Patch Load Divisions

Number of 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.
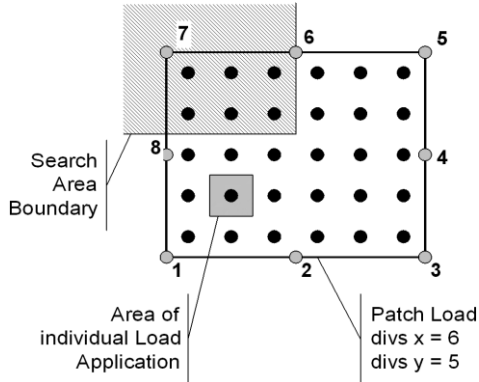
Patch Load Local Coordinates

The local coordinate set is dependent on the order in which the coordinates of the patch vertices are defined.
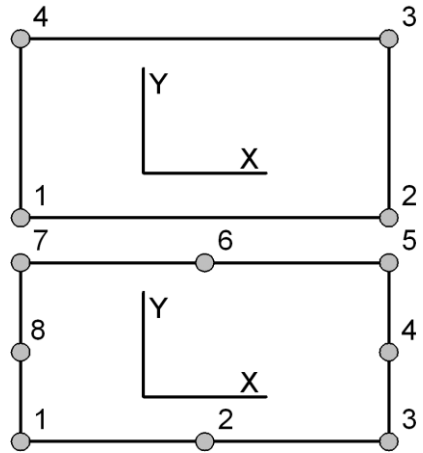
Local X Projected Load

Loads in the **local y** projected region (dark area) are lumped at nearest loading positions within the search area (light area).
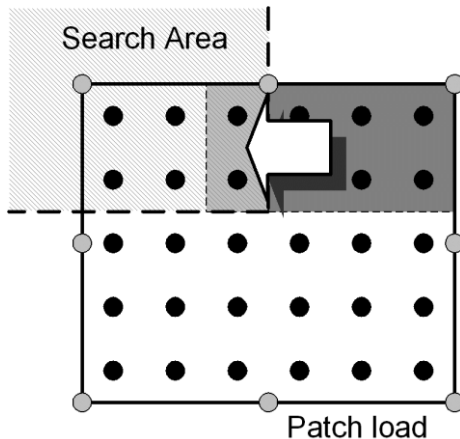
Local Y Projected Load

Loads in the **local y** projected region (dark area) are lumped at nearest loading positions within the search area (light area).
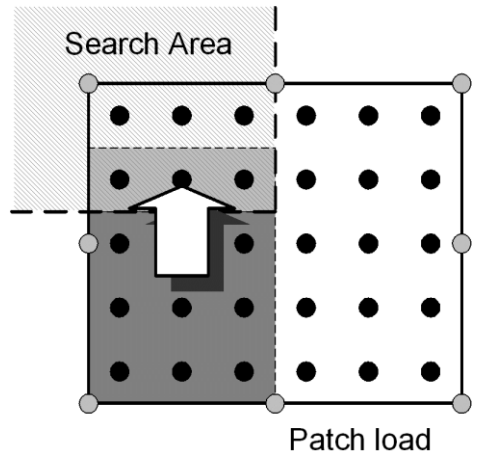
Local X and Y Projected Loads

Loads in the **local x and y** projected regions (dark area) are lumped at nearest loading positions within the search area (light area).
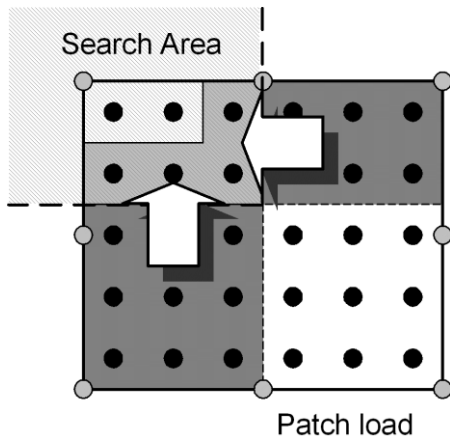
Non-Projected Load

Loads **not** in the **local x and y** projected regions (dark area) are lumped at nearest loading positions within the search area (light area).
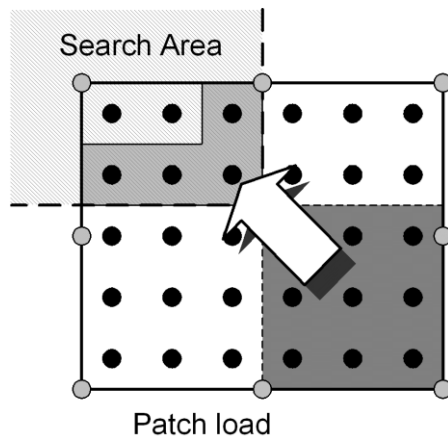
Full Local X Load

Loads in the **full local x** region of the patch (dark area) are lumped at nearest loading positions within the search area (light area).
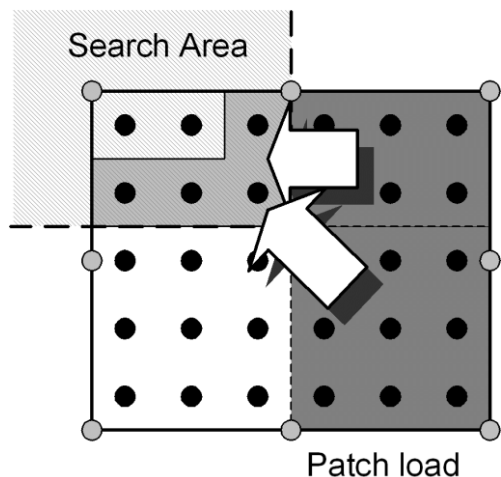
Full Local Y Load

Loads in the **full local y** region of the patch (dark area) are lumped at nearest loading positions within the search area (light area).
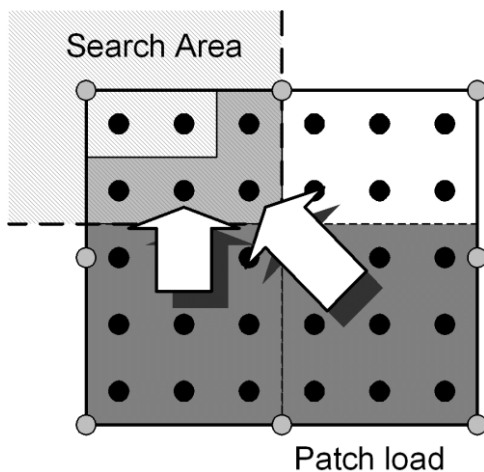
Full Load

All patch loads lying outside the search area (dark area) are lumped at nearest loading positions within the search area (light area).

Patch load
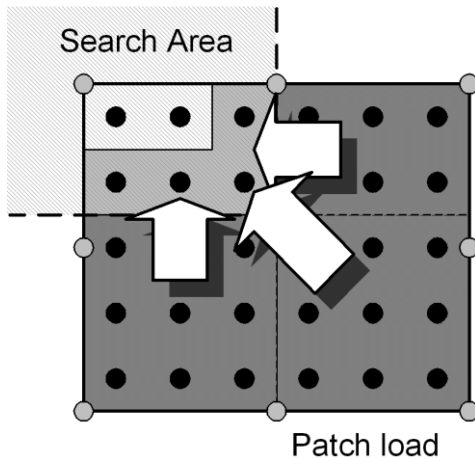
Discrete point loads over areas

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. If discrete point loads are defined by the Arbitrary option, and if the points are defined in an identifiable grid format, then the loading is applied as per the patch loading (that is, the loading components can be projected along the columns and rows of the patch load grid into the search area), otherwise the loading is applied to the nearest load location in the search area.

Discrete point loads into volumes

When discrete point loads are projected into volumes (by being assigned to a particular point on, or within the volume) the applied discrete loads are extrapolated within the elements to create equivalent concentrated nodal loads. When search areas are assigned to volumes the following options are available:

❑ **Exclude All Load (default)** - patch load components outside the volume will be disregarded and all load components within the search area will be extrapolated within the elements to create equivalent concentrated nodal loads.

❑ **Include Full Load - patch load components outside the search area will be applied to the nearest elements in the volume.**

Thermal Loading

Thermal loading is feature based and hence it is assigned to the model geometry**.** Variations in loading on a feature can be specified using a previously defined variation. 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 > Thermal** menu item. This menu item is only displayed if a Thermal or Coupled user interface is chosen on the New Model dialog when creating a new model, or when a Thermal or Coupled user interface is subsequently chosen on the Model Properties dialog.

Flux (CL)

❑ A **Flux** loading produces the LUSAS CL load type which in a field analysis applies a rate of internal heat generation (Q). Positive Q defines heat input.

❑ A total flux, a flux per unit length or a flux per unit area can be specified.

❑ Flux is defined relative to the nodal coordinate system. If the required loading directions of a global load do not lie in the global axes then a **local coordinate** may be assigned to the feature to transform the loads to local coordinate directions.

Distributed Flux (FLD)

❑ A **Distributed Flux** loading produces the LUSAS FLD load type which in a field analysis applies a rate of flux.

Internal Heat Generation (CBF/RIHG)

Defines the internal heat generation for an element. Positive loading values indicate heat generation and negative values indicate heat loss.

▪ The temperature dependent internal heat loading (RIHG) defines the rate of internal heat generation. This load attribute requires a reference temperature for each set of properties.
▪ Defining temperature dependent properties turns a linear thermal field problem into a nonlinear thermal problem.

Notes

● **Load curves** can be used to maintain or increment the RIHG as a nonlinear analysis progresses.

● Automatic load incrementation under Nonlinear Control cannot be used with RIHG loading.

Prescribed Temperature (PDSP/TPDSP)

Defines a prescribed temperature for an element.

❑ The **Incremental** prescribed load type adds to any temperatures present from a previous increment.

❑ The **Total** prescribed load type defines the total temperature at a given node at a specified increment.

Environmental Temperature (ENVT/TDET)

Models external fluid temperature and associated convection and radiation heat transfer coefficients. If an element face does not have an environmental temperature assigned it is assumed to be perfectly insulated.

❑ The temperature dependent environmental temperature loading (TDET) models properties that vary with nodal temperature. This load attribute requires a reference temperature for each set of properties.

❑ Defining temperature dependent properties will turn a linear thermal field problem into a nonlinear thermal problem.

Notes

- If heat transfer coefficients vary on a specified face the values will be interpolated using the shape functions to the Gauss points.

- If a non-zero radiation heat transfer coefficient is specified, the problem is nonlinear and **Nonlinear Control** must be used.

- Load curves can be used to maintain or increment the environmental temperature as a nonlinear analysis progresses.

- Automatic load incrementation within the **Nonlinear Control** can be used to increment ENVT loading.

Internal Heat User

Allows user-defined input of internal heat generation for an element for use with user-written software programs. Values can be entered in multi-column format. Positive loading values indicate heat generation and negative values indicate heat loss.

Concrete Heat of Hydration

Concrete heat of hydration is defined as a part of the material model to be used. See **Isotropic/Orthotropic Material**

Case Study. Temperature Dependent Loading

Temperature dependent **environmental** loading can be useful to model experimentally determined correlation for convective coefficients. For example, if the convective coefficient may be given by **C [deltat]** to the power one third where **C** is a constant, **deltat** is the temperature difference between the surface and the environment. To specify this loading in LUSAS you would define the convective coefficient at as many reference points as are required to give a good piece-wise linear approximation of the function. Each reference temperature point is defined in a loading attribute and collectively these attributes define a single loading table. The loading table is then assigned to the features as required.

1. Define a row of Surface features.

2. Use an incremental prescribed loading to fix the temperatures at one end of the model.

3. Define a convective coefficient function using environmental loading (temperature dependent).

4. Assign the loading and solve.

5. Since the problem is one-dimensional the solution may be checked to ensure that the convection coefficient has been correctly interpolated.

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
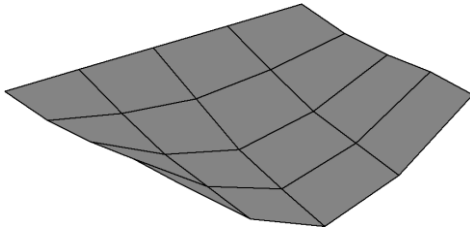
20 Masters
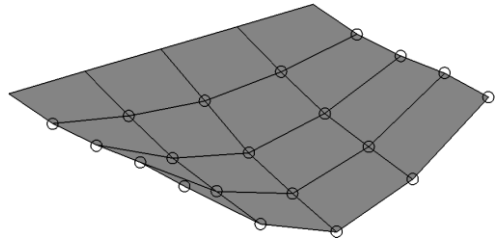
15 Masters
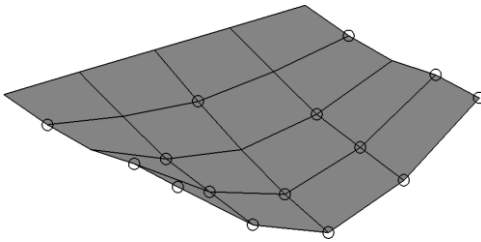
10 Masters
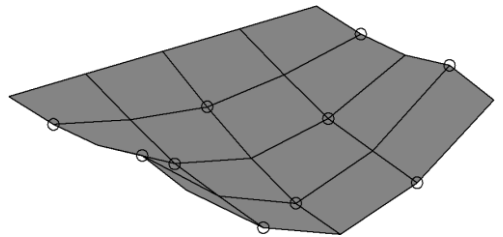
5 Masters
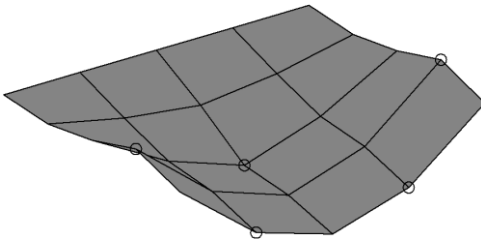
Equivalencing

The equivalence facility is used to merge coincident nodes on otherwise unconnected features. If an equivalence attribute is assigned to any features the nodes will automatically be equivalenced after meshing has been carried out.

There are several ways equivalencing can be set up to work:

❑ By assigning equivalence tolerances to certain features - only these features will be equivalenced, all others are ignored.

❑ By switching on the automatic tolerancing, and accepting the default tolerance - all features are equivalenced according to the default tolerance.

❑ By switching on the automatic tolerancing, and assigning other equivalence tolerances to certain features - all features are equivalenced according to either an assigned tolerance or the default tolerance.

Example

In this example, Surfaces 1 and 2 do not share a common boundary Line, therefore the nodes created on their common boundaries will not be joined and must be equivalenced. Node 2 will merge with node 1 if it lies within the equivalence tolerance.
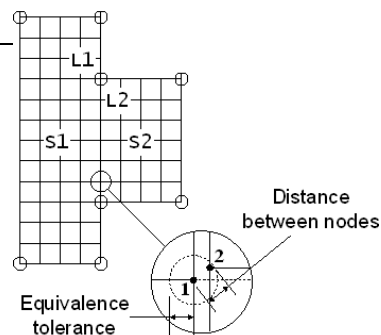
Using Equivalencing

Equivalence attributes are defined from the **Attributes** menu. They are defined as a tolerance, which is used to determine whether nodes are considered to be coincident.

The equivalence attribute is **assigned** to the features that are to be checked for coincident nodes. When an equivalence dataset is assigned to a lower order feature it will search through all higher order features for nodes to be checked. For example, in order to equivalence two Volumes at their boundaries, it is more efficient to assign the equivalence to the Surfaces on the boundaries, as a smaller number of nodes need to be checked.

Automatic Equivalencing

Automatic equivalencing can be activated from the **Meshing** tab of the **Model Properties** dialog. This will equivalence all features in the model on meshing if they are within the default equivalence tolerance, or within an assigned tolerance. Note. Remeshing occurs each time a relevant command is issued, but a forced remesh is possible using the **Utilities> Mesh> Mesh Reset** menu item. Automatic equivalencing can be time consuming for models with a large number of nodes.

Visualising Equivalences

Displays the features which have a specified equivalence assigned to them in a chosen colour and line style.

In this example different equivalence tolerances are assigned to different parts of a model to merge more coarsely or finely as required. Using visualisation, the lines to which the equivalence is assigned can be highlighted. Equivalenced nodes can also be visualised as they are removed. In this diagram they are shown using the square symbol.
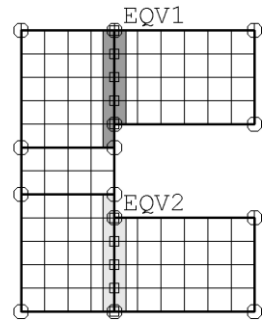
Summary

- More than one equivalence attribute may be defined in order to rationalise more than one section of the model independently.

- More than one equivalence attribute can be assigned to a feature to equivalence it within a different subset of the model.

- A check for unconnected elements and nodes can be performed using an outline mesh plot (Mesh layer properties), or by checking for duplicate node numbers using the View > Browse selection menu item and box-selecting around selected points to see if more than one point appears in the list shown.

- The equivalence tolerance must be less than the smallest distance between two nodes on the same feature, otherwise the equivalencing operation will fail.

- Equivalencing may be used to position a point load or support at a node (which is not at a defining feature Point). A Point must be created, the load or support assigned, and the Point and meshed feature equivalenced.

- Equivalencing may be used to merge nodes on the constituent Lines of combined Lines i.e. the nodes on an entire combined Line may be equivalenced, including the Lines forming it.

Age

Age attributes define the age in days between creation and activation of features in the model and are used in conjunction with the **CEB-FIP Concrete Material Model**. When assigned to a feature all elements created by that feature are assigned the specified age. Age attributes are defined from the **Attributes> Age** menu item.

See *Solver Reference Manual* for further details.

Damping

Damping is used to define the frequency dependent **Rayleigh damping parameters** for elements which contribute to the damping of the structure. Viscous (modal) and structural (hysteretic) damping can be specified. If no damping attributes are specified the properties are taken from the material properties (click on **dynamic properties** on the elastic page of the material attribute dialog).

Damping is usually specified when distributed viscous and/or structural damping factors are required for modal damping control. A modal damping analysis is performed as part of an eigenvalue analysis.

Defining Damping

Structural or viscous damping is defined from the **Attributes> Damping** menu item and **assigned** to features in the usual way. Mass and stiffness Rayleigh damping parameters are linked with the corresponding reference circular frequency value at which they apply in a damping attribute. If more than one set of damping values is defined linear interpolation is used to calculate damping values at intervening frequencies.

Birth and Death (Activation/Deactivation of Elements)

Birth and death enables the modelling of a staged construction process (e.g. tunnelling or bridge construction), whereby selected elements are activated and deactivated as the simulation process requires. Birth and death attributes are defined from the Attributes menu and are assigned and manipulated in the same way as other attributes.

All elements to be used in the model are defined at the start of the analysis. To model the absence of a part of the model, it is assigned a deactivate attribute. In structural analyses, the underlying elements have their stiffness matrix reduced in magnitude, while for field analysis the conductivity matrix (or other analogous quantity) is reduced. This ensures the deactivated elements have a negligible effect on the behaviour of the remaining model. The element stresses and strains, fluxes and gradients and other analogous quantities are all set to zero.

To model the addition of a part to the model an activate attribute is assigned. In structural analyses, an unmodified stiffness matrix is computed for the underlying elements and these activated elements are introduced in a stress/strain free state, except for any initial stresses or strains that have been defined. Strains are incremented from the point of activation and the current geometry is used to define the activated element's initial geometry. In a field analyses activation works in the same manner, except that the quantities affected are the conductivity matrix (or other analogous quantity), the fluxes and the gradients.

By setting LUSAS Solver option 385, however, loads applied to deactivated elements are preserved to enable reapplication if and when the elements are re-activated.

Percent to Redistribute

The deactivate command provides control over the way in which these internal forces are processed by specifying how much of the internal forces should be redistributed:

❑ **Zero Redistribution** 0% of the internal forces in a deactivated element may be redistributed in the system (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 may be redistributed in the system (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.

Excavation Stage 1

Top layer of soil deactivated and lining activated. Lining and soil elements duplicated in the model.
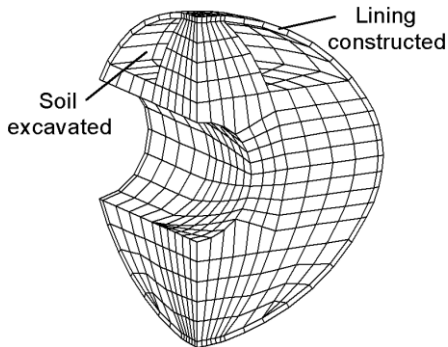
Tunnel Excavation Stage 2

Second layer deactivated as soil excavated. Surrounding lining elements activated
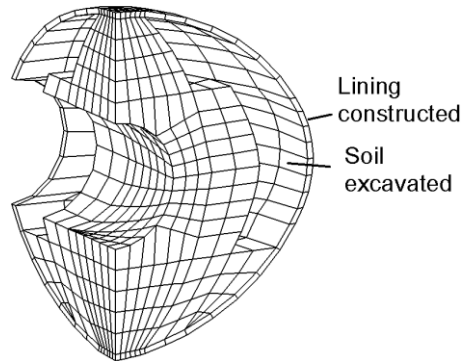
Tunnel Excavation Stage 3

Remaining second layer soil elements deactivated.
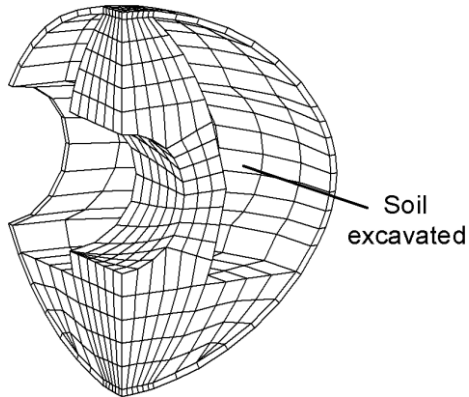
Tunnel Excavation Stage 4

Supporting soil pillar removed and top lining activated.
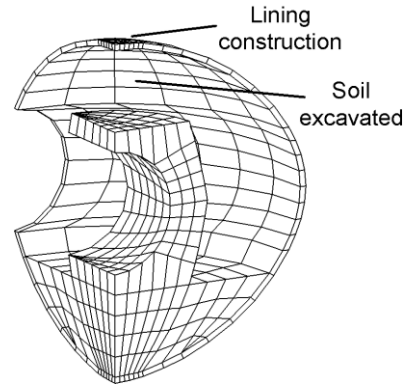
Tunnel Excavation Stage 5

Final central soil column removed.
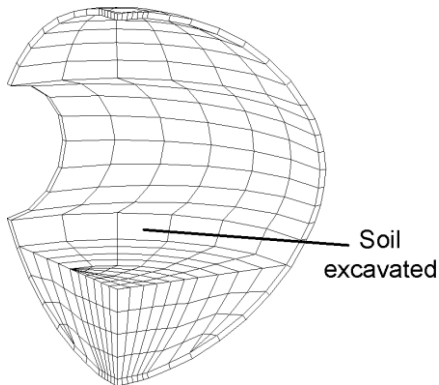
Using Birth and Death Attributes

Activate and deactivate attributes are defined from the Attributes menu. The attributes are assigned on a feature basis to control the history of the underlying elements throughout the analysis. The loadcase is specified during assignment to indicate at what point the elements are added or removed.

Notes

- Elements cannot be activated and deactivated in the following circumstances:
  - Explicit dynamics analyses.

- Fourier analyses.

- When using updated Lagrangian or Eulerian geometric nonlinearity.

- When they are adjacent to slidelines..

- Activation and deactivation can only be carried out within a nonlinear analysis.

- Deactivation and activation can take place over several increments if convergence difficulties are encountered.

- 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 changed by you. In a dynamic analysis the mass and damping matrices are also scaled down by the same factor.

- When an element is deactivated, 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.

- If required, initial stresses/strains and residual stresses may be defined for an element at the re-activation stage.

- The activation of an element which is currently active results in an initialisation of stresses/strains to zero, an update of the initial geometry to the current geometry and the element is considered to have just become active. The internal equilibrating forces which currently exist in the element will immediately be redistributed throughout the mesh. This provides a simplified approach in some cases.

- The direction of local element axes can change during an analysis when elements are deactivated and reactivated. In particular, 3-noded beam elements that use the central node to define the local axes should be avoided as this can lead to confusion. For such elements the sign convention for bending moments for a particular element may change after re-activation (e.g. it is recommended that BSL4 should be preferred to BSL3 so that the 4th node is used to define the local axes and not the initial element curvature).

- Care should be taken when deactivating elements in a geometrically nonlinear analysis, especially if large displacements are present. It may be necessary to apply prescribed displacements to deactivated elements in order to attain a required configuration for reactivation.

- It should be noted that the internal forces in the elements will not balance the applied loading until all residual forces in activated/deactivated elements have been redistributed.

Thermal Surfaces and Heat Transfer

The thermal surface facility allows thermal gaps, contact and diffuse radiation to be modelled. Thermal surfaces are used to model the thermal interaction of two distinct bodies, or two different parts of the same body through a fluid medium.

❑ **Thermal Gaps** are used to model gaps between structures that are relatively close together.

❑ **Contact** is used in a thermo-mechanical coupled analysis where contact takes place and the contact pressure effects are then included in the analysis.

❑ **Diffuse Radiation** is the process of heat transfer from a radiation surface to the environment or to another thermal surface defining the same radiation surface. Radiation is modelled by specifying radiative properties for thermal surfaces.

Thermal surfaces

Thermal Surfaces are the thermal equivalent of structural slidelines. They are defined from the **Attributes> Thermal Surface** menu item and are assigned to features of the model and manipulated in the same way as all other attributes. A thermal surface must be defined before thermal gap or radiation properties can be specified.

❑ **Radiation properties** are required when defining a radiation surface for heat transfer by radiation exchange.

❑ **Environment properties** are required when thermal environment properties exist. Used for heat transfer to the environment (convection and conduction).

Thermal Surfaces work in conjunction with **Thermal Gaps** and **RadiationSurfaces**. See below for details.

Heat Transfer

Thermal gap and radiation surface properties are used to dictate the type of heat transfer that can take place between Thermal Surfaces. They are defined from the **Utilities> Heat Transfer...** menu item. As utilities they cannot be assigned directly to features of the model as Thermal Surfaces can. A thermal surface must have been defined prior to specifying any thermal gap or radiation properties. The process of thermal surface / heat transfer definition is summarised in the following diagram.

❑ **Thermal Gaps** Thermal gaps are used to model heat transfer across a gap and heat transfer by contact when a gap is deemed to have closed. If these effects are required, the thermal surfaces defining the gap must be specified on the Thermal Gap properties dialog.

❑ **Radiation Surfaces** Diffuse radiation exchange may be modelled with a radiation surface that is defined by any number of thermal surfaces. Planes of symmetry that cut through the radiation enclosures may be defined so that it is not necessary to model the whole structure. Radiation surfaces allow for the calculation of diffuse view factors. These view factors may be output to a print file

Specifying thermal surfaces defining a gap

Pre-defined thermal surfaces can be selected on the Thermal Gap properties dialog in order to define a gap. The gap can be defined as active or inactive initially and be set to change according to loadcase.

Choosing Thermal Properties

The following flowchart guides the decision making process for choosing thermal properties. The process is simplified if the analysis only considers a single body, when only environmental thermal properties are required. For analyses where discrete (multiple) bodies are considered, factors such as body proximity and whether the bodies are touching, or are likely to touch during the analysis, become important and the choice of thermal properties changes. Follow a route through the flowchart below and define your thermal surfaces using the properties given in the shaded box.
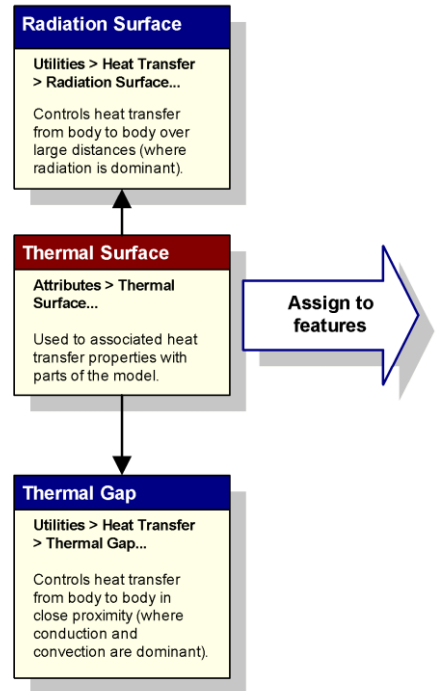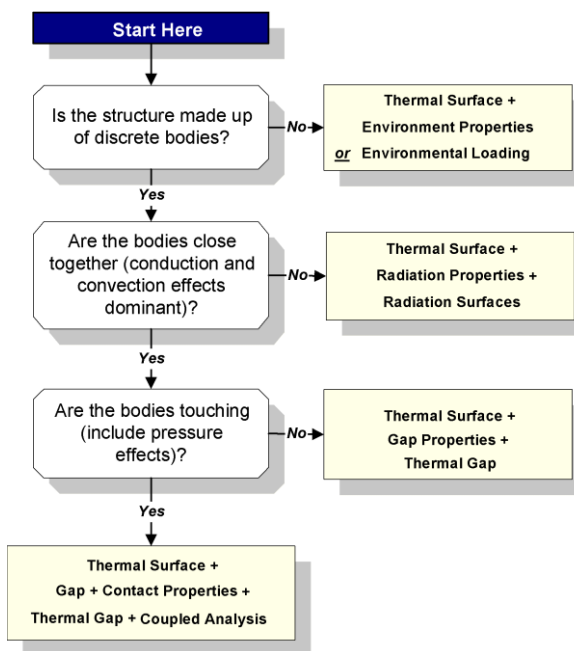
Environmental Nodes (LUSAS analysis data file)

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 analysis data file if required. See the *Solver Reference Manual* for further information.

Radiation Options

Radiation options are set from the **Model Properties** dialog. Available options are:

❑ **Suppress Recalculation of View Factors in Coupled Analysis** (Model properties, Solution tab, Thermal options). 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.

Constraint Equations

A constraint can be defined to constrain the movement of a geometric or nodal freedom. Constraint equations allow linear relationships between nodal freedoms to be set up. Constraint equations can be used to allow plane surfaces to remain plane while they may translate and/or rotate in space. Similarly straight lines can be constrained to remain straight, and different parts of a model can be connected so as to behave as if connected by rigid links. These geometric constraints are only valid for small displacements. Constraint equations can also be used to model cyclic symmetry, for example a single blade from a complete rotor may be modelled and then constrained to behave as if it were part of the complete model. As constraint equations refer to transformed nodal freedoms, any local coordinate assigned to the features are taken into account during **tabulation** when the constraint equations are assembled.

Several different types of Constraint Equations can be defined from the **Attributes> Constraint Equation** menu item. Constraints are grouped under the following types:

Displacement Control

❑ **Specified Variable** a nodal freedom takes a specified value across all the nodes in the assigned features, In this example, a specified variable constraint of Displacement in the X direction with value 1.0 is assigned to Point 1. The underlying node is then allowed to displace only by the specified distance in the specified X direction.
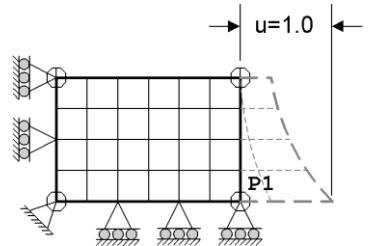
❑ **Constant Variable** used where a nodal freedom value is constant but unknown across all the nodes in the assigned features.
In this example, a constant variable constraint of displacement in the X direction is assigned to Line 1. The underlying nodes move a constant amount in that direction.
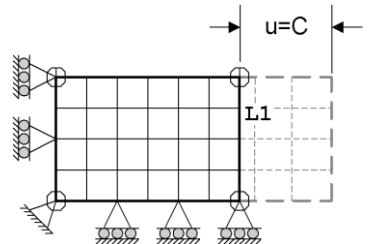
❑ **Vector Path** The nodes in the assigned features may be constrained to move along a specified vector defined by 2 Points or by 2 sets of X, Y and Z coordinates.

In this example, vertical and horizontal vectors are used to restrict movement in those directions. Note that the vectors are used purely to define a direction. Nodes can travel along a vector in either direction.
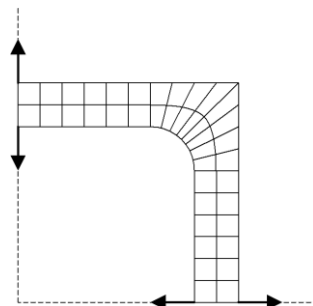
Geometric

❑ **Rigid Displacements** The nodes in the assigned features may be constrained to be rigid, the group of nodes may translate and/or rotate but their positions relative to one another remain constant. Only translational displacements can be constrained using this type of constraint. This type of constraint is only valid for small displacements. Assigning a constraint of this type to Lines on either side of a gap, as in the example shown, maintains the underlying undeformed node positions relative to each other as if a rigid block were in place between the structures.
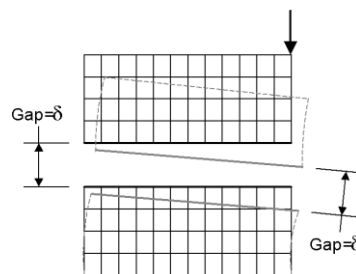
❑ **Rigid Links** Each rigid link attribute can be used at one location, to create a rigid fixity between features that it is assigned to. It is similar to the Rigid Displacements constraint type, except that rotational freedoms are also constrained to be rigid. In the example shown here, the end of a beam is rigidly linked to the shell edges around a cylinder. The plane containing beam and cylinder end will remain plane throughout the analysis.
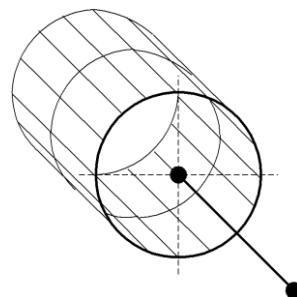
❑ **Planar Surface** A surface may be constrained to remain plane, the surface may translate and/or rotate but remains plane. Nodal positions may vary relative to other nodes on the surface. This type of constraint is only valid for small displacements. In this example, a planar Surface constraint is assigned to the top Surface to force the underlying nodes to remain planar during loading. Constrained nodes may move relative to each other as long as they remain in plane.
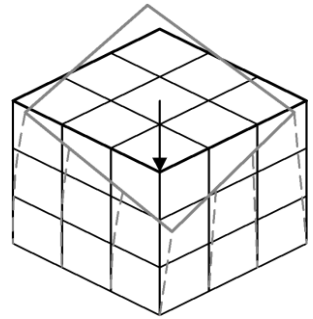
❑ **Straight Line** A straight line may be constrained to remain straight, the line may translate and/or rotate but will remain straight. Nodal positions may vary relative to other nodes along the line. This constraint type is only valid for small displacements. In the example shown, a straight Line constraint is assigned to Line 3 to force underlying nodes to remain in a straight line relative to each other during loading. Constrained nodes may move relative to each other as long as they remain in a straight line.
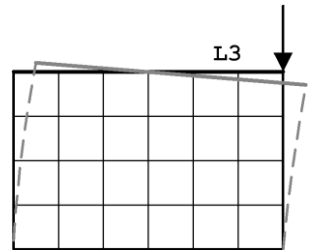
Cyclic

❑ **Cyclic Rotation** Cyclic rotational symmetry may be used to model a section from a continuous ring. The mesh on the two planes of symmetry may be different. In the example shown, the radial Lines are defined as a Master and Slave pair maintaining cyclic symmetry around the structure. Meshes on the Master and Slave Lines need not match.
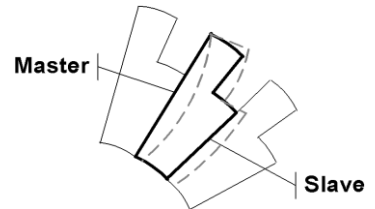
❑ **Cyclic Translation** Cyclic translational symmetry may be used to model a section from a continuous strip. The mesh on the two planes of symmetry may be different. In the example shown here, Master and Slave Surfaces define start and finish positions of repeating sections. Meshes on Master and Slave need not match.
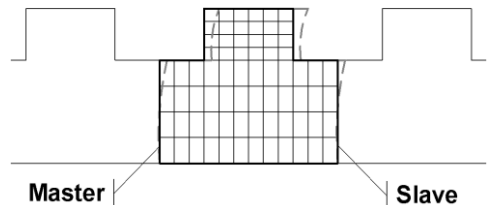
Tied Mesh

❑ **Specified Constraint** Tied meshes may be used to force two sets of assigned features to move together in a similar manner to tied slidelines. The meshes are tied along Master and Slave Lines to restrict relative movement. The mesh on the two sets of features need not match. A search direction vector is defined to limit the mesh to which it is tied. A vector defines the direction in which the constraint is applied.
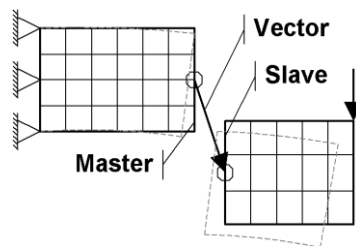
❑ **Normal Constraint** Meshes tied along Master and Slave Lines to restrict relative movement. The underlying nodes maintain their original relative positions under loading. Meshes on Master/Slave need not match. This form of tied mesh constraint uses a search direction normal to the Master/Slave surfaces to detect the mesh to which it is tied.
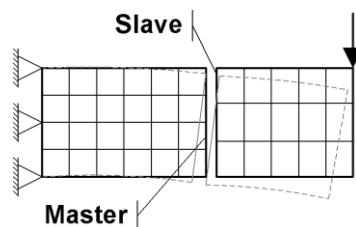
Case Study. Using Constraint Equations

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.

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.
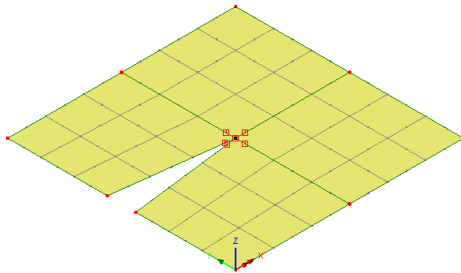
Crack tip attribute assigned to a point in a 2D model   **Crack tip attribute assigned to a line in a 3D model**
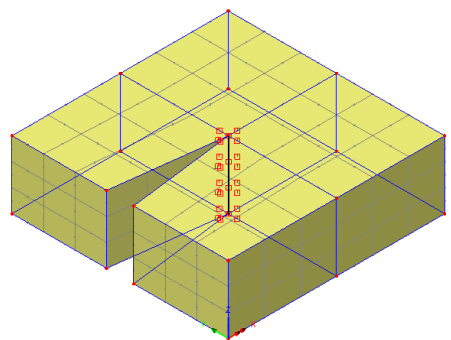
Slidelines

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.

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.
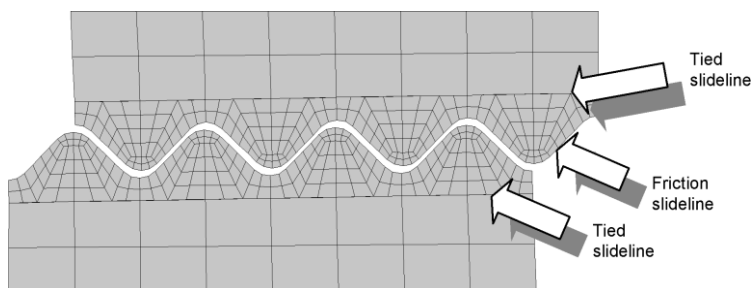
Slidelines are assigned to pairs or groups of features in a model, with one pair/group termed master and the second pair/group termed slave. An element face that lies on a slideline is called a slideline segment.

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. Except for tied slidelines, the slideline contact facility is inherently nonlinear and must be used in a nonlinear analysis.

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 interface	IAX4, IAX6, IPN4, IPN6
3D interface	IS6, IS8, IS16, IS12
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.
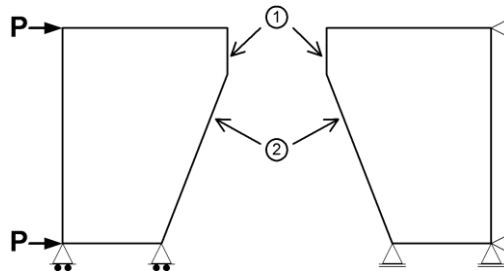
Pre-contact brings two bodies into initial contact by using interface forces that act between the slideline surfaces in order to avoid unrestrained rigid body motion. These forces act in a direction normal to each surface. One of the surfaces must be free to move as a rigid body and the direction of 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)

Type changes during analysis

Select this option if the slideline type is to change during the analysis.

The slideline can be changed from one type to being any other type at any stage in the analysis. For example, the slideline can be tied to begin with and then released at a later stage. If this option is not selected, the initial slideline type is used throughout the analysis.

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.

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
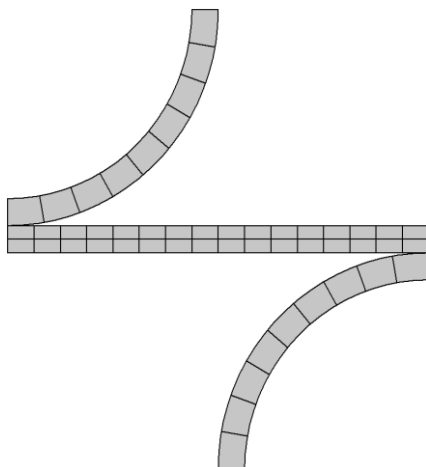
Initial configuration.	Deformed configuration.
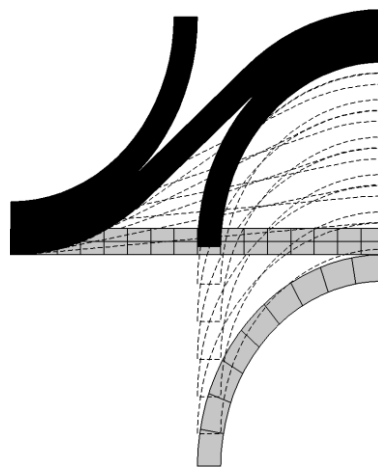
Composites

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 amount of fabric to resin in a lamina) 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.
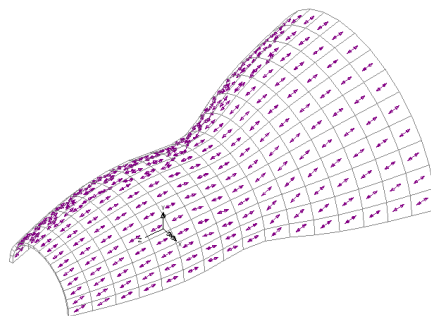
Notes

- 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:

- ❑ 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.
- ❑ 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.
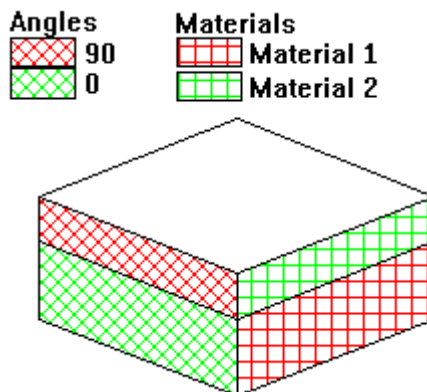
Assigning Composite Properties

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.

If a model has been meshed prior to the assignment of composite properties

Visualisation of Composites Properties

To visualise assigned composites properties the surface or volume must be meshed. The following composite properties can be visualised:

- **Fibre (ply) directions**
- **Draping grid**
- **Lamina thickness, Skew angle, Offset layer, Fibre volume fraction**

Visualisation of Fibre (Ply) Directions

Once assigned to features which have a mesh assigned, the fibre directions of assigned composite data can be examined graphically as follows:

1. Right-click on the **Attributes** entry in the 🔲 Treeview and select **Properties**.
2. On the Composite tab, click on **Settings** and select **Visualise ply directions**. The x and y axes define the warp and weft directions respectively; the item x&y displays both directions at the same time.

Lamina directions can be plotted as an x, y, z or x&y axes at any layer. For solids the axes may be placed at the top/bottom or middle of the chosen layer, for shells the axes are placed

on the mid surface of the shell element. For details of how to choose a composite layer see **Setting The Active Composite Layer**.

Visualisation of Draping Grid

The draping grid for individual lamina can be examined graphically as follows:

1. Right-click on the **Attributes** entry in the ⊟ Treeview and select **Properties**.
2. On the Composite tab, click on **Settings** and select **Visualise ply directions**. Then select **Draping grid**.

If no mesh has been assigned to a model prior to selecting this option only the draping grid (and not the ply directions) will be visualised.

For details of how to choose a composite layer see **Setting The Active Composite Layer**.

Extending the draping grid

LUSAS Draped Solids and Shells grids are automatically trimmed at Surface boundaries. FiberSIM and Simulayt generated grids are not. If required, the draping grid can be extended by one grid row to ensure the edges of the component are fully enclosed by the draping grid. For FiberSIM and Simulayt grids this is specified at the file import stage (accessed via the Attributes > Composite menu item). For LUSAS Draped Solids and Shells grids, this is done via the Draping options on the **Model Properties** dialog.
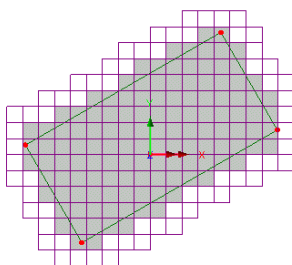
Example of draping grid being extended by one row
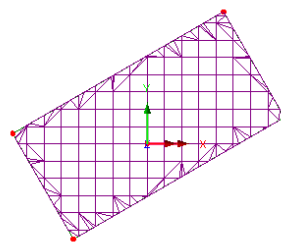
Example of draping grid being trimmed to a surface boundary

Visualisation of Other Composite Model Data

To view other composite model data such as lamina thickness, skew angles, offset layers and fibre volume fractions:

1. With a Contours layer in the ⊟ Treeview right-click on **Contours**.
2. On the **Contour Results** tab select the **Composites (model)** entity and select the composite attribute name in the right-hand panel. The fibre volume fraction and skew angle (and any other composite modelling options relevant for the model) will appear for selection in the Component combo box.

Setting the Active Composite Layer

Composite shell and solid elements have multiple layers (laminae) of different materials though their thickness. The lamina or lamina name on which results or orientation axes are to be viewed is chosen by setting that lamina active. A lamina is set active by selecting the lamina with the right-hand mouse button from the ⚛ Treeview and picking **Set Lamina Active** or **Set Name Active** from the context menu. A black dot next to a lamina indicates the active lamina..
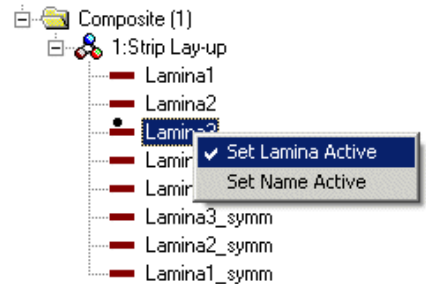
If a lamina is set active only results or orientation axes for that lamina in that composite attribute will be displayed. If the lamina name is set active, results or orientation axes will be displayed for all laminae with that name across all composite attributes.

When viewing results the material transformation should be used to display stresses on or off axis.

Local Coordinates

Local Coordinates define coordinate systems that differ from the default global Cartesian system. Local coordinates are defined from the **Attributes> Local Coordinate** menu item and have several uses:

❑ **Geometry Definition** Geometry features may be defined in a local coordinate system by set the chosen local coordinate active. When a local coordinate is active, all dialog entries relating to global X, Y and Z coordinate input use the transformed axis set as a basis for input.
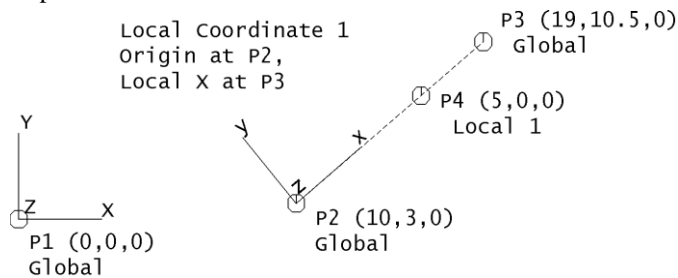
❑ **Transforming Nodal Freedoms** When assigned to features the effect is to transform the degrees of freedom of the underlying element nodes. This has the effect of transforming the directions of applied global load and support conditions. In the example below, global freedoms are transformed to radial directions by assigning a cylindrical coordinate to the Lines around the hole. This method of transforming nodal freedoms is only valid for small deflections, since the freedom directions are not updated during analysis.
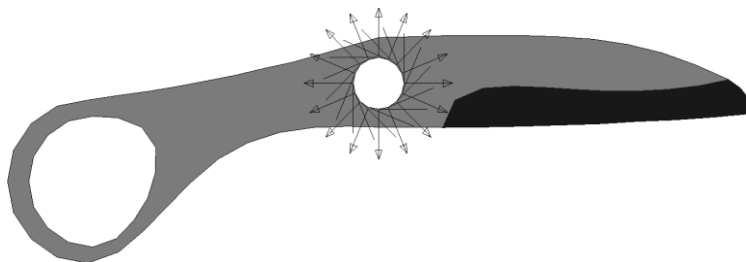
❑ **Materials** A local coordinate may be used to align orthotropic and anisotropic materials.

❑ **Variations** Variations may be defined using functions in terms of a local coordinate.

❑ **Composites** A local coordinate may be used to align composite attributes when they are assigned to the model.

❑ **Element Orientation A local coordinate may be used at the mesh assignment stage to orient beam and joint elements.**

❑ **Results Transformation** Results can be output relative to a local coordinate. For example, this is useful when looking at results on elements when the axes are not consistent.

Local Coordinate Types

Cartesian, cylindrical and spherical local coordinates are defined by indicating three positions in space defining a local xy plane (origin, x axis, xy plane). The type of coordinate chosen will dictate how the axes are defined.

❑ **Cartesian** - Based on standard x, y and z coordinates arbitrarily oriented in space.

❑ **Cylindrical** - Based on the axes of a cylinder - radius, angle and distance along the cylinder axis.

- For a local cylindrical coordinate defined along the z axis a point is specified as (r, theta, z), where:

  r is the radius perpendicular to the local z axis

  theta is the angle in degrees measured from the positive x direction of the local xy plane, clockwise about the local z axis when looking in the positive z direction

  z is the distance along the z axis.
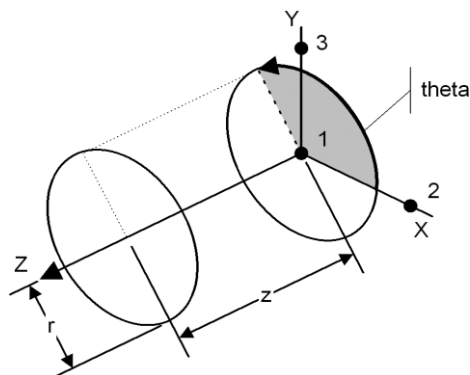
- For a local cylindrical coordinate
  defined along the y axis a point is
  specified as (r, y, theta), where:

  r is the radius perpendicular to the
  local y axis

  theta is the angle in degrees
  measured from the positive z
  direction of the local xz plane,
  clockwise about the local y axis
  when looking in the positive y
  direction

  y is the distance along the y axis
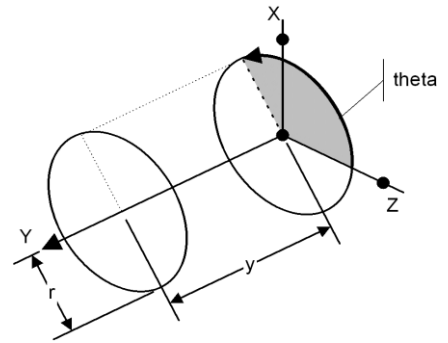
- For a local cylindrical coordinate
  defined along the x axis a point is
  specified as (x, r, theta), where:

  r is the radius perpendicular to the
  local x axis.

  theta is the angle in degrees
  measured from the positive y
  direction of the local yz plane,
  clockwise about the local x axis
  when looking in the positive x
  direction

  x is the distance along the x axis
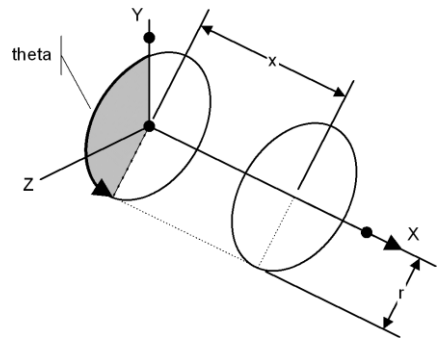
❑ **Spherical** Based on the axes of a sphere
  defined by a radius, tangential angle and
  angle around a meridian.

  Coordinates of a point are specified as (r, t,
  c), where:

  r is the radius of the sphere on which
  the point lies from the local origin

  t is the angle in degrees measured
  from the positive x direction of the
  local xz plane, clockwise about the
  local z axis when looking in the
  positive z direction

  c is the angle in degrees measured from the positive z axis to the radius line
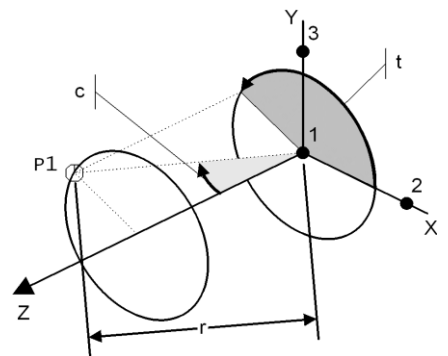
Warning. There is no equivalent spherical set in Solver, therefore freedoms cannot be
transformed using this type of local coordinate system.

- **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 can not be set active.

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.**

Note. Defining a new coordinate set does not automatically make it the active set, see Using Local Coordinates below.

❑ **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

- Local coordinate set types cannot be modified. E.g. a Cartesian sets 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.

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 Window 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 as all attributes.

Loadcases

Loadcase dependent assignments, **analysis control**, load combinations, envelopes, and loadcase results are displayed in the Loadcase Treeview 🔍. The active loadcase controls which loadcase dependent attributes are visualised in the graphics window and which results are displayed. A loadcase is set active using its context menu. At least one loadcase will always exist in the Treeview.
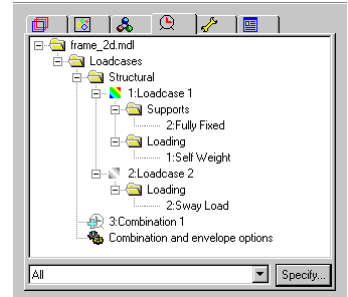
Load curves

For analyses in which the load varies with time (or increment number) **load curves** may be used. When using load curves all loads must be assigned to a load curve instead of a loadcase. All other loadcase dependent attributes (support, slidelines etc.) and analysis control is assigned to loadcases in the usual way. The analysis control assigned to each loadcase determines the time over which the load is applied. The magnitude of the applied load is computed from the time dependent function defined within each load curve. Any number of loadcases and load curves may be specified within a single analysis. Each load curve is assumed to begin at the start of the analysis (t=0). If the input values stat from t=n the load curve is assumed to be zero when t<n.

Results may be manipulated using **combinations and envelopes**, **fatigue loadcases** and **IMD loadcases** all of which are added to the 🔍 Treeview.

Creating Loadcases

New loadcases may be added to the 🔍 Treeview in the following ways:

- By creating a Structural, Thermal, Envelope, Combination, Fatigue or IMDPlus loadcase from the **Utilities** menu.

- By entering a loadcase name when a load attribute is assigned to a feature on a model.

- By right-clicking on a Loadcases, Structural or Thermal entry in the 🔍 Treeview and selecting the New loadcase menu item.

- By copying and pasting existing loadcases in the 🔍 Treeview.

Adding Gravity Loading to a Structural Loadcase

As an alternative to defining gravity by specifying and assigning a constant body force to all features on a model, gravity can also be added to a model as a property of a structural loadcase. This can be done in three ways:

- By selecting the the **Automatically add gravity to this loadcase** option on the dialog that is displayed when defining a new loadcase or displayed when editing the properties of an existing loadcase.

- By selecting the **Gravity** menu item from the context menu for an individual loadcase.

- By selecting the **Add Gravity** menu item from the context menu of the Structural folder in the Loadcase Treeview. This effectively sets gravity loading to be 'on' for all structural loadcases in the Treeview 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.

Note. 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. No visualisation of gravity loading on the model is provided for gravity defined as a property of 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.

Manipulating Loadcases

General loadcase editing commands are available from the context menu that is activated by right-clicking on a loadcase in the 🔍 Treeview. The following commands are available:

- ❏ **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 🔍 Treeview.
- ❏ **Delete** Attribute assignments must be deassigned before a loadcase can be deleted. At least one loadcase will always exist in the Treeview.
- ❏ **Rename** Allow the loadcase title to be modified. Note that loadcases are tabulated in the order listed in the 🔍 Treeview.
- ❏ **Set Active** Sets the active loadcase for the current window.
- ❏ **Close Results File** closes the open results file
- ❏ **Deassign** Deassigns attributes from the loadcase, by choosing from a list of attribute types.
- ❏ **Controls** Allows the analysis control for a loadcase to be defined.

Loadcases are solved in the order they appear in the 🔍 Treeview (from top to bottom). Loadcases may be reordered using drag and drop in the 🔍 Treeview. Loadcases can only be re-ordered when no results files are loaded.

Setting the Active Loadcase

An important concept is the active loadcase in the current window. The active loadcase is a window property, and is the loadcase that all results and visualisations will be generated from. This speeds up the process of comparing results and visualising loads and supports as a different windows can be used for each loadcase.

The active loadcase is set from the 🔴 Treeview using the context menu. A loadcase icon changes from being greyed-out to coloured when made the loadcase is set active. When modelling, the active loadcase is denoted by a coloured loading icon. When results are loaded the active loadcase is denoted by a coloured contoured results icon.

Viewing the Assignments in a Loadcase

Loadcase dependent attribute assignment are displayed in the 🔴 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 in the 🔴 Treeview. The attribute may be visualised for the active loadcase from the context menu using the **Visualise Assignments** item.

Note: Within a linear analysis, with the exception of loading, attributes assigned to the first loadcase apply to all loadcases in the analysis. For nonlinear and transient analysis many attributes may be modified 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.

Load Combinations or Envelopes

Combinations and envelopes can be defined as part of the modelling process prior to carrying out an analysis, or after carrying out an analysis. For more details see **Combinations and Envelopes** in the Viewing the Results section.

Load Curves

Load curves can be used to describe the variation of the loading in nonlinear, transient and Fourier analyses. For example, in a transient problem the loading changes with time, in a nonlinear problem the loading level varies with load increment and in a Fourier analysis the loading varies with angle.

Load curves are used to simplify the input of load data in situations where the variation of load is known with respect to a certain parameter. An example of this is the dynamic response of a pipe to an increase of pressure over a given period. The load curve factor would then consist of the variation of pressure with time.

Creating Load Curve Entries

New load curve entries may be added to the 🔴 Treeview in the following ways:

- By creating a load curve from the **Utilities** menu.

- By right-clicking on a Loadcases, Structural or Thermal entry in the 🕐 Treeview and selecting the New load curve menu item.

- By copying and pasting an existing load curve in the 🕐 Treeview.

Defining Load Curves

A load curve is defined either using a user defined time vs factor curve, a standard sine, cosine or square wave curve, or a variation.

❑ **Time vs Factor** specify values for both in a table on the load curve dialog.

❑ **Sine, cosine, square wave** input values for peak (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.

Load curves scale all loads assigned to them. Therefore, if loads have a different variation of load factor with time, several load curves should be used.

Manipulating Load Curves

General load curve editing commands are available from the context menu that is activated by right-clicking on a load curve in the 🕐 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 🕐 Treeview.

❑ **Delete** Deletes a load curve. Attribute assignments must be deassigned before a loadcase can be deleted. At least one loadcase will always exist in the Treeview.

❑ **Rename** modifies the load curve name.

❑ **Edit** changes previously entered load curve data.

Notes

- Load curves are only applicable to nonlinear, transient and Fourier **analyses**.

- When defining load curves for transient (or nonlinear) analyses the time in all load curves must be defined from the start of the analysis.

- For **Fourier analysis** the load must only be applied over an angular range of 0 to 360 degrees.

- If the interpolation variable doesn't lie within that specified within the load curve a zero load factor will be applied.

- Only line variations with distance type **Actual** can be used for defining load curves.

Evaluating a Load Curve

Load curves (and variations) may be viewed using the **Graph Wizard**.

Case Study. Pressurisation of Tanks with Multiple Load Curves

Two tanks are to be pressurised at different stages of a nonlinear dynamic analysis. This will be achieved using two different loadcases and two load curves to vary the loads individually. The following procedure outlines the steps required:

1. Use the **Utilities> Load Curve** menu item to define two user defined load curves which give the correct pressure variation with time. Note that the time is always from the start of the analysis.

2. Use the **Attributes> Loading> Structural** menu item to define a face load attribute containing a unit pressure load. Note that the pressure value in the load definition will be multiplied by the load factor used on the load curve associated with it.

3. Assign the face load to the features in the model, selecting the appropriate load curves for each tank. The accompanying diagram shows a schematic of the tanks under internal pressure with their corresponding force versus time graphs.

4. To set the size and number of time steps right click on **Loadcase 1** and choose **Controls>** from the context menu. Pick **Nonlinear and Transient** and on the dialog switch on the **Time Domain** option and choose **Implicit Dynamics** from the combo. Set the initial time step and response time as required.
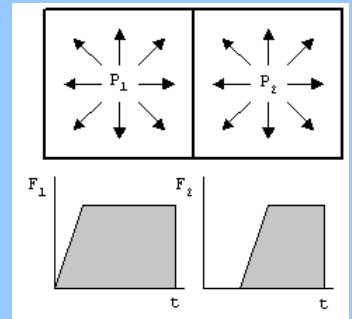
Notes

Supports should be assigned to the loadcase. If the support conditions are to be modified part way through the analysis the response time in the Nonlinear and Transient control should be set to terminate at the time the supports are to be modified and a second loadcase should be created to which the modified supports are assigned. A Nonlinear and Transient control should be then set on this loadcase which terminates at the end of the analysis or when a future support is to be modified.

Chapter 6 : Utilities

About Model Utilities

Model utilities differ from model attributes in that they are not intended for assignment to the model geometry. A utility, however, may be used in the definition of geometry or attributes, or to control an analysis, or to provide a particular functionality, such as to define a load combination or produce a report for example.

Details of the features and use of many of the utilities listed below are provided in other relevant chapters of the manual. Annotation utilities, for example, are described in the Chapter 2: Using Modeller in the Annotating the Model section. The remainder are described in this chapter.

The complete list of LUSAS utilities (accessed from the **Utilities** menu) is shown below.

- ❑ **Mesh** - mesh node measurement, the controlling of automatic remeshing. and saving a deformed mesh for re-use in a new analysis
- ❑ **Annotation** - adding of text, line, bitmap and border annotation to the view window
- ❑ **Transformation** - moving, mirroring and copying of geometry
- ❑ **Heat transfer** - specification of thermal gap properties and radiation surfaces
- ❑ **Variation**- varying attributes over features
- ❑ **Reference Path** - defines a path along which a set of multiple varying sections can be assigned.
- ❑ **Loadcase** adds a new empty loadcase to the loadcase treeview
- ❑ **Load Curve** - used to describe the variation of the loading in nonlinear, transient and Fourier analyses
- ❑ **Envelope** - create an envelope of maximum and minimum effects
- ❑ **Combination** - combine results from different loadcases with different load factors to get max and min effects
- ❑ **Fatigue** - calculate fatigue life and number of cycles to failure
- ❑ **IMD Loadcase** - create a loadcase for use with interactive modal dynamics
- ❑ **IMDPlus** - the investigation of various dynamic responses using the results from an eigenvalue analysis
- ❑ **DesignFactors** - assess the reserve strength capacity of a component or structure

❑ **Set Fourier Angle** - specify the angle around the circumference at which Fourier results are required

❑ **Background Grids** - grade the mesh pattern locally when irregular surface meshing

❑ **Graph Wizard** - plot results on x,y graphs

❑ **Animation Wizard** - animate the mode shapes or load history of a structure

❑ **Section Through 3D** - slice a model and plot results on the section defined

❑ **Slice Resultants Beams/Shells** - etc

❑ **Graph Through 2D** - slice a model and plot a graph based upon the intersection of the elements sliced.

❑ **Print Results Wizard** - printing of selected results to a grid or a file.

❑ **User Defined Results** - create results components from user-defined expressions

❑ **Vertical Axis** - sets the model X, Y,or Z direction to be the vertical axis.

❑ **Direction Definition** - sets the vertical, longitudinal and transverse axes for a model to assist with model orientation and calculation of particular effects

❑ **Library Management** - specify library locations and add and delete items from a library

❑ **Section Property Calculation** - calculate cross sectional geometric properties for a range of sections

❑ **Report Generation** - build reports containing model and results data and images from your model

Some utilities, for example Heat transfer, are only listed if the appropriate user interface is in use.

Variations

Variations allow parameters in attributes to be varied over the assigned geometry by defining the manner in which the parameter will vary. If a variation is not specified, the parameters within an attribute will be constant over the geometry to which the attribute is assigned. Geometric section property variations along a beam are best defined using tapering or multiple varying section facilities.

The different types of variation are available are:

❑ **Field** allowing variations to be defined in terms of the global Cartesian coordinate system variables. This form of variation can be used for hydrostatic and wind loading and is applicable to **all feature types except Points**. Variations on volumes are limited to field variations.

❑ **Interpolation** variations may be applied to **Lines** and **Surfaces**. The variation is defined by interpolating between values at specified local distances. The order of the interpolation may be specified as constant, linear, quadratic and cubic in either actual (local) or parametric distance.

❑ **Function** variations are expressed as symbolic functions in terms of the parametric coordinates of a feature. They can be applied to **Lines** and **Surfaces**. For Lines, the parametric distance is the distance along the Line (u), and for a Surface the distances are the local parametric u and v coordinates.

❑ **Boundary** defines values by specifying variations around the Surface boundary Lines.

❑ **Grid** defines a grid of values in Surface local x and y directions.

Using Variations

Variations are defined from the **Utilities** menu and are presented in the 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.

Notes

❑ It is possible to vary all load types except General Point and Patch loads and Internal Beam Point and Internal Beam Distributed loads, which incorporate variable loading implicitly in their definition. Values of loads which are applied to elements will be evaluated at the element centroid.

❑ Geometric attributes containing a variation are tabulated as multiple geometric properties. An additional parameter is added to the assignment to relate to the original defining attribute number for use in post-processing.

❑ To vary geometric properties along bar or beam elements use the geometric beam tapering facility.

❑ Variations in materials are limited to elastic material values and certain joint properties. Attributes containing a variation are tabulated as multiple material properties containing the material value calculated at the element centroid. An additional parameter is tabulated to the assignment data chapter in the data file to relate to the original defining attribute number for use in post-processing.

❑ When defining supports the spring stiffness values can be varied but the spring stiffness values are not scaled when drawn in post-processing .

❑ Checking of the assigned variations can be carried out by contouring the assigned data using an unsmoothed contour display.

❑ Variations of the Rayleigh parameters cannot be contoured as they are calculated at element centroid positions.

Field Variations

Field variations allow a variation according to a mathematical expression in terms of coordinate variables in either the global Cartesian or a specified **local coordinate**. Coordinates may be Cartesian, cylindrical or spherical. The expression may be cutoff if desired.

Field variations are applicable to all Lines, Surfaces and Volumes. The value of the variation at any position on the structure will be calculated by substituting the values of the coordinate variables at that position.

A field variation is defined by specifying a field expression and an optional local coordinate which will be used to specify a coordinate system other than the global Cartesian set.

These examples show field variations expressed in terms of the global X coordinate displayed along a Line parallel to the global X axis. The typical field expressions used are shown in the boxes next to each diagram.

For example, a field expression in Cartesian coordinates would typically be:

   `-9.81*y`

and in cylindrical coordinates:

   `10+r*tan(thetaz)`
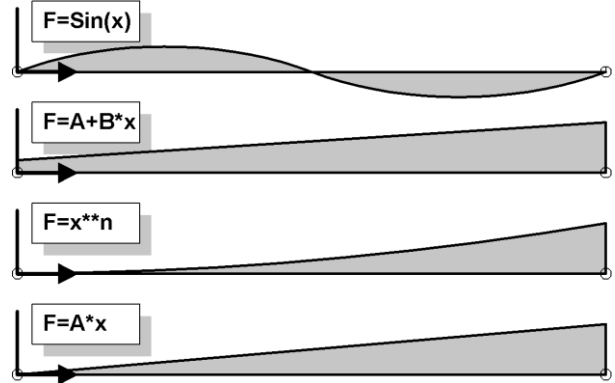
Coordinate Systems in Field Variations

The functions available in the definition of field expressions are listed below. The variables used in field expressions are limited to those used in the LPI language plus the Cartesian, cylindrical and spherical coordinate variable names. The coordinate variable names that should be used in a field expression are dependent on the type of coordinate systems in use. Definitions are given in the tables below.

In this example, a field expression referring to the global axis coordinates (XY), is also used with a local coordinate axis set (indicated by xy) to create a variation relative to a rotated system. Cylindrical and spherical axis sets can also be used.
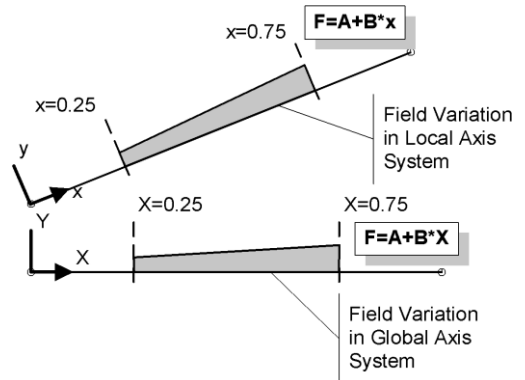
Variables

Cartesian (global/local)		Z Cylindrical (local)		Z Spherical (local)	
x	X ordinate	r	Radial distance	r	Radial distance
y	Y ordinate	thetaz	Angle about axis of cylinder	thetaz	Angle about z axis
z	Z ordinate	z	Distance along cylinder longitudinal axis	thetac	Second angle

Operators

`+ - * / ^`

Functions

Trigonometric functions	Radians	Degrees
Sine of angle	sin(angle)	sind(angle)
Cosine of angle	cos(angle)	cosd(angle)
Tangent of angle	tan(angle)	tand(angle)
Arcsine of a	asin(a)	asind(a)
Arccosine of a	acos(a)	acosd(a)
Arctangent of a	atan(a)	atand(a)
Arctangent of the specified x- and y-coordinates.	atan2(x,y)	
hyperbolic sine of a	sinh(a)	
hyperbolic cosine of a	cosh(a)	
hyperbolic tangent of a	tanh(a)	

Function return value

e raised to the power of a. The constant e equals 2.71828182845904, the base of the natural logarithm.	exp(a)
natural logarithm of a	log(a)
logarithm of a to base 10	log10(a)
square root of a	sqrt(a)
a rounded up, away from zero, to the nearest integer	ceil(a)
a rounded down, towards zero, to the nearest integer	floor(a)
absolute value of a	abs(a)
maximum value of a and b	max(a,b)
minimum value of a and b	min(a,b)
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.
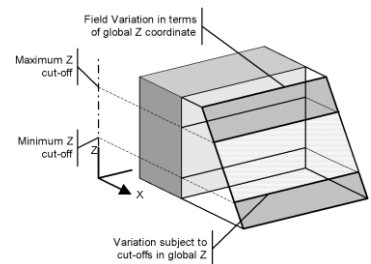
Case Study. Applying Hydrostatic Loading

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.
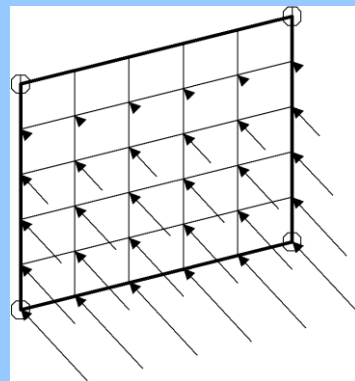
Line Interpolation Variations

Line interpolation variations allow values to be varied along a line at any number of distances. The distances may be equally or unequally spaced and the interpolation order my be constant, line, quadratic or 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.
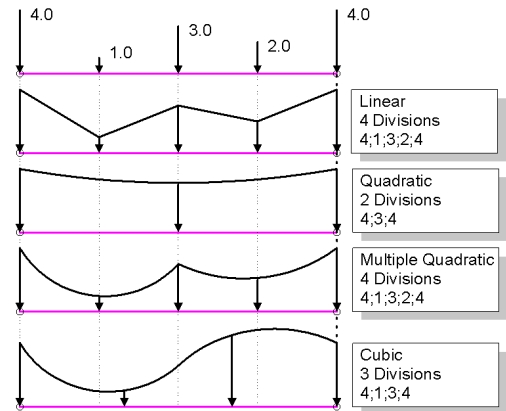
❑ **By Unequal Distances** defines values at specified distances along a Line. The distances can be entered as actual or parametric values. The actual value used will be interpolated at the appropriate distance between these values using the interpolation method specified.

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.
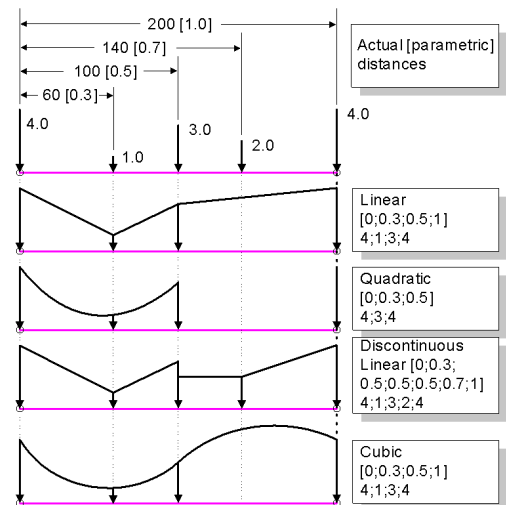
Linear variations require a minimum of two values. Quadratic variations require a minimum of three values and Cubic variations require a minimum of four values to be specified. Where

more values are specified multiple interpolation functions are used. i.e. if three values are specified for a linear variation, two straight line interpolations are used.

Line Function Variations

A Line function variation defines a variation by a series of functions specified at distances along a Line. The function is specified in terms of the parametric or actual coordinate along the Line. The interpolated value of the variation at any position along the Line is calculated by finding the interval in which the position occurs and then substituting the parametric or local distance into the function. Line function variations are defined from the **Utilities> Variations> Line** menu item and selecting Type **Function**.

❑ **By Equal Distances [in u]** defines functions in terms of u, the parametric distance along the Line. In this example, the Line is split into a specified number of distances, each with an associated function.
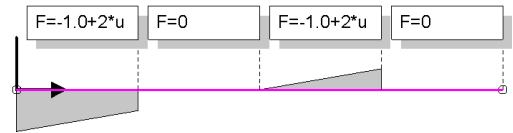
❑ **By Unequal Distances [in u]** defines a series of parametric or actual distances, and a set of functions. The

distance specified is the starting position for the function associated with it. Each distance must have an associated function specified. To enter a maximum cut-off position, associate a zero function with it. In this example, a parametric distance of 0 is associated with the value 0.0, a parametric distance of 0.33 is associated with u**2 and a parametric distance of 0.92 is associated with the value 2.0
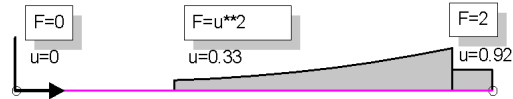
Surface Variations

On Surfaces, interpolation may be defined using a grid of values or a set of line variations applied to the boundary Lines. For interpolation by grid the interpolation order may be constant, linear, quadratic or cubic.

❑ **Surface By Grid** defines a grid of values in Surface local x and y directions. Surface grid interpolation can only be used on 3 and 4 sided Surfaces.

- **Quadratic vs Constant Surface variation**

  The quadratic variation in the local x direction is specified with three interpolation points. The constant variation in the local y direction requires no additional points. A total of three values are required.

  Local Y Constant 1 Division (1 Point)

  Local X Quadratic 2 Divisions (3 Points)
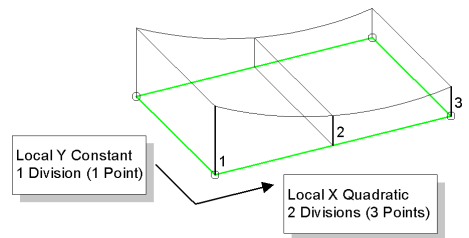
- **Cubic vs. Linear Surface grid variation**

  The local x direction takes a 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.

  Local Y (Linear) 2 Divisions (3 Points)

  Local X (Cubic) 3 Divisions (4 Points)
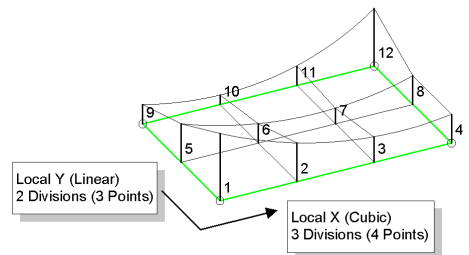
❑ **Surface By Boundary** defines values by specifying variations around the Surface boundary Lines. A variation must be specified for each Line in the Surface definition. If no variation is required along a Line, a constant order variation must be specified. Care must be taken to ensure that values at common points are common to both variations meeting at that point otherwise an error will occur. Variations are defined in the same direction on opposite sides of the Surface (see the following example) and use the Line order in the Surface definition on which to base the variation direction. Individual Line directions have no effect on variation directions.

❑ **Surface boundary interpolation using three Line interpolation variations.**

A discontinuous Line interpolation (1) is specified for first and third Lines using a Line by unequal distance variation.

Note that the Line axes drawn here dictate the variation directions and the line directions on opposite sides of a surface must match as shown. Variations are applied in a positive surface normal direction.

Same Variation As First Line (1)

Line Interpolation Constant (2) Value = 3

Line Interpolation (3) Constant Value = 4

Line Interpolation Discontinuous Linear (1) Values = 4-5, 3-2, 4-3
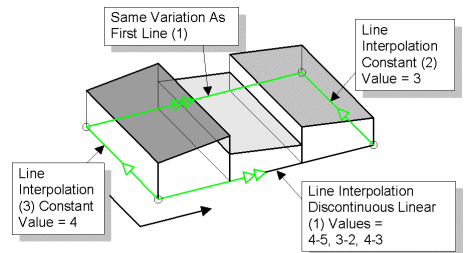
The second and fourth Lines in the Surface definition use constant interpolation variations. The variation sense is denoted by double and single arrows shown on boundary Lines. The variation along the local x axis (signified by the double arrow) is specified first. The Surface variation in this case is 1;3;1;2.

Surface Function Variations

A Surface function variation consists of a single function in terms of the parametric coordinates of the Surface u and v. The value of the variation at any point on the Surface is given by finding the parametric coordinates of the point within the Surface and substituting them into the specified function. Surface function variations are only allowed for 3 and 4 sided Surfaces.

The example shown here defines a variation using the function **max(4,10*u)** in terms of the local Surface x direction parametric distance. The **max** function takes two arguments and returns the maximum of both arguments. In this case, 4 is the maximum value until u exceeds 0.4.
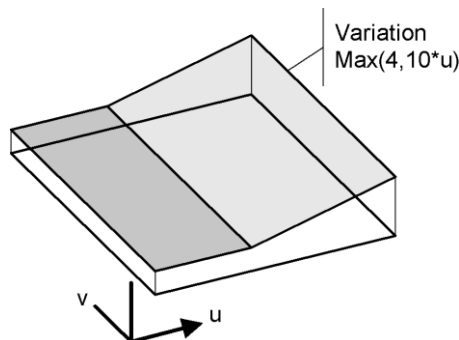
Plotting Graphs of Line & Field Variations

Line and Field variations can be evaluated along a specified Line and displayed using the **Graph Wizard** from the **Utilities** menu and Line variations can be evaluated alone. The number of points at which to sample the variation can be specified. A factor may be applied to the variation values before the ordinates are calculated. This example demonstrates the graphical visualisation of a discontinuous Line interpolation variation.
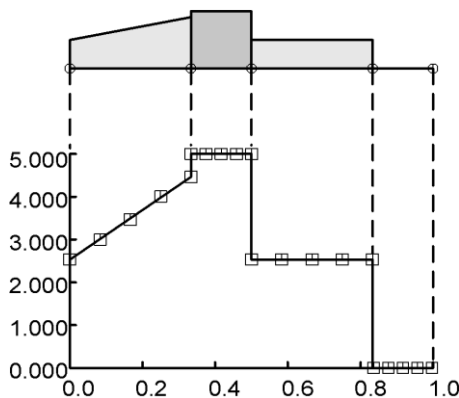
Reference paths

A reference path defines a route through the model that provides a concept of distance to each point in the model. Those distances can be used in the definition of a varying section, such that when the section is assigned to lines, the path is used to interpret which part of the section is appropriate to each line. Bridge engineers refer to this reference path concept as 'chainage'.

Once created, the data that defines the 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.

Uses

Reference paths are primarily used for line beam models (such as those that are required for staged construction analysis) and for use with grid or grillage models where longitudinal and transverse beams are modelled with individual grillage or line beam elements.

Defining paths

Paths are created by using the **Utilities> Reference Path** menu item.

In its simplest form a reference path can be defined as a line between two points (quite separate from the model data) if a straight path is to be considered, or be created from the model geometry itself and contain as many defining points as the lines from which it has been created. If the latter is done it is important to remember that the model geometry has been used to arrive at the points required to generate a reference path but 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.

Reference paths can be defined by:

- Specifying the coordinates of each defining point.

- Importing geometry and line segment information from a spreadsheet

- By defining a path from lines, arcs and splines in LUSAS Modeller.

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.

'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 equates to specifying a chainage value for a known setting-out point. This value is added to the distance value that can be displayed for each of the points defining the path.

Reference paths for 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)**
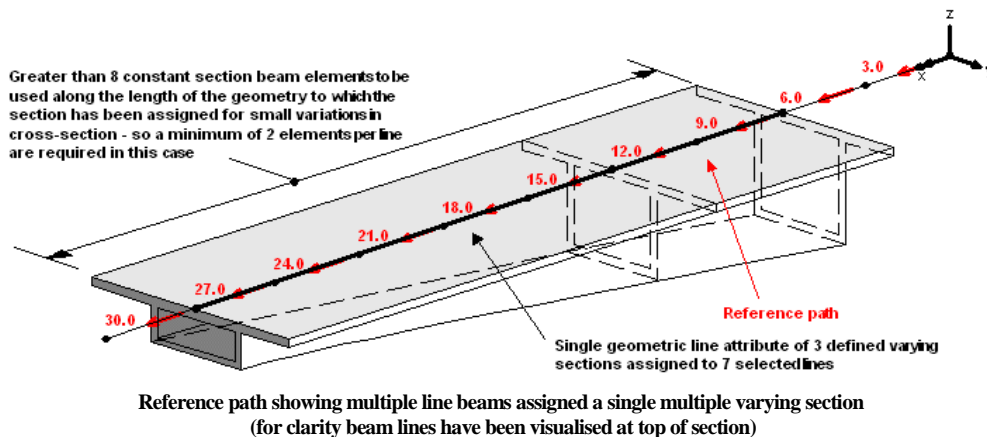
Reference paths for 3D grid/grillage-type models

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
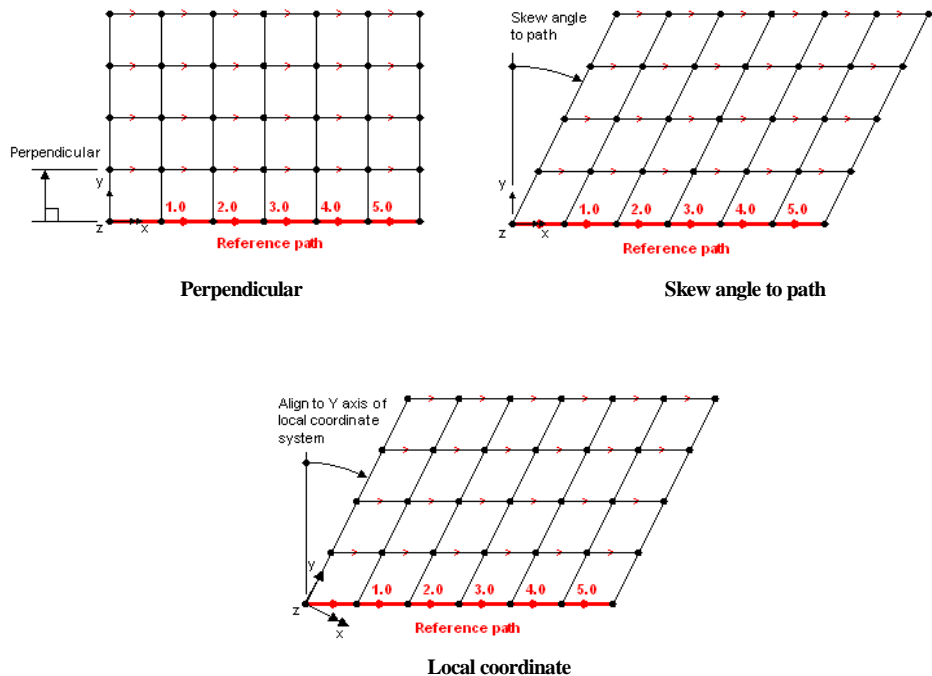
Visualisation of reference paths

By default, reference paths are drawn in red and points defining the path are labelled with their absolute distance along the path of lines. The display of path labels can be turned on and off by accessing the Utilities layer properties dialog. Direction arrows, at mid-points along each line segment, show the direction of the path.

Each path entry in the Utilities treeview has a context menu enabling the following selections:

- ❑ **Rename** changes the path name
- ❑ **Delete** removes the path entry from the Utilities treeview but the menu item is only available if a path has not been associated with a geometric line assignment.
- ❑ **Edit Path** displays the path definition dialog to enable changes to be made
- ❑ **Create geometry** converts the path into points, lines, arcs and splines in the Geometry layer.
- ❑ **Visible** turns the display of the reference path on, if off.
- ❑ **Invisible** turns the display of the reference path off, if on.
- ❑ **Visualise at points** shows what would happen if the path were associated with other lines on the model and requires explaining in more detail. See below.

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. Note that currently reference point labels do not update if the underlying geometry is updated.
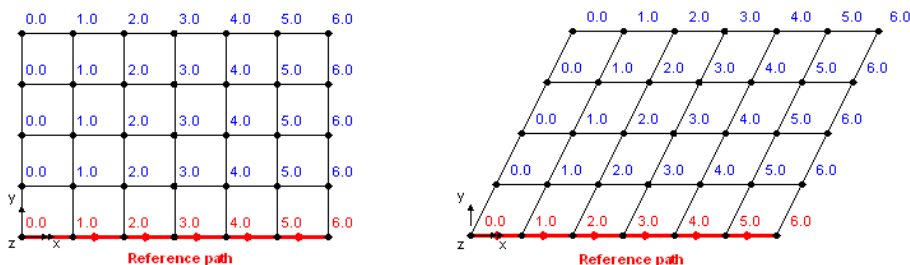
Perpendicular (orthogonal) grid/grillage   Skewed grid/grillage

Examples of valid reference path points visualised on the model

Influence Attributes

Influence type, influence direction and displacement direction are used to define an influence attribute for use in an **influence analysis**. These parameters define the type of behaviour of the structure at and around an influence point. 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.
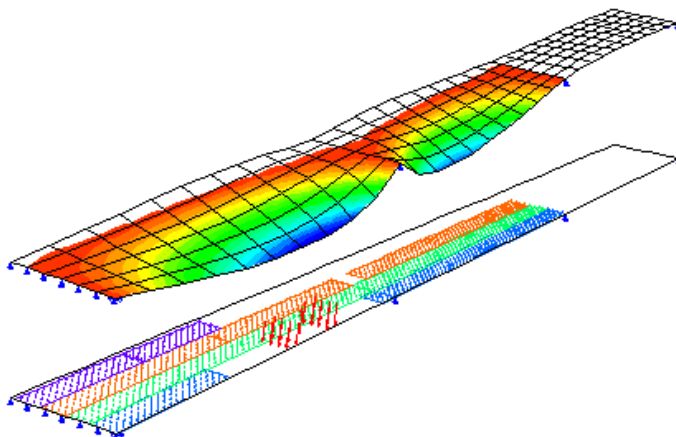
Defining Influence Attributes

Influence attributes are defined from the **Attributes > Influence** menu item. The influence type may be a Shear force, a Reaction, a Moment or a Displacement. A shear or moment influence type is mesh dependent. For each influence type the influence direction and displacement directions need to be specified. The influence direction defines the axis to be used. Once created, an influence attribute is held in the Attributes 🍄 Treeview for assignment to mesh nodes or Points on a model using the standard select, then drag and drop method.

Assigning Influence Attributes

One or more nodes or Points on the model may be selected to make an influence point assignment. Assigned influences are stored in the Utilities 🔧 Treeview. When assigned to the model LUSAS automatically determines the break-away elements in each case. Influence type symbols may be drawn at each influence location to show the type of mesh break that is being used. 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 Utilities 🔧 Treeview. Models can be solved with unassigned influences present in the Utilities 🔧 Treeview.
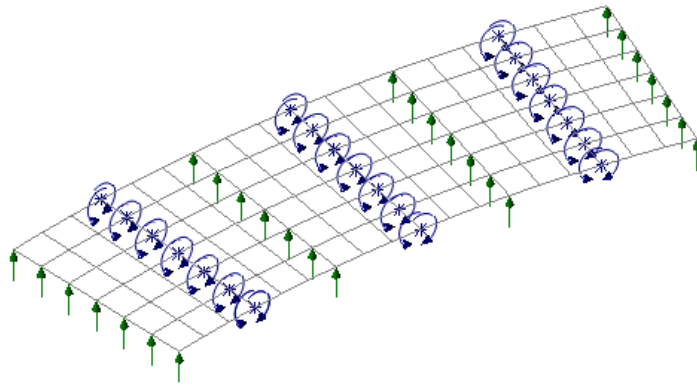
Manipulating Influence Attributes and Influence Point Assignments

 Influence attributes are listed in the Attributes 🍄 Treeview and assigned influences are shown in the Utilities 🔧 Treeview. If influence assignments are visualised the whole model may be re-displayed after viewing selected influences by right-clicking in the Graphics Window and selecting the All Visible option.

Writing a Datafile with Influence Attributes

Once all influence attributes have been assigned to a model, they can be tabulated to a data file using the menu option **Files> LUSAS 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. An influence analysis can also be run from the Civil or Bridge product menu using the **Run Influence Analysis** option.

Viewing Influence Surfaces

After carrying out an influence analysis the deformed shape for each influence point may be viewed / checked by adding a deformed mesh layer to the Layers 🔲 Treeview and then, in the Loadcase 🔍 Treeview, setting each loadcase active in turn.

See **Vehicle Load Optimisation** for details of how to generate the most adverse loading for defined influence surfaces for bridge loading.

Vertical axis

The vertical axis dialog is accessed from the **Utilities> Vertical Axis** menu item. The vertical axis setting specifies whether the model X, Y,or Z direction is to be used as the vertical axis. It has the following uses:

❑ It is 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 determine the direction that gravity loading will be applied if it has been specified as a property of a loadcase.

❑ 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 that is viewed when using the isometric, dimetric and trimetric views.

Note that setting the vertical axis on the Vertical axis dialog will supercede any vertical axis setting defined on the Direction definition dialog.

Direction Definition

Specifying a direction definition sets the vertical, longitudinal and transverse axes for a model to assist with model orientation and the calculation of particular effects. The Direction definition dialog is accessed from the **Utilities> Direction Definition** menu item. The options are:

Vertical

❑ **Global axis** This determines the direction that gravity loading will be applied if added using the **Bridge >** or **Civil > Gravity** menu item. It also defines the initial vertical axis and orientation of element types and library items as displayed on the Geometric Line dialog prior to them being added to a model. It also defines the model orientation viewed when using the isometric, dimetric and trimetric views.

Note that setting the vertical axis on the Direction definition dialog will supercede any vertical axis setting defined on the Vertical Axis dialog.

Longitudinal

❑ **Global axis** For the majority of models this will simply be the global X-axis. However LUSAS allows generic input of any direction which may even be a complex path through or along a structure, such as that defined by a set of lines forming a continuous path.
❑ **Local axis** Use a local coordinate set to define the direction. An example of use is for aligning influence attributes along a singly curved bridge deck.
❑ **Follow line path** If a path of lines is to be used, selecting the lines to be used prior to selecting **Utilities > Direction definition** will cause the correct line path definition to be automatically inserted into the line path field. An example of use is for ensuring correct alignment of influence attributes along all spans of a bridge deck when those spans are formed of multiple straight lines, arcs or any combined sequence of these two feature types in order to describe the carriageway shape.

Note that a longitudinal direction definition must always be correctly defined for **influence analysis**.

Transverse

This is assumed to be orthogonal to both longitudinal and vertical directions. This is currently only used for **influence analysis.**

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 that are drawn in LUSAS Modeller
❑ **Precast beam sections** with and without a concrete slab.
❑ **Box sections** for both simple and complex box sections, with and without an internal void.

Standard Section Property Calculator

Standard section property calculators are accessed from the **Utilities> Section Property Calculator** menu item. The following sections are supported:

❑ **Rectangular solid section** - equal and unequal thickness
❑ **Rectangular hollow section** - equal and unequal flange / web thicknesses
❑ **Circular solid section**
❑ **Circular hollow section**
❑ **I section** - equal and unequal flanges, haunch section
❑ **T section**
❑ **L section** - single and double (back to back)
❑ **C section** - lipped, unlipped, double (back to back), double (face to face), top hat
❑ **Z section** - lipped right-angle, lipped inclined, unlipped
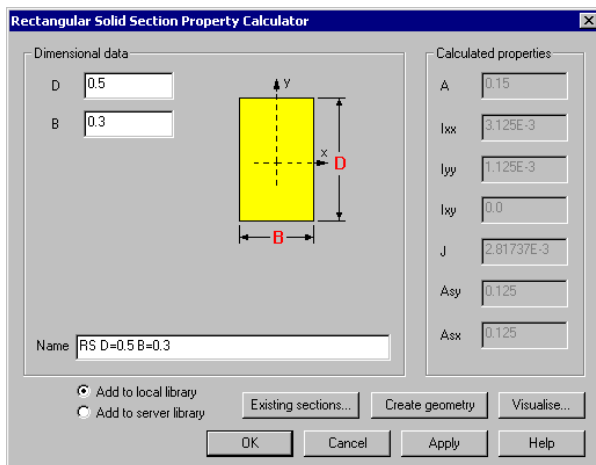
| **Typical standard section property calculator dialog** | **Section visualisation showing fibre locations** |
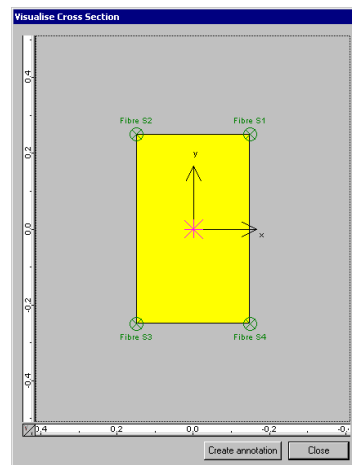
Section properties for standard cross-sections are computed instantaneously once valid user-defined dimensional data has been entered. The resulting section can be optionally visualised to check for correct values being entered and to see automatically defined fibre locations (as used when plotting stresses on fleshed beams); or be converted into model geometry if the section was to be modified in some way inside LUSAS Modeller before re-calculating the new section properties of the edited section using an Arbitrary Section Property Calculator; or be added to a local or server library for use on the current project or for re-use across other projects.

 To use the computed section properties in a model the section must be saved to a local or server library. To add a library item to the Attributes 🍥 treeview select the **Attributes>**

Geometric> Section Library menu item, then select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available.

Arbitrary Section Property Calculator

The arbitrary section property calculator is accessed from the **Utilities> Section Property Calculator> Arbitrary Section** menu item. It computes the section properties (area, moments of inertia and torsion constant) of any open or closed section and calculates extreme fibre positions for use when plotting stresses on fleshed beams. The torsion constant is computed using the soap bubble analogy which involves running a field analysis using LUSAS Solver. This process is carried out automatically.
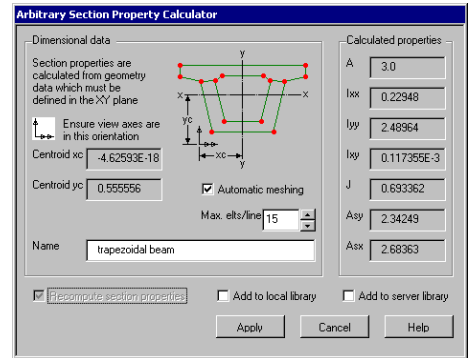
Cross-sections must be defined in the XY plane and can be created using points and lines, but the cross-section must ultimately only contain surfaces that define a single continuous shape. Individual surfaces separated by gaps do not form a valid cross-sectional shape for section property calculation purposes. Voids or holes in a section must be defined as separate surfaces. Any number of voids or holes can be included in a cross section but the section shape created must always be continuous. The use of the menu option **Geometry> Surface> Holes> Create** will effectively 'punch' a hole into an existing surface. A Boolean subtraction of a smaller surface from a larger surface would also produce the same result. Grouping all holes together into a group named **Holes** is optional for cases where a single surface contains one or more holes totally inside its boundaries. Grouping all holes together into a group named **Holes** is <u>essential</u> if a hole exists between two surfaces (See diagram). When present, the properties of a group called Holes are automatically deducted from the overall section property calculation.
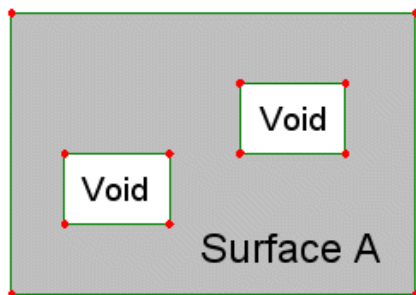
Surface defining a cross-section containing two holes
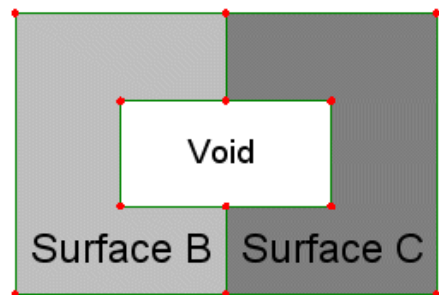
Two surfaces defining a cross-section with a hole between surfaces

Automatic meshing is normally used to control the mesh density which in turn is used to calculate the section properties. The maximum elements to be assigned to any one line helps control the density of the automatic mesh used.

To use the computed section properties in a model the section must be saved to a local or server library. To add a library item to the Attributes 🎯 treeview select the **Attributes> Geometric> Section Library** menu item, then select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available. The geometric properties can then be **assigned** to the required Line(s) in the model.

Notes

- 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 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 adjusted.

- For thin curved sections the shear areas calculations are approximate.

- Sections constructed of two or more materials can be accommodated using the modular ratio approach. Firstly an isotropic field material property attribute is created with both the thermal conductivity and the specific heat set to unity and this attribute is assigned to surfaces made from the primary material. Further material property attributes are then defined for each secondary material with the thermal conductivity set to G1/G2 and the specific heat set to E1/E2.

   where:
   G1 is the shear module of the primary material.
   G2 is the shear modules of the secondary material.
   E1 is the Young's Modulus of the primary material.
   E2 is the Young's Modulus of the secondary material.

   These material attributes should be assigned to the appropriate surfaces before running the arbitrary section property calculator.

- User-defined beam cross-sections need to have their fibre definitions defined manually in order for stress results plots to be created for fleshed beams.

The use of the arbitrary section property calculator is described further in the worked example; Arbitrary Section Property Calculation and Use. See *LUSAS Examples Manual.*

Precast Beam Section Generator

The **Precast Beam Section Generator** 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.

Library Management

Libraries are used to store standard section and materials properties. The location of the section and material libraries may be defined from the **Utilities> Library Management> Library Locations** menu item. The local section library is always located in the current working (project) directory while the server library may be located anywhere on the computer network.

Add Section to Library

Basic geometric section properties may be manually added to either the local or server section library from the **Utilities> Library Management> Add Section** menu item. This facility is intended primarily for entering section data of an unspecified cross-sectional shape. Section property calculators exist for calculating section properties of known cross-sectional shapes.

Delete Section from Library

Section properties may be deleted from either the local or server section library from the **Utilities> Library Management> Delete Section** menu item.

Chapter 7 : Running an Analysis

Preparing the Model for Analysis

By default, LUSAS will perform a linear static stress analysis. Any other type of analysis requires the analysis control to be specified. Analysis controls are properties of **loadcases**, and loadcases are displayed in the Loadcase Treeview 🔴.

The model title and units are defined on the Model Startup dialog. Consistent units must be used for all analyses.

The default solver is the standard frontal solver and is used unless the fast solver option has been licensed in which case the fast multi-frontal solver is used. An alternative solver may be set from the **Solver Options** dialog under the **Model Properties> Solution** tab.

Solution Options may be set from the **Element**, **Nonlinear** and **Coupling** dialogs under the **Model Properties> Solution** tab.

Frontal optimisation is not required for the fast multi-frontal solver and is off by default. When no optimiser is specified the Sloan optimiser will be used to optimise the front width for the standard frontal solver. An alternative optimiser may be selected from **Optimiser Options** dialog under the **Model Properties> Solution** tab.

Analysis Types

LUSAS may be used to numerically model a wide range of engineering problems. The following section briefly explains the analysis types available.

❑ **Linear Analysis** is the most common analysis carried out by engineers and unless specified otherwise, LUSAS will perform a linear elastic, static analysis or **steady state field** analysis. 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.

For other analysis types control parameters must be specified as properties of the loadcase.

❑ **Nonlinear Analysis** is used to model significant changes in geometry, material or boundary conditions. Significant geometry deformation may occur due to the applied loading. Changes in material may occur due to material yield. Changes in boundary conditions may occur due to the lift-off of supports or from changes in contact or frictional behaviour. Examples of nonlinear analyses include **Creep Analysis** and **Impact Dynamics**.

❑ **Transient analysis** is used to carried out analyses over a period of time and is progressed in a step-by-step manner, giving results at each time-step. Both Transient Dynamic Analysis and **Transient Thermal Analysis** are available.

❑ **Eigenvalue Analysis** is available to compute the **Natural Frequencies** of a structure or to carry out an Eigenvalue Buckling Analysis in order to estimate the maximum load that can be supported by a stiff structure prior to structural instability. **Eigenvalue Stiffness** may also be performed on the stiffness matrix at a selected stage of an analysis. This facility can be used in conjunction with a nonlinear analysis to predict structural instability or bifurcation points during a geometrically nonlinear analysis.

❑ **Fourier Analysis** provides an extended form of axisymmetric analysis where applied loading can be considered to be non-axisymmetric when applied using a Fourier distribution around the circumference.

❑ **Thermo-Mechanical Coupled Analysis** either performs the thermal and structural analyses simultaneously or one after the other with transfer of data between them via an additional data transfer file.

The following analysis types are also possible but the tabulation of the 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.

The following analysis type is also available:

❑ **Influence Analysis** An influence line analysis produces a deformed shape which shows the variation of a chosen function (reaction, axial force, shear force, or bending moment) at any given point on a structure due to the application of a unit load at any point on the structure. In LUSAS the presence of assigned influence attributes on a model determines that an influence analysis will automatically be carried out.

About Nonlinear Analysis

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

Three types of nonlinear analysis may be modelled using LUSAS:

❑ **Geometric Nonlinearity** e.g. large deflection or rotation, large strain, non-conservative loading.

❑ **Boundary Nonlinearity** e.g. lift-off supports, general contact, compressional load transfer, dynamic impact.

❑ **Material Nonlinearity** e.g. plasticity, fracture/cracking, damage, creep, volumetric crushing, rubber material.

The LUSAS analysis types within which nonlinear geometric and material effects may be incorporated are shown in the following table:

Analysis Type	Geometric Nonlinearity	Material Nonlinearity
Static	yes	yes
Dynamic	yes	yes
Thermo mechanical	yes	yes
Creep	yes	yes
Natural Frequency	yes	yes
Eigenvalue Buckling		
Spectral Response		
Harmonic Response		
Fourier Analysis		
Field or Thermal		yes

Geometrically Nonlinear Analysis

Geometric nonlinearities arise from **significant changes in the structural configuration** during loading. Common examples of geometric nonlinearity are plate structures which develop membrane behaviour, or the geometric **bifurcation** of truss or shell structures. The changing application of loads or **boundary conditions** are also geometrically nonlinear effects. The figure below shows two simple structural examples which serve as good illustrations of geometrically nonlinear behaviour.

For the simply supported beam (top) the linear solution would predict the familiar simply supported bending moment and zero axial force. In reality as the beam deforms its length increases and an **axial component** of force is introduced.

For the loaded strut (bottom) the linear solution would fail to consider the **progressive eccentricity** of the vertical load on the bending moment diagram.

In both these cases depending on how large the deflections were, serious errors could be introduced if the effects of nonlinear geometry were neglected.
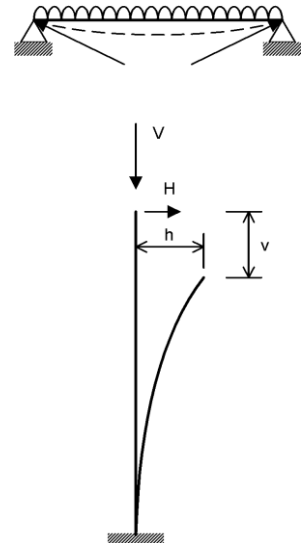
In LUSAS geometric nonlinearity is accounted for using four basic formulations:

❑ **Total Lagrangian**
❑ **Updated Lagrangian**
❑ **Eulerian**
❑ **Co-rotational**

These are defined from the **Model Properties> Solution** - Nonlinear Options tab. All four formulations are valid for arbitrary large deformations. In general, if rotational degrees of freedom are present, rotations must be small for Total Lagrangian. An exception to this rule is the Total Lagrangian formulation for thick shell elements where large rotations may be applied. Large rotations are allowed for Updated Lagrangian (provided that they are small within each load increment) or Eulerian. The co-rotational formulation is unconditionally valid for large rotations and results are generally independent of load step size.

All formulations are valid for small strains. For some elements the Updated Lagrangian formulation is valid for moderately large strains. The Eulerian formulation is also generally valid for moderate strains. In general, the Total Lagrangian is a more robust formulation, which is usually able to cope with substantial load increments. The Updated Lagrangian, and particularly Eulerian, formulations generally require smaller load increments in order to avoid a divergent solution.

Standard geometrically nonlinear formulations account for the change in position of the loading, but not the change in direction relative to the deformed configuration. Loading is always **conservative** for the Total Lagrangian geometrically nonlinear formulations (that is, the load is always applied in the same direction as was initially prescribed). Using an Updated Lagrangian formulation, the geometry is updated at the end of each increment and the applied loads may maintain the same relative orientation as to the original surface (depending on element and load types). Therefore **non-conservative** loading can be increment size dependent. True non-conservative loading may only be achieved by using the Eulerian and co-rotational formulations.

The choice of particular formulations is both problem and element dependent (the element formulation determining which strain formulations are available). The availability of each formulation is given for each element in the *Element Reference Manual*. For further details regarding the geometrically nonlinear formulations refer to the *Theory Manual*.

Nonlinear Boundary Conditions

Deformation dependent boundary condition models account for the modifications to the external restraints resulting from support lift-off, or smooth or frictional contact within an analysis. Within LUSAS node on node contact may be accounted for using **joint** elements and arbitrary contact may be accounted for using **slidelines**.

Consider the simple example shown in the figure right in which the structure and its supporting surface can resist being pushed together, but not being pulled apart. The required contact condition may be imposed by using joint elements to connect between the structure and the rigid support, and specifying a nonlinear contact joint model incorporating large and zero local stiffness in compression and tension respectively.
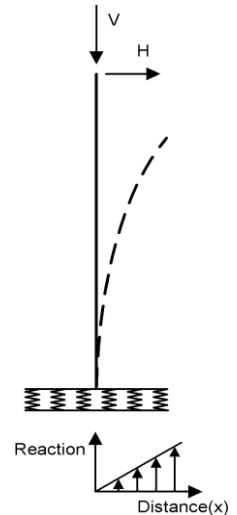
Materially Nonlinear Analysis

Materially nonlinear effects arise from a **nonlinear constitutive model** (that is, progressively disproportionate stresses and strains). Common examples of nonlinear material behaviour are the plastic yielding of metals, the ductile fracture of granular composites such as concrete, or time-dependent behaviour such as creep.

LUSAS incorporates a variety of nonlinear constitutive models, covering the behaviour of the more common engineering materials. Details of these material models and their applicability to each LUSAS element are described in **About Material Properties** which should be read in conjunction with the *Element Reference Manual*.

Nonlinear Solution Procedures

For nonlinear analysis, since it is no longer possible to directly obtain a stress distribution which equilibrates a given set of external loads, a solution procedure is usually adopted in which the total required load is applied in a number of **increments**.

Within each increment a linear prediction of the nonlinear response is made, and subsequent iterative corrections are performed in order to restore equilibrium by the elimination of the residual or out of balance forces.
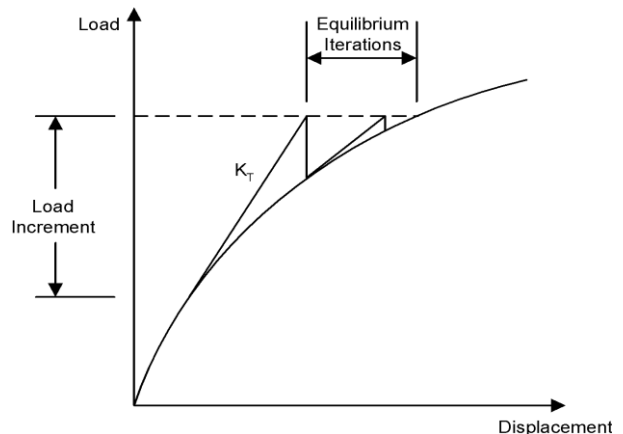
The iterative corrections are referred to some form of **convergence** criteria which indicates to what extent an equilibrium state has been achieved. Such a solution procedure is therefore commonly referred to as an **incremental**-**iterative** (or **predictor**-**corrector**) method shown in the figure above. In LUSAS, the nonlinear solution is based on the Newton-Raphson procedure. The details of the solution procedure are controlled using the nonlinear control properties assigned to loadcase.

For the analysis of nonlinear problems, the solution procedure adopted may be of significance to the results obtained. In order to reduce this dependence, wherever possible, nonlinear control properties incorporate a series of generally applicable default settings, and automatically activated facilities.

Iterative Procedures

In LUSAS the incremental-iterative solution is based on Newton-Raphson iterations. In the Newton-Raphson procedure an initial prediction of the incremental solution is based on the **tangent stiffness** from which incremental displacements, and their iterative corrections may be derived.

Standard Newton-Raphson Procedure

In the standard Newton-Raphson procedure each iterative calculation is always based upon the **current tangent stiffness**. For finite element analysis, this involves the formation (and factorisation) of the tangent stiffness matrix at the start of each iteration.

Although the standard Newton-Raphson method generally converges rapidly, the continual manipulation of the stiffness matrix is often expensive. The need for a robust yet inexpensive procedure therefore leads to the development of the family of modified Newton-Raphson methods.

Iterative Acceleration (Line Searches)

A slow convergence rate may be significantly improved by employing an **iterative acceleration** technique. In cases of severe and often localised nonlinearity, encountered typically in materially nonlinear or contact problems, some form of acceleration may be a prerequisite to convergence.

In LUSAS, iterative acceleration may be performed by applying **line searches**. In essence, the line search procedure involves extra optimisation iterations in which the potential energy associated with the residual forces at each iterative step are minimised. Line search application is controlled via parameters on the Iteration section of the Nonlinear Control properties.

The selection of line search parameters is problem dependent and largely a matter of experience. However, a maximum of 3 to 5 line search iterations with a tolerance of 0.3 to 0.8 is usually sufficient (the closer the tolerance is to unity, the more slack the minimum energy requirement).

Separate Iterative Loops

In problems where both material and contact nonlinearities are present, convergence difficulties can arise when evaluating material nonlinearities in configurations where the contact conditions are invalid because the solution is not in equilibrium. To avoid this situation contact equilibrium can be established using elastic properties from the previous load increment before the material nonlinearity is resolved. The option to define separate iterative loops is defined on **advanced solution strategy** dialog which can be found on the nonlinear control dialog.

See the *Theory* Manual for further details.

Incremental Procedures

For the Newton-Raphson solution procedures it is assumed that a displacement solution may be found for a given load increment and that, within each load increment, the load level remains constant. Such methods are therefore often referred to as **constant load level** incrementation procedures.

However, where **limit points** in the structural response are encountered (for example in the geometrically nonlinear case of **snap-through** failure) constant load level methods will, at best, fail to identify the load shedding portion of the curve and, at worst, fail to converge at all past the limit point. The solution of limit point problems therefore leads to the development of alternative methods, including displacement incrementation and constrained solution methods.

Constrained Solution Methods (Arc-Length)

Constrained methods differ from constant level methods in that the load level is not required to be constant within an increment. In fact the load and displacement levels are constrained to follow some pre-defined path.

In LUSAS two forms of arc-length method have been implemented:

❑ **Crisfields** modified arc-length procedure in which the solution is constrained to lie on a spherical surface defined in displacement space. For the one degree of freedom case this becomes a circular arc.

❑ **Rheinboldts** arc-length algorithm which constrains the largest displacement increment (as defined by the predictor) to remain constant for that particular increment.

The use of the arc-length method has the following advantages over constant load level methods

- Improved convergence characteristics

- Ability to detect and negotiate limit points

In LUSAS, control of arc-length solution procedures is via the Incrementation section of the Nonlinear Control properties. If required, the solution may be started under constant load control, and automatically switched to arc-length control based on a specified value of the current stiffness parameter (defined as the scaled inner product of displacements and loads). The required stiffness parameter for automatic conversion to arc-length control is input in the Incrementation section of Nonlinear Control properties.

Where limit points are encountered LUSAS will automatically determine the sign of the next load increment by the sign of the **determinant of the stiffness matrix**. This is a reliable method in most cases, however, it will often fail in the vicinity of bifurcation points when negative eigenvalues may cause premature unloading. In such cases the load reversal criteria may be optionally changed to be dependent on the sign of the **current stiffness parameter**. This method is better at coping with bifurcation points, but will always fail when a **snap-back** situation is encountered.

Note. In certain circumstances, notably in the presence of strain-softening, the arc-length method may converge on alternative, unstable equilibrium paths.

Bracketing Critical Points And Branch Switching

Bracketing can be used to locate a limit or bifurcation point during a geometrically nonlinear analysis. The nonlinear analysis is executed and one of three methods is used to isolate the first critical point.

- ❑ **Bi-section**
- ❑ **Interpolation**
- ❑ **Riks semi-direct approach**

Two further options for the bracketing procedure exist depending on whether the material response is elastic (reversible) or plastic/path dependent (irreversible). Only the first critical point can be processed and a subsequent eigenvalue analysis must be invoked to determine whether the critical point encountered is a limit or bifurcation point. A limit point may be defined as the point at which load starts to decrease as displacements increase (e.g. the snap through of a shallow arch). In this instance the structure may not have failed completely and could subsequently still be capable of carrying more load. A bifurcation point indicates that the solution of the nonlinear differential equations has encountered an alternative unstable solution path or paths which may be followed instead of the stable equilibrium path. The branch switching procedure must then be undertaken if an unstable equilibrium path is to be followed.

The branch switching procedure should only be carried out within a restart analysis after bracketing has been successfully completed. Two options exist for guiding the solution onto a secondary path.

- ❑ **Eigenmode injection**
- ❑ **Artificial force and Rheinboldts arc**

Incremental Loading

Incrementation for nonlinear problems may be specified in four ways:

- ❑ **Manual Incrementation** where the loading data in each load increment is specified separately.
- ❑ **Automatic Incrementation** where a specified loadcase is factored using fixed or variable increments.
- ❑ **Mixed Incrementation** Mixed manual and automatic incrementation.
- ❑ **Load Curves** where the variation of one or more sets of loading data is specified as a graph of load factor vs. load increment or time step.

The choice and level of incrementation will depend on the problem to be solved.

Automatic Load Incrementation

Two methods of automatic incrementation are available:

- ❑ **Uniform Incrementation** By default, uniform incrementation will be applied. That is, for each increment the current load factor will be multiplied by the specified load components to generate the applied load.
- ❑ **Variable Incrementation** Alternatively, variable incrementation may be requested. In this case the current load factor will be automatically varied according to the iterative performance of the solution. The variation is a function of the required number of iterations and a specified desired iterative performance. Thus, where the number of iterations taken is less than the desired value the incremented load factor will subsequently be increased, and conversely, if the number of iterations is greater than the desired value, it will be decreased. Variable incrementation may be used in conjunction with either constant load level or arc-length solution methods and is an effective way of automatically adapting the performance of the solution procedure to the degree of nonlinearity encountered. The overall effect is therefore to increase and decrease the numerical effort in the areas of most and least nonlinearity respectively.

Mixing Manual And Automatic Incrementation

If required, manual and automatic incrementation procedures may be mixed freely. When mixing manual and automatic incrementation the following rules apply:

- Loadcases may be respecified as often as required.

- If the automatic procedure is specified, it will continue until one of the termination criteria is satisfied.

- In switching from manual to automatic control, any loading input under the manual control is remembered and held constant, while the automatic procedure is operating.

- In switching from automatic back to manual control, any loading accumulated under automatic control is forgotten and must be input as a manual loadcase if required.

- If prescribed displacements are being used, then in any switching from one type of control to another, the effect of prescribed displacements will be remembered and will not need to be input again.

Automatic Increment Reduction

Where an increment has failed to converge within the specified maximum number of iterations it will be automatically **reduced and re-applied**. This will be repeated according to values specified in the step reduction section (Advanced Nonlinear Incrementation Parameters dialog) until the maximum number of reductions has been tried. In a final attempt to achieve a solution the load increment is then increased to try and step over a difficult point in the analysis. If after this the solution has still failed to converge the solution terminated.

Solution Termination

When using manual incrementation, the solution will automatically terminate following execution of one increment. With automatic incrementation, the solution progresses one Nonlinear Control chapter at a time. The finish of each Nonlinear Control chapter is controlled by its Termination parameters.

Termination may be specified in 3 ways:

❑ **Limiting the maximum applied load factor.**
❑ **Limiting the maximum number of applied increments.**
❑ **Limiting the maximum value of a named freedom.**

Where more than one criteria is specified, termination will occur on the first criteria to be satisfied.

Failure to converge within the specified maximum number of iterations will either result in a diagnostic message and termination of the solution or, if automatic incrementation is being used, a reduction of the applied load increment. If required, the solution may be continued from an unconverged increment (Option 16, 17), although the consequences of such an action should be appreciated.

In addition, the solution will be terminated if, at the beginning of an increment, more than two **negative pivots** are encountered during the frontal elimination phase.

Use of Load Curves

Load curves are used to simplify the input of load data in situations where the variation of load is known with respect to a certain parameter. An example of this could be the dynamic response of a pipe to an increase of pressure over a given period. The load curve would consist of the definition of the load and its variation with time.

Nonlinear Solution Convergence Criteria

The convergence criteria specifies to what extent the numerical iterative procedure has reached the true equilibrium state. The specification of convergence therefore involves two considerations:

- Type of convergence criterion.

- Convergence tolerance.

The types of convergence criteria incorporated in LUSAS are as follows:

- ❑ **Absolute residual norm**
- ❑ **Root mean square residual norm**
- ❑ **Displacement norm**
- ❑ **Residual force norm**
- ❑ **External work norm**
- ❑ **Incremental displacement norm**

The convergence tolerance for each criteria is specified in the Solution parameters and advanced solution parameters section of the Nonlinear Control properties. The selection of a convergence criteria, and the associated tolerance, is problem dependent. However, the following points should be considered:

Clearly, the convergence criteria must not be too slack so as to yield an inaccurate solution, nor too tight so as to waste computer time performing unnecessary iterations. In general, sensitive geometrically nonlinear problems require a tight convergence criteria, whereas with predominantly materially nonlinear problems, larger local residuals may be tolerated.

Where more than one criteria has been specified, convergence will be assumed only on the satisfaction of **all** specified tolerances.

The following considerations apply to individual convergence parameters:

- ❑ **Absolute Residual Norm** is of limited use owing to its dependence upon the units being used. It is a strict criteria and for some problems, especially those involving plasticity, it may be very difficult to reduce locally large residuals. However, in sensitive geometrically nonlinear problems near bifurcation points, it can sometimes be necessary to ensure that large **residuals** are completely eliminated.
- ❑ **Root Mean Square Residual Norm** is the square root of the average of the squares of the residual forces and is generally more applicable than the above, but is still dependent upon the units being used.
- ❑ **Displacement Norm** is the sum of the squares of all the iterative displacements as a percentage of the sum of the squares of the total displacements and is a useful measure of how much the structure has moved during an iteration. Being a scaled norm it is not affected by units but convergence is not guaranteed. Typical values of slack and tight norms are (5.0 - 1.0) and (0.1 - 0.001) respectively.

- ❑ **Residual Force Norm** is the sum of the squares of all the residual forces as a percentage of the sum of the squares of all the external forces. This is the most versatile of the five criteria. Typical slack and tight values are (10.0 - 5.0) and (0.1 - 0.00001) respectively.
- ❑ **External Work Norm** is the work done by all the residuals acting through the iterative displacements, as a percentage of the work done by the loads on iteration zero of the increment. Since all freedoms are considered it is very versatile (the default displacement and force norms consider only the translational freedoms). However, it should be noted that a minimum detected potential energy need not necessarily coincide with the equilibrate state. Typical values of slack and tight norms are (0.1 - 0.001) and (10-E6 - 10-E9) respectively.
- ❑ **Incremental Displacement Norm** is the sum of the squares of all the iterative displacements as a percentage of the sum of the squares of the total displacements for the increment. This norm is an incremental form of the total **displacement norm** previously described and the same comments regarding usage apply.

Nonlinear Output Control

Nonlinear analyses may generate a vast amount of output. In addition to the normal nodal and element output controls, the frequency of nonlinear solution output may be restricted via the Output section in the Nonlinear Control properties.

The restart output facility enables failed or terminated analyses to be restarted from the last saved restart output dump. This is particularly useful where the termination of the analysis was due to a failure of the solution process rather than that of the structure. In this way, the solution may be restarted from the last converged increment with a different or modified solution strategy. For example, a failed increment may be restarted under either constant load or arc-length control. Restarts are not supported by LUSAS Modeller and hence must be defined directly in a LUSAS Solver data file.

The Nonlinear Logfile

During the course of a nonlinear analysis, various information is output to the screen or logfile, so that you may assess the performance of the solution. For more information refer to the *Solver Reference Manual.*

Creep/Viscoelastic Analysis

Nonlinear viscous behaviour occurs when the relationship between stress and strain is time dependent. The viscous response is usually a function of the material properties, stress, strain and temperature history. Unlike time independent plasticity where a limited set of yield criteria may be applied to many materials, the viscous response differs greatly for different materials.

A creep/viscoelastic analysis may be carried out using a linear or nonlinear material model within a nonlinear, transient dynamic or thermo-mechanically coupled analysis. When carried out in a nonlinear analysis inertia effects are neglected and the time component is introduced

using viscous control. 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.

Eigenvalue Analysis

An **Eigenvalue extraction** analysis is the extraction of the natural modes of vibration of a structure, or a natural frequency analysis. It can also be used to solve the following problems:

❑ **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 dynamic or spectral (harmonic/IMD) analysis if desired.

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 with Shifts** The inverse iteration method allows the computation of the eigenvalues and corresponding eigenvectors within a specified eigenvalue 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. Experience suggests that the number of starting vectors should be determined from the expression:

```
nivc = min (2*nroot, nroot+8, nvbz )
```

where

nivc is the number of starting iteration vectors.

nroot is the required number of eigenvalues.

nvbz the number of degrees of freedom in the structure.

Occasionally insufficient eigensolutions are computed in the initial eigenvalue analysis. The number of eigensolutions can be increased by using a restart and re-specifying the Eigenvalue control. This second eigenvalue analysis utilises the previous results in order to compute the extra eigensolutions thus saving on computational effort. Note that this option is not available with the Fast Lanczos solver, although repeated eigenvalue analyses may be performed during a run by re-specifying the eigenvalue control.

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.

Starting Vectors for Subspace Iteration

The first step of the subspace iteration method is the computation of the starting iteration vectors. Two methods of constructing these starting vectors are available in LUSAS. One method is based on the observation that the vectors should be constructed to excite the degrees of freedom associated with a large mass and a small stiffness. Alternatively, the starting iteration vectors can be obtained by using the solution from a Guyan reduction analysis. This method allows you greater freedom in selecting the starting iteration vectors. The starting vectors are defined by specifying master freedoms within the **retained freedoms**

(as for a Guyan reduction analysis). Then the eigenvalue control properties are used to control the eigenvalue extraction but a Guyan reduction analysis is carried out automatically prior to the subspace iteration algorithm which uses the approximate eigensolution from the Guyan reduction as the first estimate of the exact solution.

By making the correct assumptions and approximations it can be shown that a Guyan reduction analysis produces the same results as the first iteration of the subspace method with the starting iteration vectors as constructed by the first method; details of this can be found in the *Theory Manual*.

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.

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

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 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.

Note. 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 a 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. The maximum number of Lanczos steps to be taken is set to 100 by default, but can be altered, and should always be greater than the number of modes requested.

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.

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.

Notes

- The relationship between the eigenvalue, λ, and the angular frequency, ω, is:

$$\lambda = \omega^2 \quad \lambda = (2\pi f)^2 \quad f = \frac{\sqrt{\lambda}}{2\pi}$$

- An eigenvalue analysis of the stiffness matrix has no physical meaning except that a zero magnitude implies a critical point of some description.

- 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..

- Ensure that mass normalisation is chosen for the eigenvalue analysis if it is to be followed by a spectral or harmonic 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 magnitude of the eigenvalue shift required for an unsupported structure is usually taken as the expected fundamental eigenvalue.

- 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 the original buckling problem is recast to a form where all eigenvalues are positive, 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 (unless a smaller number of solutions has been specifyed).

Eigenvalue Buckling Analysis

A linear buckling analysis is a useful technique that can be applied to relatively stiff structures to estimate the maximum load that can be supported prior to structural instability or collapse. The assumptions used in linear buckling analysis are that the stiffness matrix does not change prior to buckling, and that the stress stiffness matrix is simply a multiple of its initial value. Accordingly, the technique can only be used to predict the load level at which a structure becomes unstable if the pre-buckling displacements and their effects are negligible. As this procedure involves assembly of the stress stiffness matrix, only elements with a geometric nonlinear capability can be used in a linear buckling analysis.

The main objective of an eigenvalue buckling analysis is to obtain the critical buckling load, which is achieved by solving the associated eigenvalue problem.

For buckling analyses involving constraint equations, the Fast Lanczos solver will only find eigenvalues either side of zero, i.e. in the range $(-\infty, 0)$ or $(0, \infty)$. If a range of eigenvalues is required in an interval which contains zero, two separate analyses must be carried out, where the interval is divided into two sub-intervals either side of zero.

Alternative Eigenvalue Buckling

Occasionally the initial stress stiffness matrix may not be positive-definite, causing the eigensolution method to fail. To overcome this problem the original buckling problem may be recast into a form where all eigenvalues are positive except when the buckling load factor is less than unity. When using this technique the load level must be adjusted to ensure that all the load factors are greater than unity. In other words, the load applied should be below the lowest expected buckling mode of the structure. An accurate load factor will however only be obtained if the specified load is close to the collapse load.

It should be noted that this procedure is not without its problems. Depending on the structure and the load level considered the eigenvalues can be very closely spaced, causing convergence problems in the iterative solution.

Output From Buckling Analyses

For a linear eigenvalue buckling analysis, the buckling load is obtained from the print results wizard. This buckling load is directly related to the eigenvalues extracted and will be in the following format

MODE	EIGENVALUE	LOAD FACTOR	ERROR NORM
1	33.0456	33.0456	0.190087E-10
2	64.3432	64.3432	0.179595E-07
3	130.903	130.903	0.202128E-11

The buckling load for a mode is obtained by multiplying the actual magnitude of the applied loading by the load factor (33.0456 in the case of the 1st mode).

Absolute displacement output is not available from any eigenvalue analysis. It is available, however in a normalised state. For buckling analyses the eigenvectors (mode shapes) are normalised to unity, where the maximum translational degree of freedom is set to one (mass normalisation is not applicable to buckling analyses).The mode shapes are, therefore, accurate representations of the buckling deformation but do not quantitatively define the displacements of the structure at the buckling load.

Reactions, stresses and strains represent the distribution at the buckling load, again their magnitude is not quantitative.

Spectral Response Analysis

To study the effects of ground motion excitation on structures it is necessary to input the intensity of the motion. One practical measure can be obtained from a knowledge of the response spectra. Spectral response analysis seeks to determine the response of a structure subjected to a specified support excitation using modal superposition. This can be achieved without recourse to direct integration of the model over the complete duration of an event.

A spectral response analysis is available using the **IMD** loadcase.

Starting Procedure

Before specifying the spectral response data the eigenvalues and eigenvectors of the system are computed using an eigenvalue extraction analysis (note that the computed eigenvectors must have been normalised to the global mass).

Spectral Response Data Input

The spectral curve, spectral curve type and percentage damping are specified in the spectral curve which is part of the IMD loadcase definition.

The spectral curve may be defined as:

❑ **Frequency or period vs displacement**
❑ **Frequency or period vs velocity**
❑ **Frequency or period vs acceleration**

To compute the participation factors it is necessary to specify the direction of excitation. The excitation may be specified simultaneously in three directions. The factor specified in each direction is used to scale the spectrum intensity in that direction.

For each mode the spectral displacement is determined from the frequency and this is multiplied by the participation factor and the excitation vector to determine the response.

Damping may be specified for each mode of the structure or at known frequencies of vibration. If damping is only specified for the first Eigen mode this value is applied to all modes. When the percentage damping specified on the spectral curve differs from that specified in the viscous damping a correction is made to the spectral displacement based on the formula chosen when defining the spectral combination. Damping may also be described in terms of the Rayleigh damping parameters and transferred from LUSAS Solver.

Normally the number of modes included should ensure the sum of the mass participation is not less than 90% in all significant excitation directions.

To obtain the design values some form of combination may be used. Within LUSAS the following methods of combination are available:

❑ **Square root of the sum of the squares (SRSS)**
❑ **Complete quadratic combination (CQC)**
❑ **Absolute Sum**

Note: When zero damping is specified the CQC gives exactly the same results as the SRSS technique.

Transient Dynamic Analysis

Where loading may not reasonably be considered to be instantaneous, or where **inertia** or **damping** forces are to be considered, a transient dynamic analysis (sometimes referred to as **step-by-step**) may be carried out. A dynamic analysis is controlled using the nonlinear and transient loadcase control properties.

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:

$$\mathbf{d}_{t,} \, V_{t-1/2,} \, A_{t-1}$$

This means that in the output for any time step, the displacements will relate to the current response time while the accelerations effectively lag one time step behind the displacements.

Impact Dynamics

In addition to using nonlinear joint models to represent contact and impact, a specialised procedure is available for modelling impact in dynamic analysis. This procedure uses a **slideline** technique, and permits the surfaces of 2D, axisymmetric, and 3D structures to register and react to contact with one another.
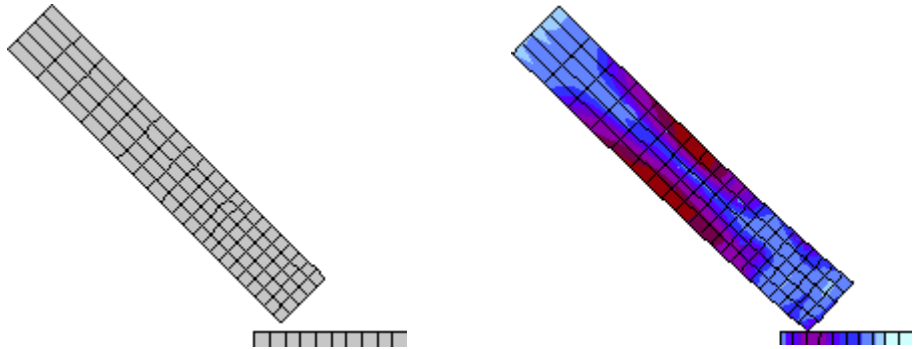
Coupled Analysis

The flow of heat through a body and the corresponding distribution of temperature is described by the quasi-harmonic equation; the body geometry is assumed to remain constant. The displacements of the same body, subjected to various forces, is described by equations of static or dynamic equilibrium; the temperature distribution is assumed not to vary with displacement. To include the effect of the change in geometry in the thermal analysis, and the change of temperature in the static analysis, requires that this information is separately calculated by the appropriate analysis and then transferred. This process is known as **thermo-mechanical coupling**.

Thermo-mechanical coupling may be sub-divided into two classes depending on the nature of the problem.

❑ **Semi-*c*oupled** analysis

❑ **Fully coupled** analysis

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.

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.

Field Analysis

Where a solution is required to the quasi-harmonic equation a field analysis may be performed. 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 variable, are listed in the table shown right.

Application	Field Variable
Thermal conduction	Temperature
Seepage flow	Hydraulic head
Incompressible flow	Stream function
Soap film	Deflection
Elastic torsion	Warping function
Elastic torsion	Stress function
Electric conduction	Electric potential
Electrostatics	Electric potential
Magnetostatics	Magnetic potential

Two types of field analysis may be performed:

❑ **Steady State** field analysis
❑ **Transient** field analysis

Facilities for **thermo-mechanically coupled** analysis are also available.

The solution of this class of problem follows an identical process to that of the structural problem. The domain is discretised using a series of field elements, **thermal material properties** are specified, **thermal loads** are applied, and the equations solved for the values of the field variable at each nodal point. Thermal link elements or the specification of **thermal surfaces** determine how heat is conducted, convected or radiated across gaps and

---

spaces between different domains. Since the most common application is that of thermal conductivity, subsequent discussion will be directed towards this type of analysis.

Steady State Thermal Analysis

In a manner similar to static structural analysis, steady state field analysis assumes that the loaded body instantaneously develops an internal field variable distribution so as to equilibrate the applied loads. It should be noted that the use of temperature dependent material properties or loads renders the problem nonlinear.

Transient Thermal Analysis

Where time effects are significant in a field problem a transient field analysis should be performed. 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.

Transient field analyses are controlled using the nonlinear and transient control. The initial conditions of the body must firstly be prescribed. This may be done by performing the appropriate linear or nonlinear steady state analysis.

Linear transient analysis

Integration Scheme	beta
Crank-Nicholson	1/2
Euler	0
Galerkin	2/3 (default)
Backward difference	1

The transient problem is integrated through time using a 2-point integration scheme. The type of integration scheme maybe changed on Advanced Time step parameters dialog accessed from the nonlinear and transient control dialog by specifying the parameter `beta`. Some of the more common 2-point integration schemes and their associated `beta` values are shown in the table above.

Notes

- In the limit the final solution should be the same as the steady state analysis subject to the new loading and boundary conditions; the transient analysis merely models the thermal inertia in moving from the initial to the final conditions. The body property which is used to describe this inertia is the effective heat capacity.

- When choosing an increment of time, the stability of the incrementation scheme must be examined. When `beta` is greater than or equal to 0.5 the solution is

unconditionally stable (the Crank-Nicholson, Galerkin and Backward difference schemes are of this form).

- When `beta` is between the limits 0 and 0.5 the solution is stable provided that:

$$\Delta t < \frac{2}{(1 - 2\beta)\lambda_{max}}$$

where β is the input parameter `beta` and λmax is the maximum eigenvalue of the system.

- The time step used for implicit algorithms is dependent upon the number of modes that influence the response of the system. Generally, the major part of the response is governed by the lower modes so that:

$$\Delta t < \frac{1}{3\lambda}$$

where λ is the minimum eigenvalue of the system.

- The Galerkin scheme is recommended since it generally provides good accuracy and is the least susceptible to oscillations.

Nonlinear Transient Analysis

For nonlinear transient analysis the backwards difference algorithm must be used (`beta` = 1.0). The backward difference algorithm is unconditionally stable, and the time step length considerations are the same as for linear analyses.

For analyses including a phase change, there is either an absorption or release of energy in order to create or break the molecular bonds. This is modelled by varying the effective heat capacity in the transient analysis. To do this the material property of enthalpy is introduced.

Enthalpy, H, is defined as:

$$\frac{dH}{dt} = \frac{dH}{d\varphi}\frac{d\varphi}{dt} = C\frac{d\varphi}{dt}$$

where C is the effective heat capacity including the effects of the latent heat of evolution due to phase changes and φ is the temperature. In the material data input both H and C may be specified. For analyses where phase changes are not represented, the effective heat capacity value C is used in the calculations.

For analyses where phase changes are represented, tabular input should be used to define the variation of H with temperature, together with an initial value of C. Providing a variation in temperature exists at a point, the effective specific heat is then interpolated from the enthalpy values. If no variation exists, for example, in an area of the problem that has experienced no change in temperature from the initial temperature, then the initial value of C is used.

For nonlinear analysis the nonlinear control parameters are used to define the iterative strategy. The convergence section is utilised to provide tolerances for defining steady state, and either a field norm (temperature equivalent to the displacement norm), or a residual flow norm (equivalent to the residual force norm) may be used.

Fourier Analysis

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.

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, in the table shown, 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.

Load curve definition

equivalent

Turbine
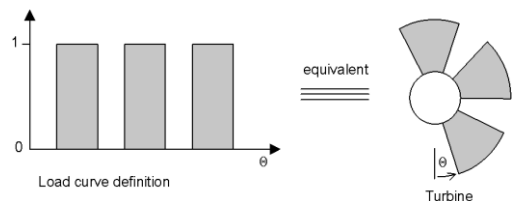
Thermal Problems

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**.

Frontal Optimisation and LUSAS Solvers

The frontal optimiser is set from the **Model Properties> Solution - Optimiser options** tab and is only invoked when the standard frontal solver is chosen.

Selecting the Frontal Optimiser

Front optimisation is only required when using 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.

Choosing the Solver

Direct (e.g. frontal) solvers are more robust than iterative solvers and are applicable to all types of analysis. Direct solvers should always be used for very **ill-conditioned** problems since the time taken to obtain a solution is independent of the problem conditioning. Iterative (e.g. conjugate gradient) solvers are usually only applicable to static, linear analyses, and will perform best on large, well conditioned problems, since the time taken for solution is less dependent on the size of the problem than for direct solvers. Iterative solvers require far less storage (memory + disk space) than direct solvers. This means that iterative solvers sometimes have the advantage of remaining in memory where a direct solver would have to run "out of memory". Iterative solvers are only applicable for a single loadcase.

Support for specification of various solver types is available within LUSAS Modeller. The current solver options supported are as follows:

❑ **Direct (Frontal)** selects 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. There is a choice of two direct solvers:

- **Standard Frontal**- an element-by-element frontal solver which does not require assembly of the global stiffness matrix. This solver is applicable to all types of analysis present in LUSAS Solver.

- **Fast Multi-Frontal**- a global frontal solver which assembles global stiffness and load data. This solver is applicable to all analysis types except for superelements, Guyan reduction and non-linear problems involving branching and bracketing.

❑ **Iterative (Conjugate Gradient)** selects an iterative, sparse solution technique for solving static, linear analyses. The global stiffness matrix and load vector(s) are assembled, and is designed to run entirely in-memory. Three preconditioning techniques are available to assist the convergence rate of the conjugate gradient method:

- **Standard -** The incomplete Cholesky preconditioning technique is the most robust (provided an appropriate drop tolerance is chosen), and is applicable to all analyses for which the conjugate gradient solver may be used.

- **Decoupled -** The decoupled incomplete Cholesky preconditioning technique may be used for all analyses except those involving tied slidelines, thermal surfaces and Fourier elements. It generally leads to faster overall solution times than Incomplete Cholesky preconditioning, although more iterations are required for convergence. For less well conditioned problems, the conjugate gradient algorithm may not converge using this technique, so care should be taken.

- **Hierarchical** - The hierarchical decoupled incomplete Cholesky preconditioning technique is only available for models consisting entirely of two- and three-dimensional, solid continuum, quadratic elements, and offers excellent convergence properties. It is by far the most effective technique for models of this type, and when used in conjunction with fine integration allows solutions to be obtained for relatively ill-conditioned problems. For very ill-conditioned problems of this type (e.g. where the average element **aspect ratio** is high), an extra preconditioning option exists which will often yield a solution faster than using a direct solver.

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. Also, the convergence of the iterative solver is

related to the condition number of the stiffness matrix which becomes worse for ill-conditioned problems.

The conjugate gradient iterative solver can be configured using the following parameters:

- **Drop tolerance**- a value between 0.0 and 1.0 which measures the amount of new non-zero entries (known as *fill-in*) allowed to remain in the preconditioning matrix during the incomplete Cholesky factorisation of the stiffness matrix. The default value is 1.0, leading to a very sparse preconditioning matrix suitable for well conditioned problems. For more ill-conditioned problems, however, this value should be decreased exponentially, and values in the range [1.0e-3, 1.0e-6] are recommended. The lower this parameter becomes, the larger the preconditioning matrix will be, giving rise to fewer iterations during the conjugate gradient solution, although each iteration will take longer to process. This parameter affects all preconditioning techniques, although the effect is less pronounced for the decoupled techniques.

- **Maximum number of iterations**- an upper limit on the number of conjugate gradient iterations to be processed; the default value is 5000. If the convergence criterion has not been satisfied when the iteration limit is reached, LUSAS Solver will issue a warning and then continue the analysis.

For the iterative solver, 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*

Support with Modelling and Analysis Problems

The engineers in LUSAS Technical Support are available to help all clients with a current support and maintenance contract and assist with any problems that may be encountered. However, to identify the cause of a problem will take time, especially if the analysis is large. To reduce the time taken in diagnosing input errors the analysis model should be thoroughly checked. It is good practice to systematically carry out checks as a matter of course whether or not there appears to be a problem with the solution obtained.

With the aim of producing a more efficient service, some general pointers are given as to what information should be made available when calling the support desk. This 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 or data file causing problems.

- 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 complex or difficult to describe problems, email or fax a simple diagrammatic representation before calling to aid any discussion.

Try to be logged onto the machine when calling or be close enough to the machine to try a suggestion provided by the support engineer.

For an explanation of the errors which may occur during the Solution see **Appendix B - LUSAS Solver Trouble Shooting**

Sending files to LUSAS Technical Support

Use the **LUSAS Support Tool** to create a compressed file of all relevant data to send to LUSAS Techincal Support

Pre-Analysis Checks

1. Check the consistency of your co-ordinate systems between the finite element model and any engineering drawing that you have worked from.

2. Check key drawing dimensions against co-ordinates of respective points in the model.

3. **Check the mesh for cracks and voids**. Checks for cracks must be made to ensure that the features form a continuous structure.

4. **Check for correct material properties and assignments**.

5. Check for consistent units.

6. **Check for correct orientation of beam properties.**

7. **Check for correct boundary conditions (loads/supports).**

8. **Check element thickness against original model data (plates/shells).**

9. **Check reversed normals for plates/shells/2-D.**

10. Check element shapes for **aspect ratio**, skew, **warp**, taper, **curvature** and **centrality of mid-side nodes**. Warning messages will be present in the output file for all of the above.

11. From the LUSAS datafile dialog, **File> LUSAS 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.

- When carrying out nonlinear or transient analysis it is always best to run a linear analysis first. No time is really wasted as the model can easily be subsequently converted for the nonlinear or transient analysis afterwards.

- It is advisable to add in nonlinear behaviour in stages. For example in a material and geometrically nonlinear run containing slidelines it would probably be advisable to start with only slidelines, then add the geometric nonlinearity and finally add the nonlinear material effects. In this way you can ensure that each nonlinear procedure is stable before progressing to the next.

Running an Analysis

To create a LUSAS datafile and run an analysis either click on the Solve button , or use the **File> LUSAS Datafile** menu item.

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.**

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.

Chapter 8 : Viewing the Results

Introduction

This section deals with procedures for results processing. It covers manipulation of results files, selection of the correct results type and loadcase, and differences between results viewing coordinate systems.

- ❑ **Results Processing** provides an overview of results processing.
- ❑ **Results Files** covers manipulation of results files.
- ❑ **Results Selection** covers selection of the active loadcase, a fibre location and a composite layer. It outlines all the different results types available during post-processing.
- ❑ **Results Transformation** presents the options for transforming results.

Results Processing

Results processing, also known as post-processing, is the manipulation and visualisation of the results produced from an analysis. Prior to visualising and extracting results, further calculations may be carried out to create or assemble results, or the results model can be manipulated to create results at particular model locations for a particular results viewing use.

Depending on the type of analysis any of the following results calculation, manipulation or viewing can be carried out:

Further Calculations

- ❏ **Combinations and envelopes**
- ❏ **Wood and Armer reinforcement**
- ❏ **Fatigue analysis of a linear model**
- ❏ **Design factors**
- ❏ **Composite failure criteria**
- ❏ **Modal analysis**
- ❏ **Beam stress resultant from beam and shells**
- ❏ **Expansion of Fourier analysis results**
- ❏ **User defined results**

Manipulating the Results Model

- ❏ **Plotting results for groups**
- ❏ **Results on a section through the model**
- ❏ **Slideline results processing**
- ❏ **Thermal surface results processing**

Visualising and Extracting Results

- ❏ **Deformed mesh**
- ❏ **Contour plots**
- ❏ **Results as vectors**
- ❏ **Bending moment and force diagrams**
- ❏ **Concrete crack patterns**
- ❏ **Yield of nonlinear material**
- ❏ **Peak values**
- ❏ **Plotting beam stresses**
- ❏ **Plotting results for assigned attributes**
- ❏ **Graphing of results**
- ❏ **Animation sequences**
- ❏ **Printing results to the screen or a file**
- ❏ **Generating a Report**

Analysis Results

Further calculations
Combinations/Enveloping
Wood and Armer reinforcement
Fatigue analysis of a linear model
Design factors
Composite failure criteria
Modal analysis
Beam stress resultant from beam and shells
Expansion of Fourier analysis results
User defined results

Manipulating the model
Plotting results for groups
Results on a section through the model
Slideline results processing
Thermal surface results processing

Visualising results
Deformed mesh
Contour plots
Results as vectors
Bending moment and force diagrams
Concrete crack patterns
Yield of nonlinear material
Peak values
Plotting beam stresses
Plotting results for assigned attributes
Graphing of results
Animation sequences
Printing results to the screen or a file
Generating a Report

Results Files

When an analysis is performed by LUSAS Solver a results file will be created. For historical reasons this has a **.mys** extension. By default the results file is automatically loaded into LUSAS Modeller on top of the model file after **LUSAS Solver** has been run.

The information in the 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 Results Files

Results may be loaded on top of a model or used stand-alone. Results are usually loaded automatically when analysis is carried out and are offered for opening if a results file of the same name as a model file is detected when a model is opened. Results files may also be opened in isolation manually using the **File> Open** menu item.

❑ **Load results on top of current model**

Results are normally processed by reading the results file on top of an existing model file. This enables the visibility of the model to be controlled by the assigned attributes and all model data including group information is present to aid results manipulation. Supports and loading attributes as assigned for each loadcase can be seen. When results are loaded on top of a model file only the results are loaded from the .mys file. Multiple results files may be loaded when accessing results from a number of analyses at the same time. Subsequent results files may be loaded on top of an existing model or an existing results file.
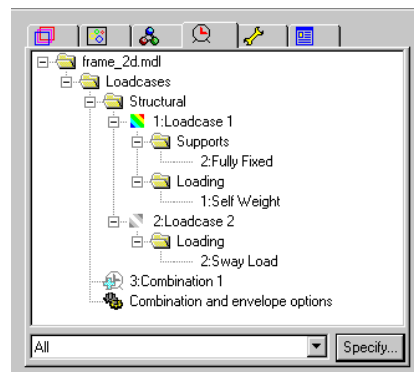
The 🕔 Treeview above shows a results file loaded on top of a model file.

❑ **Results Only**

When results files are opened stand-alone the mesh definition and all of the results are read 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. Subsequent results may be loaded on top of the first results file loaded.

The ⏰ Treeview above shows the listing obtained for the same analysis when only the result file is loaded .
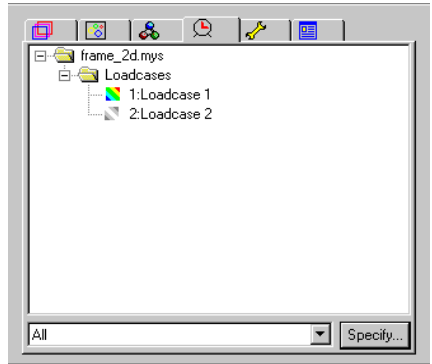
Results Selection

Setting the Active Loadcase

The active loadcase is the loadcase that provides the results for the current window. In this way a single window is used to plot results from a single loadcase, and multiple windows can be used to compare results from different loadcases.
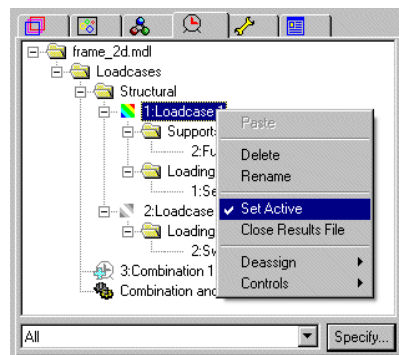
Setting the Active Fibre Location

For plotting diagram results on bars and beams <u>only</u>, the active fibre location for which results will be plotted must be chosen. Fibre locations available for these elements can be seen and can be set active in the ⚙ Treeview. Fibre locations can visualised by double-clicking on the Geometric Line name and selecting the Visualise button on the dialog presented. Setting the Fibre Active shows results for just that fibre. Setting the Name Active shows results for all similarly named fibres throughout the model.
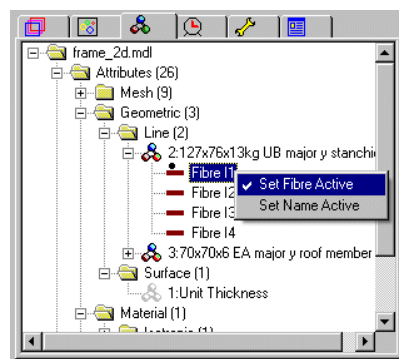
Setting the Active Composite Layer

With composites analysis, in order to plot results on composites laminae a particular lamina must be made active. Setting a Lamina active shows results for just that lamina. Composites layer information can be seen and visualised by double-clicking on the Laminate stack name and selecting the Visualise button on the dialog presented.
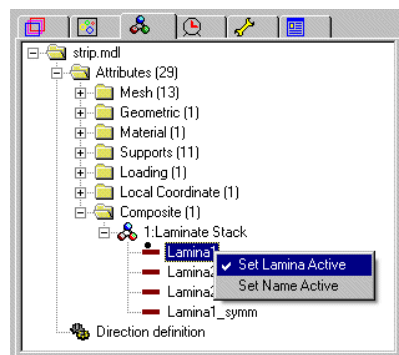
Results Types

There are several different types of results entities available in each results file. Full details of the results available for each element type may be found in the *Element Reference Manual*.

Each time results are displayed the results entity must be specified. Only result entities that are actually contained in the loaded results file are presented for selection. The following is a summary of all the results types:

Structural Analysis		Thermal Analysis
❑ Displacement	❑ Plastic Strain	❑ Potentials
❑ Stress	❑ Creep Strain	❑ Fluxes
❑ Strain	❑ Rubber Stretch	❑ Gradients
❑ Loading	❑ Strain energy	❑ Thermal Surface
❑ Reaction	❑ Plastic work	
❑ Reaction Stress	❑ Slideline	
❑ Residual	❑ Named variables	
❑ Velocity	❑ State variables	
❑ Acceleration		

Notes

- Stresses are stress resultants for beams, plates and shells (i.e. forces for beams and force/unit width for plates and shells).

- For plates and shells top middle and bottom stress and strain are available.

- When applicable, Wood Armer results, composite failure values, design factors and beam stresses are available from the Stress results type.

- The results calculation and display may be controlled independently using the Result Plots dialog activated from the top level of the Groups context menu. See **Groups** for details.

Results Transformation

By default, displacements, loads, reactions, residuals, velocities and accelerations are output relative to the global Cartesian axis. For beams, joints and shells, stress and strain output is relative to element local axes. For all other elements stress/strain and creep/plastic strain results are output relative to the global system.

Sometimes it is useful to transform the results to a consistent or alternative coordinate system. For example if the elements in a model are orientated such that their local axes vary from one another, the results may be transformed to a consistent direction.

Results can be transformed by editing the contour, values or vector layer properties or when using the print results wizard.

Local and Global Results

When specifying a results entity the results may be transformed relative to:

- ❑ **Local coordinate** Transforms results relative to a specified local coordinate. Only available if a local coordinate has been defined.
- ❑ **Local** Transforms results according to the local transformed freedoms for displacements, etc. or according to element local directions for stresses.
- ❑ **Material direction** Transforms results relative to the local element material directions.
- ❑ **Specified angle** (in the XY plane) Transforms results by defining a transformation angle in degrees about the global Z axis. The transformation angle is measured in a positive, anti-clockwise direction from the global X axis.

The validity of each of the above transformation methods for different results entities is given in the table below.

Results Entity	Transformation	Notes
Stresses/Strains, Creep/Plastic Strains, Gradients	Local coordinate	Note that results for beams and joints will always be relative to the element local direction.
	Local element direction	Gives results relative to the element local axes. A warning is issued to remind you to ensure a consistent set of directions are available for contouring.
	Material direction	Gives results relative to the element material directions. A warning is issued to remind you to ensure a consistent set of directions are available for contouring.
	Angle	Note that beam and joint results will always be relative to the element local direction.
Displacements, Loads, Reactions, Residuals, Velocities, Accelerations, Potentials	Local coordinate	Cartesian sets existing in the LUSAS Solver data file will be used to automatically create LUSAS Modeller local coordinate systems for use with this command.
	Angle	For 3D elements care must be taken to ensure the transformation is valid by limiting the shell/plate elements showing results to those parallel to the global XY plane.

Notes

- When viewing Wood Armer results this facility 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 may be defined as rt, tz or rz where r is the radial direction, t is the theta direction, and z is the cylindrical axis.

Combinations and Envelopes

Combinations and envelopes can be defined for manipulating results loadcases.

❑ **Combinations** enable results from different loadcases to be added together. For example, the results from a *self-weight* loadcase and a *concentrated load* loadcase might be added, or perhaps the results from a *dead load* loadcase and an IMD loadcase used for *spectral* analysis might be added.

❑ Combinations can be created from loadcases, or other combinations or envelopes.

❑ **Envelopes** are used to pick out the maximum and minimum values from a number of results loadcases, combinations or other envelopes. When an envelope has been defined, a pair of loadcases are created containing the maximum and minimum envelope values.
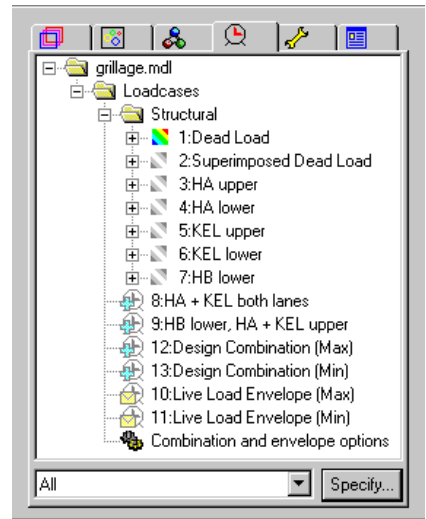
Defining a Combination or Envelope

The Utilities menu is used to add a new combination or envelope. Combinations or envelopes may be defined at either the modelling stage prior to analysis, or at the results processing stage. Once defined, combinations and envelopes are treated as loadcases and are listed in the Loadcase 🔍 Treeview. Their properties may be edited by double-clicking on the entry in the Treeview.

When the first load combination or envelope is added to the Treeview, a 🐞 *Combination and envelope options* object is also created. Double-clicking on this object displays a dialog on which results components can be selected for calculating and saving in the **Modeller results file**. When results components are selected prior to an analysis being carried out the results for the primary components chosen will be available for results processing immediately after the results file is loaded because results for these components will have been automatically saved (cached) in the Modeller results file. When results components are selected on the

Combination and envelope options dialog after an analysis has been carried out an option to calculate results for the components selected is provided. These results will also be cached in the Modeller results file to speed-up results viewing. If a model is saved these calculated results will also be saved.

Combinations and envelopes can consist of any other available loadcase results, including other combinations and envelopes. Combinations and envelopes may contain results from more than one **results file**, if there are other results files open.

Two types of combinations may be defined.

❑ **Basic combinations** enable a single factor to be applied to each set of results included in the combination properties.

❑ **Smart combinations** enable two factors to be applied to each set of results. The first factor is known as the permanent factor as it is always applied. The second factor is known as the variable factor as it is only applied if the load effects are adverse.

A maximum smart combination (Max) will assemble results from the loadcases selected using just the permanent factors for negative load effects, and using permanent and variable factors for positive load effects. A minimum smart combination (Min) will assemble results from the loadcases selected using just the permanent factors for positive load effects, and using permanent and variable factors for negative load effects.

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. Firstly, the maximum combination will include only positive load effects, all negative load effects will be discarded. Likewise for the minimum combination, which will include only negative load effects. Secondly, 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.

Different factors for permanent and variable effects may be specified for each combined results loadcase. The **Permanent load factor** is always applied while the **Variable load factor** is only applied if the effect is adverse. These options may be optionally displayed as **Beneficial load factor** and **Adverse load factor** respectively if the appropriate check box has been selected on the dialog displayed for the 🐛 *Combination and envelope options* object.

For further details see **Appendix A.**

Visualising The Results From Combinations And Envelopes

Once defined, combinations and envelopes may be manipulated in the same way as other **loadcases**.

LUSAS will display the results from the active loadcase. The active combination or envelope is selected from the 🔍 Treeview by selecting the appropriate envelope or combination with the mouse, clicking the right hand mouse button, and selecting the **Set Active** menu item from the context menu.

When using smart combinations or envelopes the **Set Active** menu item will prompt for the primary component on which to base the combination or envelope. For an envelope this component will be used to decide which is the maximum or minimum loadcase and for a smart combination which factor to apply to each loadcase. When displaying or printing results the values for other components will be the coincident effects. For envelopes, if no component is specified all components are enveloped independently.

An active loadcase is identified by a coloured icon in the 🔍 Treeview. Non-active loadcases are greyed-out. For envelopes and smart combinations either the maximum or minimum can be set active.

Notes

- Combinations are only applicable to linear elastic analyses.
- Envelopes and combinations may be saved in the model file or in a new model file when a results file only is opened.
- When a smart combination includes both envelope (Max) and envelope (Min), one of the envelopes will be ignored. Smart combination (Max) ignores envelope (Min) and smart combination (Min) ignores envelope (Max).
- 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 combining or enveloping results from multiple results files the mesh must be identical across results files.
- When envelopes of envelopes or combination results are calculated they are automatically cached. Saving the model will also save the cached results to the Modeller Results File so they are available for future use. Subsequent access to these results will be similar to accessing results from a single loadcase
- If the loadcase IDs which contain the most adverse effects are not required, enveloping can be significantly speeded up by placing an envelope within an envelope.

Tips

- Sometimes, due to hardware restrictions, it may be convenient to run an analysis which contains many loadcases in parts. The loadcases from the separate results files may then be subsequently enveloped or combined.
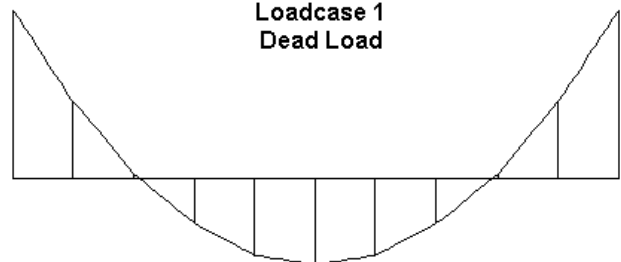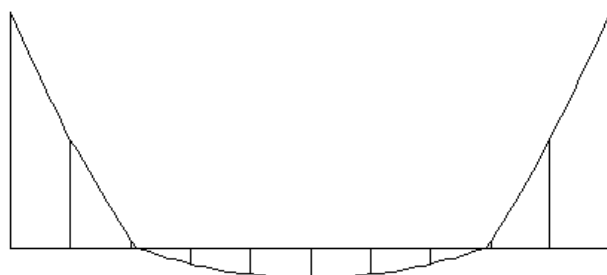
Basic Combination Example

LOAD CASE = 1
HA & KEL upper
RESULTS FILE = 1
STRESS
CONTOURS OF Mx

-71.0253
-60.8788
-50.7324
-40.5859
-30.4394
-20.2929
-10.1465
0
10.1465

Max 12.57 at Node 127
Min -78.75 at Node 196

1.0 x Loadcase 1

+

LOAD CASE = 2
HA & KEL lower
RESULTS FILE = 1
STRESS
CONTOURS OF Mx

-71.5926
-61.3651
-51.1376
-40.9101
-30.6826
-20.455
-10.2275
0
10.2275

Max 13.09 at Node 156
Min -78.96 at Node 190

1.0 x Loadcase 2

=

Combination 3
STRESS
CONTOURS OF Mx

-87.3605
-76.4405
-65.5204
-54.6003
-43.6803
-32.7602
-21.8401
-10.9201
0

Max 2.317 at Node 42
Min -95.96 at Node 190

Combination

Smart Combination Example

Loadcase 1
Dead Load

Loadcase 2
Live Load

Smart Combination =
Dead Load * 1.0 (Permanent Factor) & 0.0 (Variable Factor)
Live Load * 0.0 (Permanent Factor) & 1.0 (Variable Factor)

Smart Combination (Max)
Most Adverse Hogging
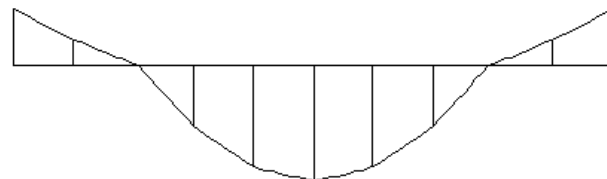
Smart Combination (Min)
Most Adverse Sagging

For details of smart combination calculations see **Appendix A**

Enveloping Example

Individual Loadcase Bending Moment Results
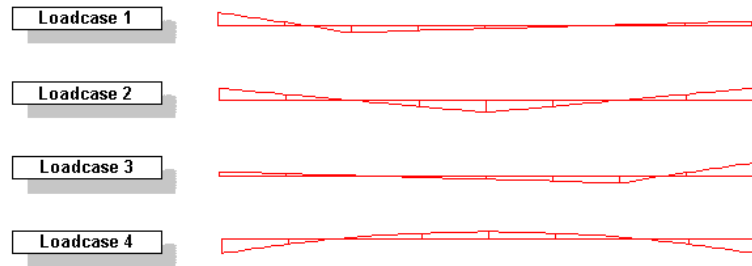
Maximum Enveloped Bending Moment Diagram

Minimum Enveloped Bending Moment Diagram

Wood Armer Reinforcement

The Wood Armer facility allows reinforced concrete slabs to be designed to resist a combination of moments [Mx, My] and a twisting moment [Mxy] using orthogonal (or skew) reinforcement. Following a linear elastic analysis the Wood Armer facility determines the design moments [Mx(T), My(T), Mx(B), My(B)]. The procedure was originally developed for a moment field obtained from a plate analysis but may also be applied to shell or grillage analyses.

For a slab modelled as a shell, subject to a moment field [Mx, My, Mxy] and a stress field [Nx, Ny, Nxy] the Wood Armer principles are extended (using the Clark Neilsen calculation). The final output [Nx(T), Nx(B), Ny(T), Ny(B), Fc(T) and Fc(B)] incorporates both bending and in-plane load effects. In order to obtain equivalent in-plane forces from the applied moments, it is necessary to establish the distances between the centroid of the reinforcement layers and the middle surface of the slab. This is calculated using the thickness of the shell elements entered in the geometric properties and the distance to the reinforcement centroid from the face of the slab, entered in the Wood Armer dialog.

An approximate approach is also available for grillage elements whereby bending and twisting moments are converted into 'equivalent' plate moments so that the Wood-Armer equations can then be used. An extra geometric property, 'effective width', has to be defined

for the grillage to compute these equivalent moments. The effective width is defined in the **grillage geometric attribute**.

Wood Armer properties are specified on the Wood Armer dialog which is activated when a Wood Armer component is selected from the contour, values or vector layer properties or from the print results wizard. The properties specified in the Wood Armer dialog are applicable to all layers.

Wood Armer Assessment

The Wood Armer calculation is generally based on a rationalised set of equations intended to enable efficient design. However, it is sometimes necessary to consider arrangements of reinforcement which have design strengths based on a different rationale. A typical application is the assessment of an existing structure.

If the capacity of the section in the direction of the reinforcement is known the 'utilisation factor' based on optimal use of the available capacity can be computed. A utilisation factor greater than 1 signifies the slab is under-reinforced and extra reinforcement is required.

Alternatively, the Wood Armer 'K factor' may be entered to proportion the applied twisting moment between the reinforcement directions. This allows any spare capacity that exists in either direction to be utilised.

Notes

- Grillage Wood-Armer results are only valid for small skew angles.

- When calculating Wood Armer results the x direction may be **transformed** to set an angle between the global axes and the local axes.

See the *Theory Manual* for further information.

Fatigue Calculations

Fatigue calculations can be performed on the results of a linear finite element stress analysis using the total life approach. This can be done for continuum elements only. The fatigue life may be expressed in terms of the damage that is done to the structure by a prescribed loading sequence or as the number of repeats of the sequence that will cause failure of the structure. Contour plots illustrating the fatigue life of the entire structure can be generated. The results from fatigue calculations may be viewed using any of the standard plotting techniques.

Fatigue calculations of the life of a structure are defined from the **Utilities> Fatigue** menu item.

S-N Curves

S-N curves contain the variation in stress/strain values with the number of cycles to failure and are defined on a Log Log scale. An S-N curve is used to calculate the number of cycles to failure for each loadcase. Miner's rule is then used to combine the damage for each loadcase to give the total damage to the structure for the specified loading sequence.
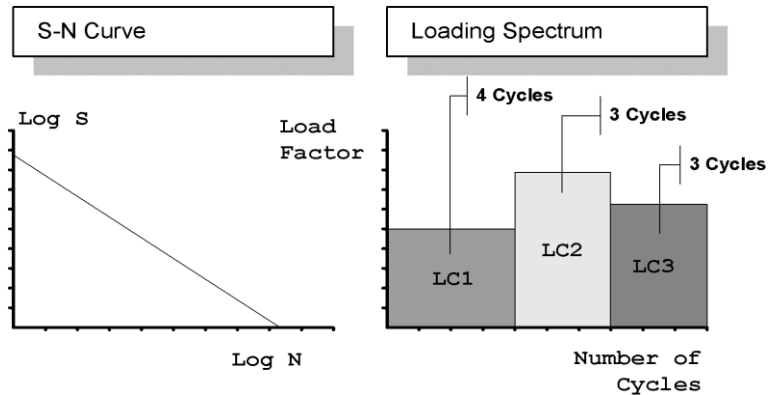
Notes

- Log values are used because of the large variation in magnitude of the input data.

- If plotting the number of cycles to failure, then the number of cycles entered in the fatigue spectrum should sum to unity.

- For information on standard S-N curves refer to BS5400.

Fatigue loadcase

Fatigue loadcases contain the loading spectrum defining the loading sequence in terms of a series of loadcases, each of which has an associated load factor, the number of cycles, and the component to be used in the fatigue calculation.

Once defined fatigue loadcases are saved in the loadcase Treeview 🕒. Their properties may be edited by double-clicking on the fatigue loadcase.
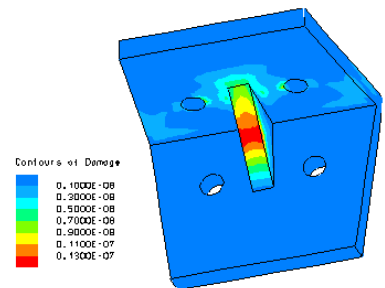
Fatigue Results

Results from the fatigue calculations may be viewed using any of the standard methods, once the loadcase is **set active**. Fatigue results come from a component of stress, which is specified in the Fatigue loadcase. Two results are obtained from the calculations:

❑ **Loglife** is the life expectancy of the structure based on the applied load. Results are presented as $\log_{10}$ of the number of cycles to failure.

❑ **Damage** is a factor representing the damage the material has sustained due to the applied loading and number of cycles. A value greater than 1 indicates failure.

Notes

- Fatigue calculations are only applicable to linear elastic analyses and for continuum elements only.

- Fatigue loadcases may be saved in the model file or in a new model file when a results file only is opened.

Fourier Results

Before the results for the Fourier can be established, it is necessary to define a combination in which the Fourier harmonics are combined to provide an overall result. The combination is created using **Utilities> Load Combination> Basic…** menu item.

Once the combination has been generated for all the harmonics required and the analysis run to create a results file, the combination loadcase should be set active.

There are then two ways to view results from a Fourier analysis:

❑ By manually specifying the angle around the circumference at which the results are required and using contours, values and vectors to view the results at that position in the model.

❑ By using the graph wizard to display the variation of results around the circumference for a specified node.

Design Factors

Design factors provide a means of assessing the reserve strength capacity of a component or structure. To use this facility the results file must be loaded on top of the model file. The material strength and design factor are defined from the **Attributes> Design Strength** menu item and assigned to the model. The reserve strength can then be visualised over the entire model using the **Utilities> Design Factors** menu item. The following criteria are available:

❑ **Maximum Stress Theory (Rankine)**

❑ **Maximum Shear Theory (Coulomb, Tresca)**

❑ **Maximum Strain Energy Theory (Beltrami)**

❑ **Maximum Distortion Energy Theory (Huber, von Mises, Hencky)**

❑ **Maximum Strain Theory (St Venant)**

Results from each of the above criteria can be displayed as:

Design Factor	Calculation	Failure
Failure/Yield Index	actual/allowable	>1
Factored Failure/Yield Index	(DF*actual)/allowable	>1
Factor of Safety	allowable/actual	<DF
Reserve Factor (RF)	allowable/(DF*actual)	<1
Margin of Safety	RF-1	<0

where:

> DF = Design factor defined with material strength
>
> allowable = allowable stress
>
> actual = actual stress

See the *Theory Manual* for further information.

Composite Layers

When viewing composite results it is often useful to change the results orientation to material directions to view the results in fibre and off fibre directions. See **Local and Global Results** for more details.

Note. A layer of a solid composite element is treated like a shell. Thus top, middle or bottom stresses can be obtained.

Composite Failure Criteria

Composite failure criteria is defined as an attribute from the **Composite> Composite Failure** menu item. It provides a means of assessing the reserve strength capacity of composite components without carrying out a full nonlinear analysis. To use this facility the results file must be loaded on top of the model file.

- ❏ Longitudinal and transverse tensile and compressive strengths must be defined along with a shear strength (which cannot be zero)
- ❏ Interaction type can be set to use **Default**, **User**, or **Cowin** values

Once created the Composite Failure attribute should be assigned to the model geometry so that the reserve strength can be visualised over the entire model. Once assigned to the model the following stress components are available for display via selections made on the Contours properties dialog:

- ❏ **Tsai Hill**
- ❏ **Hoffman**
- ❏ **Tsai Wu**
- ❏ **Hashin Fibre**

❑ **Hashin Matrix**

Results may be printed or displayed using standard results processing facilities. Note that a failure criteria greater than 1 indicates failure. See the *Theory Manual* for further information.

Composite Failure Contours

When using the Hashin failure material model the failure indicator (IFFLR) can be contoured. The indicator has the following values:

Indicator (IFFLR)	Description
0-1	No failure
1-2	Matrix failure
2-3	Fibre failure
3+	Matrix and fibre failure

Results may be printed or displayed using standard results processing facilities.

Interactive Modal Dynamics

The Interactive Modal Dynamics (IMD) facility within Modeller calculates the modal response of a system to a given input using the eigen modes and eigenvectors from an **eigen analysis**. (Note that the eigen analysis must have been performed with mass normalised eigen modes). An IMDPlus software option extends this capability to solve 2D and 3D seismic and moving load analyses using modal superposition techniques in the time domain. For details see the *IMDPlus User Manual*

Interactive Modal Dynamics calculations within Modeller may be performed on single or multiple eigen modes, and at a single node or over the whole structure:

❑ **Response at a node** Use the Graph Wizard to produce a graph of a specified results type against sample frequency range or time steps.

❑ **Response for all nodes** Use an IMD loadcase to calculate the modal response of the whole structure to a specific frequency or at a particular response time. The results from the IMD calculation are then viewed using any of the standard plotting techniques such as contouring.

Assumptions

The working assumptions for the modal dynamics facility are as follows:

❑ **Linear** The system is linear in terms of geometry, material properties and boundary conditions.

❑ **No Cross-Coupling** There is no cross-coupling of modes caused by damping. This is reasonable as long as the damping of the structure does not exceed 10% of critical damping.

❑ **Low Modes Dominant** The response is dominated by the lowest few modes.

Performing Modal Response Calculations

Both the **Graph Wizard** and **IMD Loadcase** commands are initiated from the **Utilities** menu. The basic steps for both methods are as follows:

1. Decide whether to perform a response analysis on a single node (Graph Wizard), or the whole structure (IMD loadcase).

2. **Select Eigen Modes** Specify which modes to include in the analysis.

3. **Damping** Specify the amount of modal damping.

4. **Excitation** Apply a form of dynamic excitation to a specific node or at the supports.

5. **Results Type** Choose the IMD calculation required, then set the required parameters.

Modal Damping

Modal damping is the damping associated with the displacements defined by the eigenvectors. Its value has no physical significance since the eigenvector contains an arbitrary normalising factor.

Damping values can be specified explicitly or alternatively can be extracted from the results file. LUSAS Solver will only provide modal damping estimates if the relevant damping control data has been included in the eigenvalue analysis. By default, LUSAS Solver values will be zero. Damping is specified using percentage viscous and structural damping values.

Notes

- Structural damping is not used in modal response calculations in the time domain.

- Structural damping is not applicable in spectral response calculations.

- All damping ratios are expressed as percentages of critical damping.

- Modal superposition techniques are not usually appropriate for structures with damping ratios higher than 10%, due to coupling between modes. Step-by-step dynamic analysis should be considered in such cases.

Dynamic Excitation

Eigenvalue analyses do not consider any applied load. Therefore, to carry out a dynamic structural analysis a load must be applied to the structure. These loads are specified either in terms of **forces**, or as motion by use of the **large mass method**:

❑ **Point Force** To set the modal excitation to point force via a node number and a nodal freedom. This is equivalent to applying a unit concentrated force at the selected nodal degree of freedom. The value of the unit force depends on the chosen system of units.

In the case of SI units, a force of 1 Newton or a moment of 1 Newton metre will be applied.

❏ **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.

The Large Mass Method

Interactive Modal Dynamics frequently employs an analysis technique referred to as the Large Mass Method. The reason for this is to earth the structure via a 'moveable' object rather than a strict support to ground. This allows subsequent application of a force to the mass, in effect applying an acceleration to the structure. The size of this mass should be sufficient to ensure **mass dominated** local response, so that the motion of the point is described by Newton's Second Law (i.e. F = ma). A mass of 1E6 kg works well for most structures. Unduly large values may cause ill-conditioning problems. A force equal to the large mass is applied at the **support** point, thereby inducing a unit acceleration response.
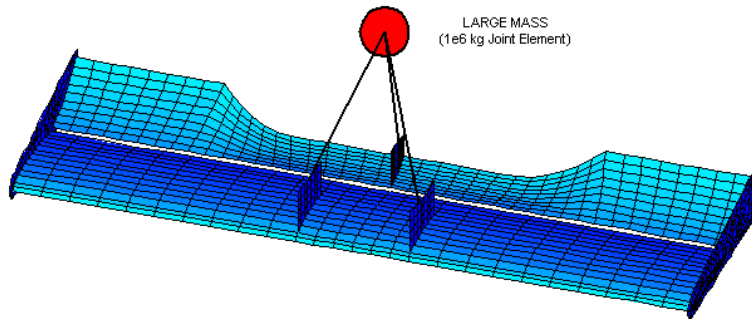
Modal Dynamics Results Types

❑ **Frequency domain response** Harmonic, or forced, response analysis is used to investigate the effects of structural resonance (where structures are forced to vibrate harmonically at or near their own natural frequencies). Solution of the harmonic response problem as a modal analysis avoids the need to perform a full transient dynamic analysis. Simultaneously applied excitations may contain phase differences.

❑ **Time domain response** (impulse or step-by-step dynamics) to dynamic excitation. The forcing function and the consequent response of a structure are defined in terms

of time histories. The Fourier transform of the time domain gives the corresponding quantity in the frequency domain.

❑ **Spectral response analysis** An analysis in which a defined response spectra for a generic earthquake ground motion is used to estimate the maximum displacement or pseudo-velocity or acceleration during the earthquake, without the need for direct integration of the model over the complete duration of the event. Dynamic excitation is applied to all the supports simultaneously. A response spectrum curve defines the magnitude of excitation. If the damping in the response spectrum curve differs from that defined for the model a damping correction may be applied using one of formula provided. The maximum displacements, forces and stresses are computed throughout the structure for each eigenmode. These values may then combined to produce a single positive result using a spectral combinations. The spectral combinations available are CQC (default), SRSS and Absolute Sum. For further details see the *LUSAS Theory Manual*.

❑ **Power Spectral Density** (PSD) Analyses the frequency response to a random modal vibration, such as aerodynamic loads acting on an aircraft component. A frequency PSD defines the frequency content of the random loading. Dynamic excitation should be applied to all the supports via support motion excitation.

Frequency and Time Domain Response

Frequency and time domain response calculations are the most commonly used. Usually, a node is excited across a frequency (or time) range to generate a graph for the frequency or time response (using the **Utilities> Graph Wizard** menu item). From the response across a range, a single frequency may be selected to perform an IMD calculation on the whole structure (using an IMD loadcase). The following diagrams illustrate this procedure:

Results across a frequency range, or time history

Response at a node calculated across a specified frequency range
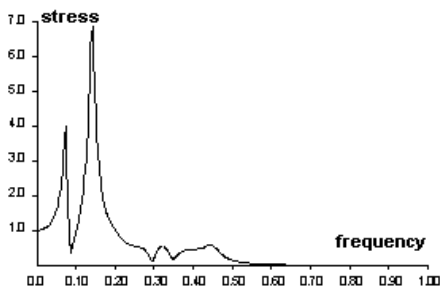
Response at a node calculated across a specified time history

Initial Eigenvalue analysis

mode 1   mode 2

mode 3   mode 4
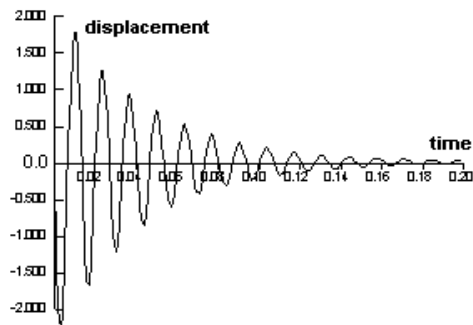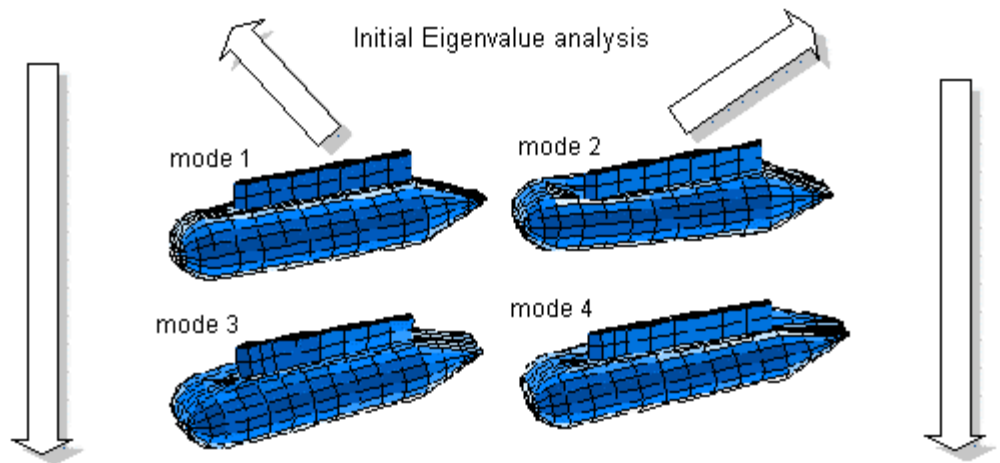
Results at a specific frequency, or time step

Response calculated at a specified frequency for the whole structure

Response calculated at a specified time for the whole structure
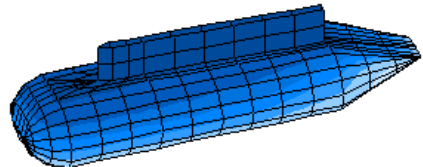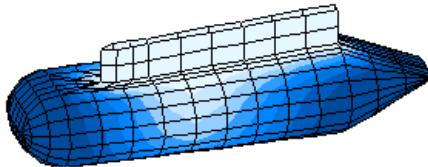
Frequency Power Spectral Density

A Power Spectral Density (PSD) defines the frequency content of a random loading, such as turbulent pressure acting on an aircraft component, and are for use in modal random vibration response analysis. At present, random modal vibration calculations are restricted to single-input systems, where the loading at all points is fully correlated.

The value of PSD used in response calculations will be interpolated from a table of frequency/PSD values. A range of linear and logarithmic interpolation schemes are provided, in accordance with typical PSD specifications. Dialog input requires: Interpolation type, options are: Linear/Linear, Log/Linear, Linear/Log, Log/Log; Frequency; Amplitude. More copious PSD tables may more easily be defined by copying and pasting the data from a text file or spreadsheet.

Response Spectrum

Response spectrums for use in spectral analyses are defined from the IMD loadcase dialog.

The value of the frequency used in spectral calculations will be interpolated from a table of frequency-amplitude or period-amplitude values. More copious Response Spectrum tables may more easily be defined by copying and pasting the data from a text file or spreadsheet.

- ❑ **Frequency/Period/Displacement/Velocity/Acceleration** The value of the frequency used in spectral calculations is interpolated from the values defined. The type of values entered, i.e. displacement, velocity, or acceleration should match the type of support motion which is used to excite the structure. For earthquake analyses it is usual to specify Acceleration, and to later specify a support motion using acceleration.
- ❑ **Spectral curve damping** This value defines the percentage damping inherent in the response spectrum curve itself. If the Eurocode or Kapra damping correction formula are specified, the spectral response curve is adjusted to the viscous modal damping value specified in the IMD loadcase. When using other damping correction formulae the spectral curve is adjusted using only the viscous damping. For more details see *Theory Manual*.

Case Study. Forced Vibration of a Simply Supported Cantilever

Consider the forced vibration of a simply supported cantilever beam. The beam is supported at one end and subjected to a uniformly distributed load. An eigenvalue analysis is carried out for 9 modes. Normalisation with respect to Global mass must be selected when defining the eigen control.

Calculation of the modal results in the frequency domain is required for **Displacement** type response for the end node for the **Y-direction** displacement. Results are to be calculated over a suitable frequency range using a specified frequency step. The results type required will be **Amplitude**.

Modal response calculations are carried out as follows:

1. Read in the results file from the eigenvalue analysis. Use **File> Open** and specify the results file name.

2. Click on **Utilities> Graph Wizard**. Choose **Modal Expansion** then click on the **Next** button.

3. Click on **Frequency** to specify the frequency domain.

4. Choose a **Point** force excitation, then click on the **Set** button to define the parameters. Excite the structure at the unsupported end node in the Y-direction. Enter the node number or, if the node is selected, choose the number from the drop-down list. Click on the **OK** button, then click on the **Next** button.

5. Specify the results entity as **Displacement**, component **DY** and the results type as **Amplitude**. Take care to specify a realistic start, end and frequency step. Specify which node to calculate the response at, then click on the **Next** button.

6. Specify a Title and X and Y axis labels, then click on the **Finish** button. This will create a standard frequency vs. amplitude plot for the response from the specified excitation. Two graph datasets will be created, the first containing the frequency range in the steps specified and the second containing the required amplitude values.

User Defined Results

Defining expressions
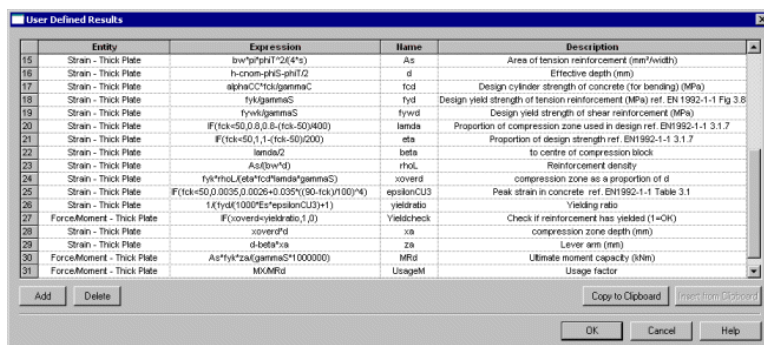
The User Defined Results dialog is accessed from the **Utilities> User Defined Results** menu item. It allows results components to be defined by creating arithmetic expressions based upon LUSAS results entities, components, model data and other user-defined results component calculations. A component name and description can also be entered.

A results file must be loaded in order to define a user results expression.

- Clicking on the drop-list button ![icon] in the Entity cell lists all entities available for the particular results file loaded.

- Clicking on the launch dialog button ![icon] in the Expression cell displays a dialog populated with valid variables for the selection made in the Entity field. By use of these variables and usual arithmetic syntax a user defined results expression can be built and assigned a name and a description.

For details of expressions and functions supported see **Input and Output of Real Numbers in LUSAS** in Appendix E.

Using named expressions

After definition, the user 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.
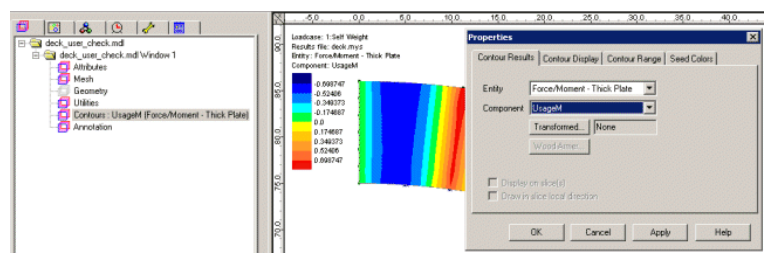
Visualising The Results

Results visualisation is performed using results **layers** in the ![icon] Treeview . Each method of viewing results, described below, involves adding a layer to the 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.

❑ **Contour** Display the results on the model as colour fringes or lines of equal value.

❑ **Vector** Displays vector results quantities as an arrow (or pair of arrows) on the model.

❑ **Values** Marks result values on the model as symbols and/or values.

❑ **Diagrams** Displays beam element shear force and bending moment diagrams.

In addition to those layers listed above, an Annotation layer is used to hold contour key and other annotation data that may be added to the results viewing window.

All results visualising will use the results from the **active loadcase** for the current window. The active loadcase is shown with a coloured icon in the 🔍 Treeview, and may be changed using the loadcase context menu. Non-active loadcases are shown with a greyed-out icon.

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 contained in the current window. Alternatively, with nothing selected, right-click in the graphics area to display the context menu and add the appropriate layer. Another alternative is to select a layer name from the context menu of the Window name in the 🗖 Treeview.
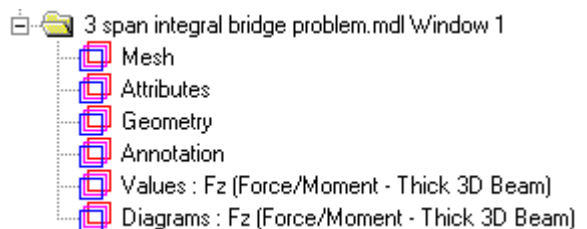
The display of layers in the current window can be turned on or off by right-clicking on the layer name in the 🗖 Treeview and selecting / deselecting the **On/Off** option. See **Using Layers** for more information

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, or when creating an animation sequence, use a global scale (and a global contour range) so that scaling and contouring in each frame is relative to the first frame.

Layer names for results layers

Contours, Diagrams, Vectors and Values results layer names as added to and seen in the Layers 🗖 Treeview have the component (such as Mx, My or Fz, for example), appended to the layer name followed by the entity type such as

└─ 📁 3 span integral bridge problem.mdl Window 1
     ├─ 🗖 Mesh
     ├─ 🗖 Attributes
     ├─ 🗖 Geometry
     ├─ 🗖 Annotation
     ├─ 🗖 Values : Fz (Force/Moment - Thick 3D Beam)
   └─ 🗖 Diagrams : Fz (Force/Moment - Thick 3D Beam)

(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.
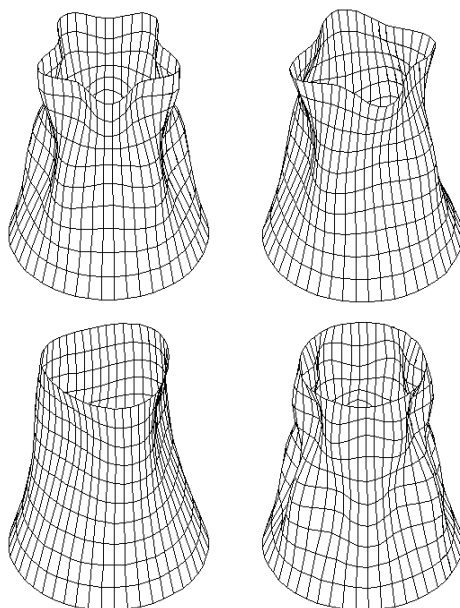
Comparing The Deformed And The Undeformed Shape

Drawing the mesh and the deformed mesh together but in different pens is useful for visualising the deformations.
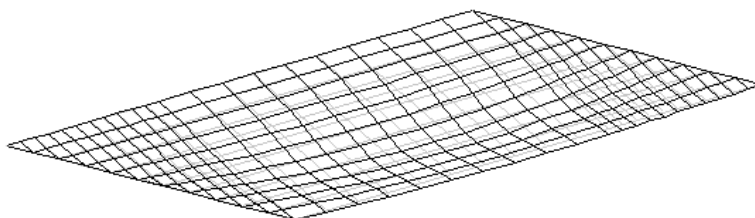
In the example of the grillage shown, the scale of the deformed plot is greatly exaggerated using a scale factor of 10. The undeformed plot is drawn using a lighter pen colour to further clarify the display.
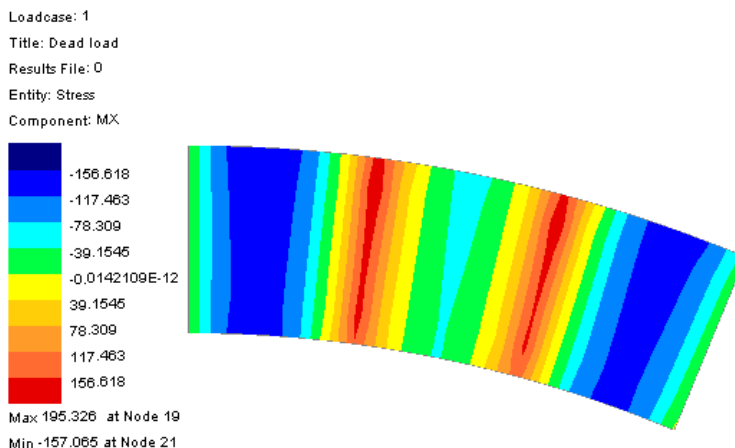
Notes

- Other results visualisations may be displayed on the undeformed or deformed shape as required.

- When carrying out contact analysis a unit deformation factor should be used to avoid misleading results.

- When an analysis involves the activation and deactivation of elements, inactive elements may be hidden using the **show activated only** option on the visualise page of the mesh properties dialog.

Contours

Contours display the results of the **active loadcase** on the model as colour fringes or lines of equal results value.

Contour Display Features

❑ Contours may be plotted using colour fill and/or contour lines. Fill and line contours displayed together are useful for emphasising the contour levels. Contour labels are available if required.

❑ Contours may be plotted on the deformed (loaded) shape or the undeformed shape.

❑ Contours may be plotted using averaged nodal results to give a smoothed plot, or unaveraged nodal results to contour the results on an element-by-element basis, revealing any inter-element discontinuities. This is useful for checking mesh discretisation error and for displaying results across geometry or material discontinuities.

❑ Contours can be plotted on shells, solids, bar and beam elements, on fleshed members and on layers of composite elements.

❑ The maximum and/or minimum results value and node location can be annotated.

❑ The appearance of the contour key can be adjusted to specify the number of significant figures, draw an outline around each colour in the key and draw red or blue uppermost.

Loadcase: 1
Title: Dead load
Results File: 0
Entity: Stress
Component: MX

-156.618
-117.463
-78.309
-39.1545
-0.0142109E-12
39.1545
78.309
117.463
156.618

Max 195.326 at Node 19
Min -157.065 at Node 21

Setting the Contour Levels

By default the levels at which contours are plotted are calculated automatically but they can also be set explicitly. A contour level corresponds to the boundary between adjacent colour fringes, or the line of equal value.

❑ Setting the range automatically can be done by specifying either the number of contours or the interval value between contours.

❑ Automatic contours levels can be set to pass through a certain value. The maximum and minimum contours may also be specified.

❑ A global range can be used to fix the contour levels between different contour displays. This is often used before animating contours, so that all animation frames use the same contour range. It is also useful to fix the contour levels manually when comparing results from different loadcases or when using multiple windows.

Tip. Contours use the colour map to define the colours used. It is often useful to adjust the colour map so that low stress contours are set to white. This makes contour plots easier to understand, and also avoids excessive use of single colour ink when printing.

Vectors

Vectors are used to visualise both the **magnitude** and **direction** of specified results components. Vectors may be displayed at either nodes or Gauss points.

Scale

By default, vectors will be scaled such that a vector length of 6mm represents the maximum displayed results component. Alternatively, a scale factor can be specified.

Style

Vectors may be drawn as lines or arrows. By default the colours used for drawing vectors in tension and compression are red and blue but the pens can be altered as required.

Deformed Shape

This image shows displacement vectors displayed on the deformed shape.
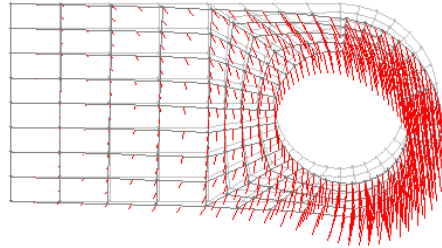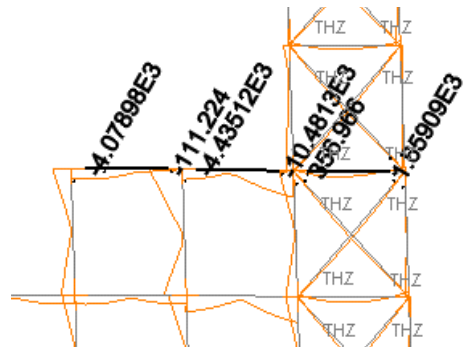

Values

Values are used to identify the location and value of results for both averaged nodal (smoothed), element nodal (unaveraged) and Gauss point values. Either maximum and/or minimum values may be visualised, and a percentage may be specified to determine whether values lying within a maximum/minimum range are displayed. Values may also be displayed at selected nodes.

The Values layer is also used to display the locations of yield symbols and crack/crush planes.
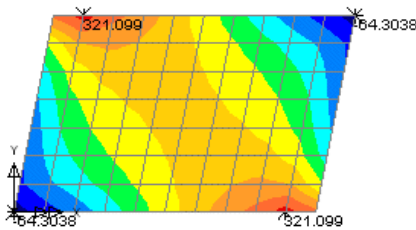
Maximum and minimum values marked

Values plotted for selected (shrunk) elements

Visualisation of yielded material
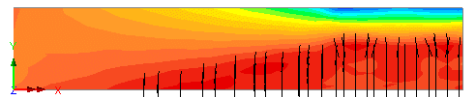
Visualisation of concrete crack patterns on a 2D model

Tips:

- The percentage range sets the range of values to display, starting from the maximum and/or maximum values. Setting this value to 100% displays all values, this is useful for displaying nodal results on the screen for a subset of a model. Setting this value to 0% shows only the extreme maximum and/or minimum setting.

- To prevent text labels from overlapping each other the elements on which that are drawn may be shrunk by specifying the % of elements remaining on the Mesh layer properties dialog. Additionally text may be given a rotation angle on the Values layer properties dialog to prevent text labels from overlapping each other.

- Use the **Show values of selection** option to isolate only those values of interest, or to restrict labelling on complex models to selected nodes and elements only.

- Gauss point values display the computed values from the analysis. These values may be particularly useful when examining results for nonlinear materials.

- Unaveraged values display the computed values after they have been extrapolated to the element nodes but before they have been averaged. These values may be of particular interest when examining results around discontinuities in geometry of materials.

Diagrams

Bending moment and force diagrams may be drawn for any 2 or 3 dimensional frame structure comprised of **bar** or **beam** elements. All of the results for diagrams are located within the Stress results entity, and the results components available will depend upon the element type (see *Element Reference Manual*). The diagrams may be drawn using the element axes or screen axes.

The following quantities may be represented:

❑ **Axial Forces** local x direction beam forces.

❑ **Shear Forces** through-thickness shear forces.

❑ **Bending Moments** bending moment results.

❑ **Torsional Moments**
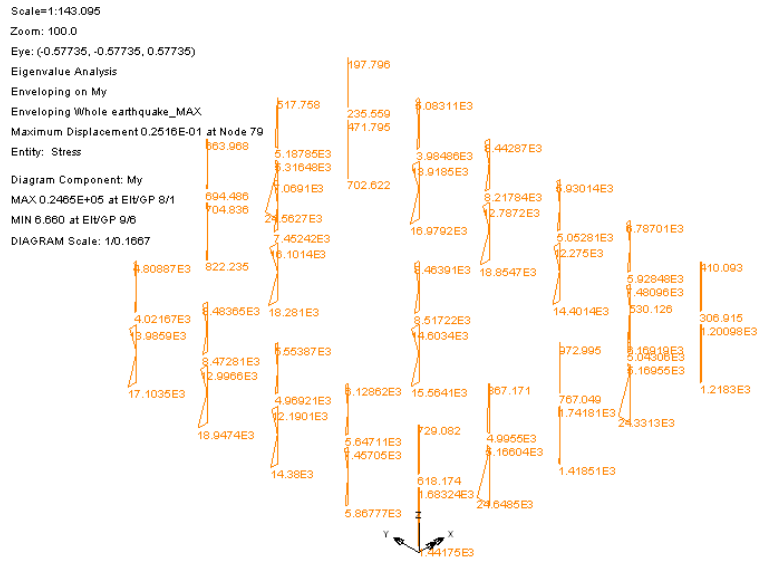
Plotting Results for Groups

By default results are computed and displayed on the visible model. When appropriate layers are present in Layers ▣ Treeview, results can be selectively plotted for groups held in the 🔶 Treeview by choosing the following Results Plots context menu items for each named group:

❏ **Show Results**

❏ **Do Not Show Results**

❏ **Show Results Only On This Group / Attribute**

When combined with group visibility options that can also be accessed via the context menu for each group name parts of the model can be isolated and have results plotted just for those regions.

Pairs of symbols adjacent to each group name in the 🔶 Treeview show the status of model visibility and results display.

When viewing results:

✓ (green tick) All of the objects in this group are visible, but no results are being shown.

✓ (blue tick) Some of the objects in this group are visible, but no results are being shown.

✗ (red cross) None of the objects in this group are visible and no results are being shown.

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

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

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

Notes

- When choosing to show results for a material, geometric property or element mesh type that does not support (for example) the same Contour entity as that used for the previously plotted item, no results will be shown until a valid Entity and Component for the new selection is picked on the Contours property dialog accessed via the Layers Treeview.

- A black dot next to a group symbol denotes the current group into which all new geometry will be added when created and hence has no relevance when viewing results.

Plotting Results for Assigned Attributes

By default results are computed and displayed on the visible model. With appropriate layers present in Layers Treeview, results can be selectively plotted for attributes held in the Attributes Treeview by right-clicking on the attribute name and then choosing Results Plots. The following context menu commands are available for Results Plots:

❑ **Show Results**
❑ **Do Not Show Results**
❑ **Show Results Only On This Attribute**

When combined with visibility options that can also be accessed via the context menu for each attribute selected features of the model can be isolated and have results plotted just for those features. This provides a means of producing isolated results for particular material types, geometry, or element mesh types without having to define individual groups for each of these items. But, if required, groups of features or elements may be defined and the Results Plots entry may be used to display selected results for a chosen group.

Notes

- When choosing to show results for a material, geometric property or element mesh type that does not support (for example) the same Contour entity as that used for the previously plotted item, no results will be shown until a valid Entity and Component for the new attribute selection is picked on the Contours property dialog accessed via the Layers Treeview.

Nonlinear Material Results Display

Two types of nonlinear material results may be displayed. Yield flags and crack/crush patterns. The availability of each depends on the material model assigned to the elements during the modelling stage. Crack and crush patterns are available for concrete models and yield flags are available for all nonlinear materials.

❑ **Yielded material**

Available from the **Values layer**, specify the results entity as **Stress**, and the type as **Yield**. Yield flags show the extent of the yielded material within a structure and are plotted at Gauss points. The nonlinear example here demonstrates how the spread of yielded material is visualised using a symbol at element Gauss points.
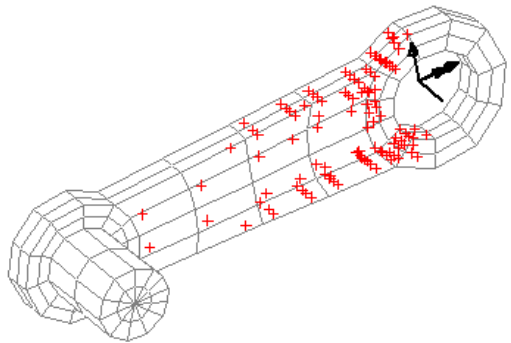
This display is especially useful when used in conjunction with contours or an animated sequence when the onset and spread of yield can be highlighted.

❑ **Crack and Crush Patterns** Crack and crush patterns can be displayed for models that use the Concrete material model. Crack patterns are visualised using the **Values layer** by specifying the Results Entity as **Stress** and the Type as **Crack/Crush.**

The patterns displayed show the extent of cracking/crushing and the orientation of the cracked and crushed planes.
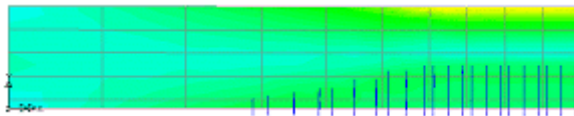
Crack pattern
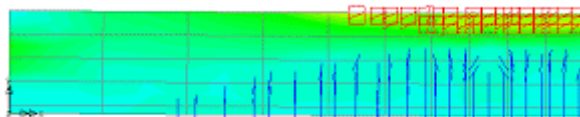
Crack and crush pattern

Results On Sections / Slices Through A Model

Results can be plotted on sections or slices through a model by choosing the following menu commands available from the **Utilities** menu:

❑ **Section through 3D** Cross-sections may be taken through a three-dimensional solid model to display results on a slice. Results are calculated at pseudo-nodes formed at the intersections of the slice with the element edges by linear interpolation of the nodal results.

❑ **Graph through 2D** Graphs of results may be created along a line through a 2D continuum model or on a line through a section from a 3D model. Results are calculated at the intersections of the line section and the element edges hence a finer mesh will produce more sampling points. Force and moment values along the slice may also be computed.

Section Through a 3D Model

Slice sections may be cut at arbitrary positions through the model using the cursor to define either a horizontal or vertical slice in the View window. Slices may be generated in any plane by rotating the model to the desired orientation before a section is cut.

By default the location of an arbitrary slice section through a model is not saved with the model. However, when cutting a slice section through the model an option to create an annotation polygon is provided. This annotation polygon effectively defines the location and orientation of the cutting plane and does get saved with the model. Annotation polygons may be re-selected if a model is reloaded at a later date in order to create a slice at the same location.

Slice sections can also be created using surfaces that are created at slice section locations. These surfaces do not have to surround a model, they must simply be defined in the orientation of the cutting plane required.

To use a saved cutting plane or a surface defining in cutting plane instead of indicating the cut using the cursor, the annotation polygon or surface should be selected before choosing the **Utilities> Slice through 3D** menu item, and **By selected polygon / surface** option.
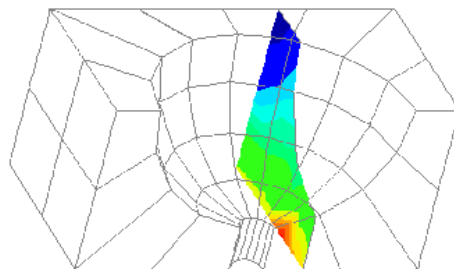
Notes

- No results visualisation or printed results for slices are available unless the **Display on slice(s)** option has been selected on the Contours and Values properties dialogs or on the Print Results wizard dialog.

- 3D sectioning/slicing is only available for 3-Dimensional solid models. For surface models refer to **Graph Through 2D** below.

- A slice can form the basis for a line section using **Graph Through 2D** to graph results along a line through the centre of a three dimensional solid.

Manipulating The Slice

Once a 3D slice section has been defined, a slice exists as a **group** in the Treeview, and may be manipulated using the **View> Group** menu item or from the group's context menu. For more details on viewing results on groups see **Plotting Results for Groups**.

Notes

- The mesh and nodes on the slice are displayed using the mesh layer properties.

- The slice local axes and origin are displayed and moved from the slice properties accessed from the slice context menu.

- The resultant local forces on the slice and the slice properties may be printed from the slice context menu.

- Groups created from slice sectioning a 3D model cannot currently be retained when a model is saved.
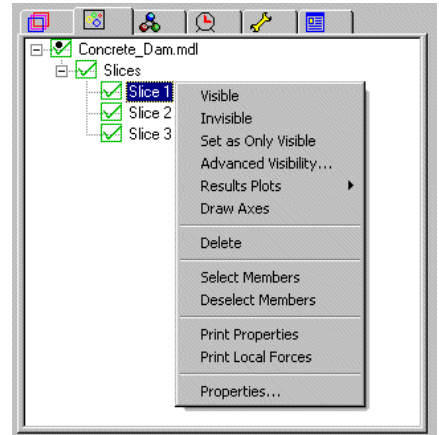
Graph Through 2D

Arbitrary line sections may be taken through any surface model or on a slice cut through a three dimensional solid model. The process of cutting a slice will generate two graph datasets, the first containing the distance along the line section and the second containing the specified results along the line. The graph datasets are plotted automatically using the **Graph Wizard**. Graph datasets are stored in the Utilities Treeview.

By default line sections may be cut at arbitrary positions through the model using the cursor. Lines can also be defined to start and finish at points located on an underlying grid. When cutting a line section through the model an option to create an annotation line is provided. This may be used later for repeating the cut if a graph along the same line is required. To use an existing line (or annotation line), instead of indicating the cut using the cursor, the line (or annotation line) should be selected before choosing the **Utilities > Graph through 2D** menu item.

Notes

- Care should be taken when slicing through voids, or holes, in the model as this can give misleading results.

- Care should be taken when slices pass through parts of the model with non-uniform properties, such as parts of different materials or of different thickness.

- Sometimes, due to model size, grid points are too close together to be usable. In these cases simply increase the grid size so that individual grid points can be selected.

Graph Wizard

Two types of graphs may be plotted:

❑ **Results quantities** Any available result entity may be plotted against distance along the slice.

❑ **Axial force and bending moment** This generates three datasets of distance, axial and bending stress along a line through a section: the axial force per unit width, moment per unit width and distance of the neutral axis to the midpoint of this line. For axisymmetric solids the moment per unit radian is printed as well. From these datasets, graphs of axial and bending stress versus depth of section are plotted.

For more details on viewing results on graphs see **Plotting Results on a Graph**.

Displaying Beam Stresses

Beam stresses can be displayed on the fleshed section or at selected **fibre locations**. These stresses are computed using engineering beam theory which assumes that the normal stress is constant across the width of the beam cross section. This assumption can introduces significant errors due to shear lag when wide flanged sections are being used so these stresses should be used with caution.

Beam stresses can be displayed as contours on the fleshed section or as values, diagrams or beam contours at fibre locations using the standard layer controls. When viewing stresses at fibre locations the value, diagram or contour is drawn at the actual position of the fibre on the cross section.

Examples follow:

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

Maximum 7.97335E6 at Gauss point 1 of element 11

Minimum -8.05031E6 at Gauss point 1 of element 9
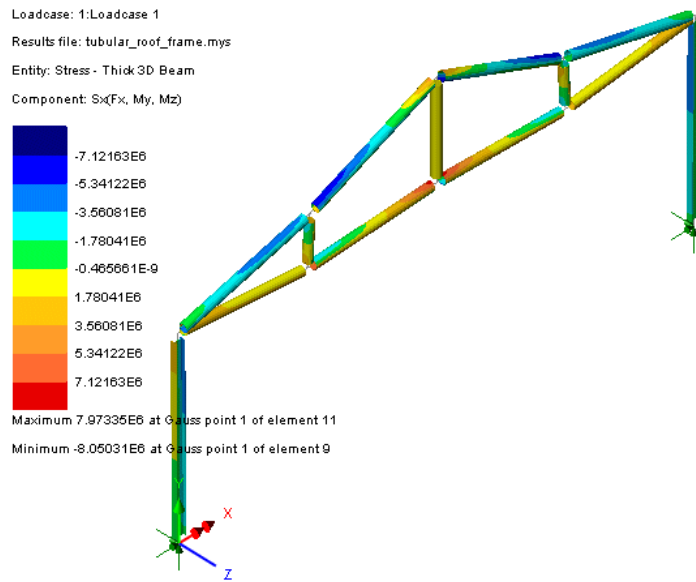
Filled contours plotted on fleshed beam sections

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

Maximum 7.97335E6 at Gauss point 1 of element 11

Minimum -8.05031E6 at Gauss point 1 of element 9

Line contours plotted on fleshed beam sections

Loadcase: 1:Loadcase 1
Results file: tubular_roof_frame.mys
Fibre: R2
Cross Section: Line Geometric 2
Entity: Stress - Thick 3D Beam
Component: Sx(Fx, My, Mz)

-6.5655E6
-4.92412E6
-3.28275E6
-1.64137E6
0.465661E-9
1.64137E6
3.28275E6
4.92412E6
6.5655E6

Maximum 7.97335E6 at Gauss point 1 of element 11
Minimum -6.79901E6 at Gauss point 11 of element 9

Line contours plotted at top beam fibre location

Loadcase: 1:Loadcase 1
Results file: tubular_roof_frame.mys
Fibre: R4
Cross Section: Line Geometric 2
Entity: Stress - Thick 3D Beam
Component: Sx(Fx, My, Mz)

-6.6375E6
-4.97813E6
-3.31875E6
-1.65938E6
0.465661E-9
1.65938E6
3.31875E6
4.97813E6
6.6375E6

Maximum 6.88407E6 at Gauss point 11 of element 11
Minimum -8.05031E6 at Gauss point 1 of element 9
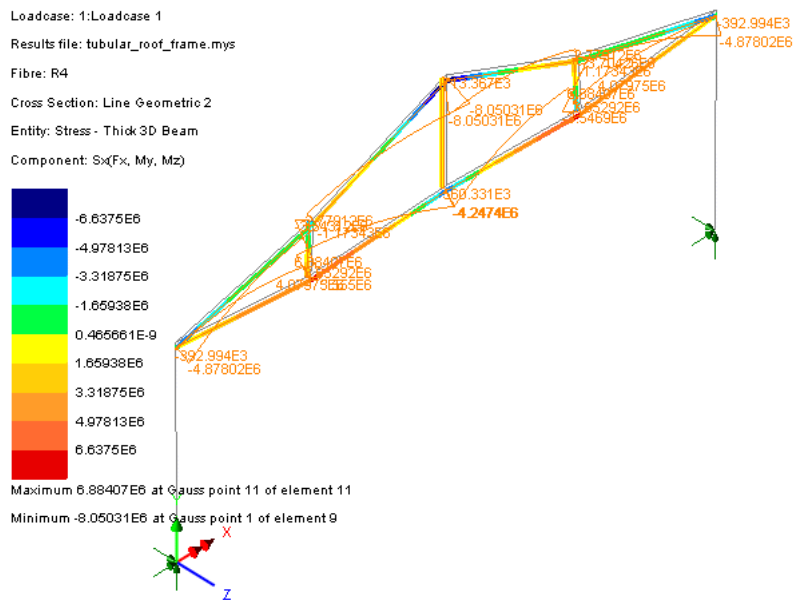
Line contours and diagram stress results plotted at bottom beam fibre location

Fibre locations

When plotting selected contour or diagram results for beams, a fibre location must be used to specify the place(s) on the beam section at which the results should be calculated and plotted. Results for individual fibres are plotted by setting that fibre location active for a particular geometric line attribute in the 🔩 Treeview. The active fibre is denoted with a black dot next to the fibre name.

🔩 R1

- Right-clicking on a fibre name in the 🔩 Treeview and choosing **Set Fibre Active** will display results for just that fibre.
- Right-clicking on a fibre name in the 🔩 Treeview and choosing **Set Fibre Nam**e will display results on all members with that fibre name.
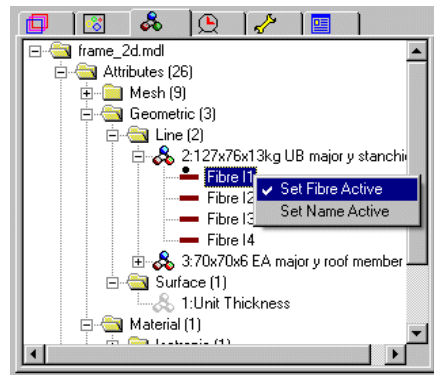
Notes

- 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

The **slice resultants from beams and shells** facility allows the computation of the equivalent beam stress resultants for flat or curved thin and thick shell models. This allows the conversion of the results of a complex shell model into an equivalent beam analogy for use in design codes of practice. The slice resultants are computed using valid visible 3D beam and shell elements. Invisible elements are ignored.

Slice locations

The **slice locations** are defined using a path which can consist of straight lines and arcs or combined lines that contain straight lines and arcs in the selection. The path must be continuous without any branching characteristics. The slice path orientation is defined by either the order of selection when more than one line/arc is selected or the orientation of the

line/arc if only a single line/arc has been selected. The locations for the slices along the path can be defined using three methods:

❑ **From points or nodes in the selection** If nodes or points are selected, these are projected onto the path perpendicular to the appropriate path segment tangent to obtain the slicing locations

❑ **Incremental distances from start of path** Incremental distances can be entered to define the distances along the path for the slicing. Distances can be both positive and negative but the running total distance should remain within the length of the path. For example, 1@0;1@10;1@-5 will cut slices at distances of 0, 10 and 5 along the path

❑ **Absolute distances from start of path** Absolute distances can be entered to define the distances along the path for the slicing. Distances must be positive and within the length of the path. For example, 0;5;10 will cut slices at distances of 0, 5 and 10 along the path

❑ The **Distance from reference origin to start of path** can be also be entered. If this value is non-zero, the value is added to the distances along the path.

The orientation of the slice local axes is defined from both the tangent of the path at the location of the slice and the direction that the model is viewed from. If the slicing path is in the plane of the screen the positive slice local z axis will be defined by the tangent of the path and the positive slice local y axis will be orientated in the out of screen sense. If slicing path is not in the plane of the screen, the positive slice local y axis will be defined perpendicular to the path tangent in the path tangent/out of screen plane. The positive slice local x axis will be defined perpendicular to both the slice local y and z axes.

❑ Options are available for the calculation of the moments 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.

Slice options

Additional options are available for the control of the slicing. These include:

❑ **Effective width** If the effective width option is selected, the width of visible elements to include in the calculations can be specified. This effective width is centred on the slicing path in the screen plane at a perpendicular to the path. If the effective width is not used, all valid visible elements will be included in this direction. For both options, the slice is infinitely deep in the slice local y axis

❑ **Include whole elements only** If the effective width option is selected, the option to include whole elements only is available. If selected, partial elements intersected by the current slice will be ignored in the calculations

❑ **Smooth corners on path** If the smooth corners on path option is selected, the average tangent of the path at a connection between two lines/arcs will be used for the slicing if the distance along the slice path exactly matches this path connection. If the smooth corners on path option is not selected, two slices will be taken at the connection using the tangents for both of the lines/arcs connecting at this location

❑ **Slicename prefix** Allows user to input a user defined prefix for the slice names

Loadcase

Slice resultant results can be output for one or more loadcases

❑ **Active** - prints slice results for the active loadcase to a slice output window.

❑ **All** - prints slice results for all loadcases to a text file named SliceResultantsBeamsShells.prn in the current working directory.

❑ **Selected** - prints slice results for entered loadcases to a text file named SliceResultantsBeamsShells.prn in the current working directory. As an example, entering 1-5,7 would select loadcases 1 to 7 excluding loadcase 6.

Notes

- The assumption that plane sections remain plane which is required for the calculation

- Linear variation of stress is assumed for the approach and therefore low order flat shells are supported (3 or 4 noded thin and thick shells) and high order flat shells are calculated ignoring the mid-side nodes if they are coplanar with collinear edges. For curved high order shells the element is subdivided into constituent pseudo elements and each pseudo element interrogated using linear interpolation

- The slicing path must be defined using straight lines, arcs or combined lines containing only these two line types. Splines and annotation lines cannot be used

- Slice forces can only be computed relative to the intersection of the slicing path with the slice plane or the neutral axis based on the sliced section. No facilities are available for the transformation of slices without defining a separate path and recalculating the slices

- Taking slices at the free end of a structure can lead to overestimation of the forces and moments on the section. This occurs due to the stresses in the section not returning to zero at a free and unloaded end

- Taking slices at a supported end of a structure can lead to discrepancies in the forces and moments when compared to a beam model due to the end effects taken into account by the full 3D shell modelling

- Where a slice passes through a node in the structure, results are presented for the contributions of the elements on both sides of the slice. This is presented as results for the negative and positive local Z sides of the slice

- Engineering thick beam elements are supported (BMS3 and BTS3) along with thin beam (BS3, BS4, BSL3 and BSL4) elements

- Option 380 must be used for BTS3 elements when using eccentricitices/offsets

- When using 3 or 4-noded shell elements the mesh density should be sufficiently fine to capture the behaviour of the structure

- When modelling curved structures it is recommend that regular quadrilateral elements wherever possible

- Printing results for more than one loadcase will overwrite the text file named SliceResultantsBeamsShells.prn each time.

Slideline Results Processing

When a results file is read into Modeller from an analysis that involves slidelines, a group is created for each slideline surface in the model. These groups can be accessed from the Group Treeview.

A context menu for each group name provides options for making the members of a group visible or invisible and for selecting and deslecting the members of a group.

Limited results for a selected slideline group can be printed by using the Print Results menu item on the context menu for that group. Full slideline results for the model as a whole can be printed using the Print Results Wizard.
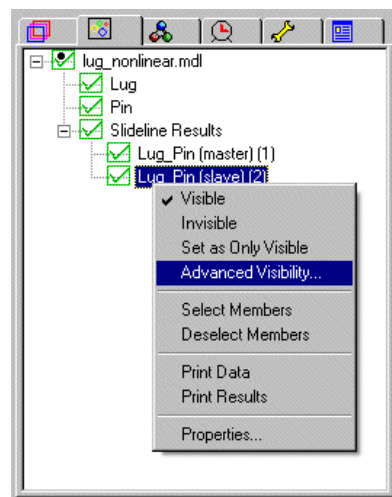
Printing Slideline Results

The print results wizard is accessed from the Utilities menu. When Slideline results are selected the following six types of results can be chosen for printing:

- ❑ **Summary**
- ❑ **System forces**
- ❑ **Gap Forces**
- ❑ **Contact forces**
- ❑ **Contact Stresses**
- ❑ **Section Results**

Each type of result is described below.

361

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 is used to draw XY graphs. The following results graphs may be plotted if the results are available:

❑ **Time history** A history for a specified results type throughout an analysis with respect to time or increment. Graphs may be plotted for a named variable such as Total Load Factor or Response Time verses a specified results quantity such as displacement in X, equivalent stress or sum of reaction over a set of specified nodes. The following history datasets are available:

- **Nodal** Averaged or summed.

- **Gauss Point** results for a selected element and Gauss point (Element Gauss numbers may be determined using the labels layer).

- **Named** Named result variables for a linear analysis such as loadcase ID, a transient analysis such as response time, or a nonlinear analysis such as total load factor.

- **Strain energy and plastic work** Total strain energy or total plastic energy for the elements showing results.

- **Previously defined**

❑ **Fourier expansion** The displacements, stresses and strains output from a Fourier analysis are coefficients of corresponding sine and cosine functions. The evaluation of

these functions around the circumference of the model is achieved graphically using the Graph Wizard.

❑ **Modal expansion** Graphs the modal response of a structure to dynamic excitation using the results from an eigenvalue analysis. Various results entities may be plotted against a frequency range or sampling time for selected eigenmodes.

❑ **Load curve** Graphs a defined load curve.

❑ **Variation** Graphs a variation function.

❑ **Specified datasets** Graphs two previously defined datasets.

❑ **Thermal surfaces** Graphs results along a thermal surface.

❑ **Slideline (assigned to line)** Graphs results along a slideline**.**

Note: To graph a specified results entity against distance along a slice through a planar structure or on a slice of a three dimensional solid structures see **Graph Through 2D**.

Graph Properties

LUSAS uses the **Graph Wizard** to take you through each step of creating the X and Y datasets and placing them into a graph. The graph wizard is started from the **Utilities** menu. The X and Y datasets are then stored in the Utilities Treeview 🔧.
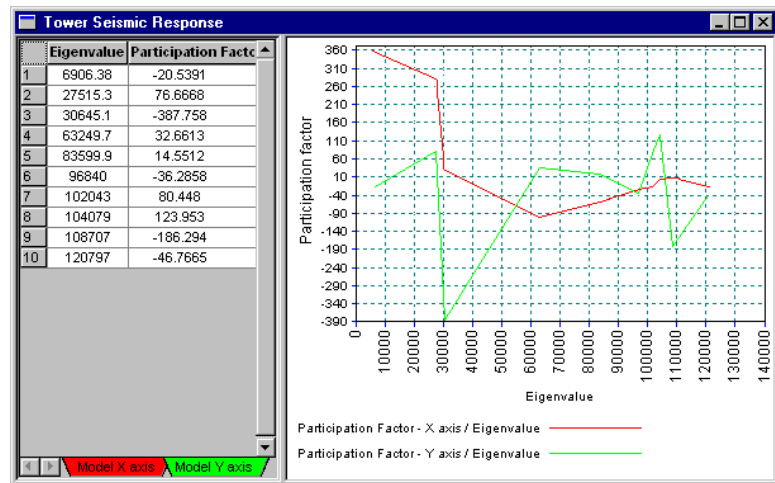
The graph window is split in to the **graph area** on the right and the **graph data table** on the left.

Tip. Zoom in on a part of the graph by boxing with the mouse. To unzoom right-click on graph area to display the context menu and select **unzoom.**

Plotting Families Of Curve Data On The Same Graph

If a graph already exists, then further curves may be added to the first graph by choosing the **Add to existing graph** option of the final page of the Graph Wizard. In this way families of curves may be drawn on the same graph using a different colour for each one.

Editing Graphs

Once a graph has been plotted, the appearance and even the data on the graph may be modified. This may be done by selecting the **Edit> Graph Properties** menu item from the graph context menu.

Editing Graph Data

To change individual data points on the graph (for example to add an origin to a curve), make the graph data table editable by checking the **Editable graph table** box on the General Graph Properties page of the Graph Properties dialog. The following facilities are then available:

❑ New data points may be added by adding new rows to the grid either by using **Insert Row** from the **Edit** menu or by pressing the **Tab** key when the cursor is in the final cell.

❑ Data points may be deleted by deleting the data in the grid.

❑ Blocks of data from a spreadsheet may be pasted. New rows will be added to the grid as necessary.

Note. These changes are made to the graph only and are not stored in the corresponding graph dataset.

Pasting Graph Data To Spreadsheets

To paste graph data to a spreadsheet, use **Copy** from the **Edit** menu when the data has been highlighted in the graph data table, then paste into the spreadsheet.

It is also possible to export selected or all graph datasets to a .csv file by using the context menu for the Graph entry or graph dataset name in the Utilities treeview. Columns of data are created with graph dataset names being written to row one of each column.
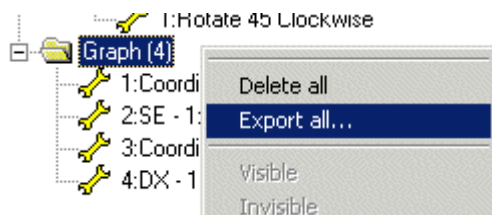
Printing Graphs

Graphs may be printed using the **File> Print** command. Because graphs are created as separate windows a single graph will be printed on each page.

To print multiple graphs together use **Copy** from the **Edit** menu when the graph area is active, then paste into a suitable word processor.

> ### Case Study. Plotting Families Of Curves
>
> To compare results at different nodes a graph will be plotted of the stress throughout an analysis at three nodes, say 6, 25, and 60. Using a suitable results file, first select the nodes with the cursor, then:
>
> 1. Start the **Graph Wizard** from the **Utilities** menu.
>
> 2. Choose **Time History**, click **Next**.
>
> 3. For the X axis choose **Named** variable, click **Next**, choose **Response Time**, click **Next**.
>
> 4. For the Y axis choose **Nodal**, click **Next**, specify the results entity as **Stress**, specify node number **6**, click **Next**.
>
> 5. Either type suitable graph and axes titles, or leave them blank to use default names, click **Finish**. The graph is displayed.
>
> 6. Repeat steps **1** and **2**.
>
> 7. For the X axis choose **Named** variable, click **Next**, but this time choose **Previously defined**, click **Next**. Select **Response Time** from the drop-down list, click **Next**.
>
> 8. For the Y axis choose **Nodal**, click **Next**, specify the results entity as **Stress**, specify node number **25**, click **Next**.
>
> 9. Choose **Add to existing graph**, make sure to specify the correct graph from the list if there is more than one, click **Finish**. The new data will be added to the first graph.
>
> 10. Repeat steps **1**, **2** and **7**.
>
> 11. For the Y axis choose **Nodal**, click **Next**, specify the results entity as **Stress**, specify node number **60**, click **Next**.
>
> 12. Repeat step **9**.

Creating Animation Sequences

Animations are useful for checking a staged construction modelling process has been defined correctly or for 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. An animation displays a sequence of pictures showing the status of the model or results type for selected loadcases.

The structure may be animated in two ways. Both types of animation are created from the **Utilities> Animation Wizard** menu item.

❑ **Active Loadcase** (for results loadcases only) The results from a single loadcase, or eigenvector mode shape, may be animated according to a trigonometric function (sine, square or saw tooth). A full sine wave (-1 to 1) is useful for animating mode shapes obtained from an Eigen analysis while a half sine wave (0 to 1) is useful in animating a static load.

❑ **Load History** animates chosen model or results loadcases producing a animation frame for each.

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, add a contour layer to the current window prior to running the animation wizard.
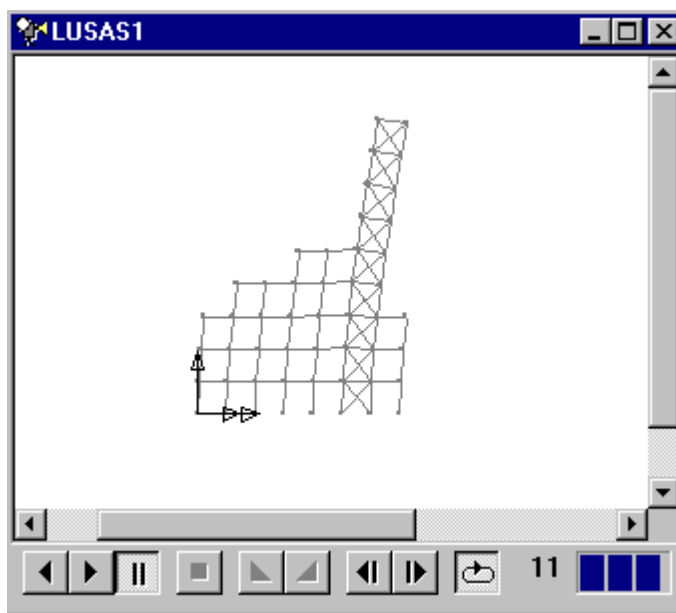
Notes

- If contours are to be included in an animation, it is useful to fix contour levels across multiple loadcases using a global or manual scale before creating the animation sequence. Setting the contour levels for the highest level of loading, will fix contour levels on each screen of the animation sequence relative to the others. The spread of stress or other entity can then be seen more readily.

- When animating deformed models it is recommend that the resize button ⊕ is switched off before creating the animation to prevent re-scaling during the animated sequence.

- Animation is carried out using pixel dumps, hence a complicated picture is no more time consuming to display than a simple picture, but may take more time to assemble initially. However, increasing the number (or size) of segments will require a proportionately larger amount of memory.

- 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.

- 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, and looped. The **1:1** button controls the aspect ratio when the window is resized.

Saving Animation Files

Animation files may be saved as Windows standard **.avi** files using the **File> Save As AVI** menu item. The **.avi** format can be viewed by double clicking on the file from within the File Explorer. When creating **.avi** files it is recommended the **"Microsoft Video 1"** compression method is used to reduce the file size. Selecting this option will produce a good reduction in file size and, perhaps more importantly,should enable trouble free playback on the majority of PCs.

Using the Animation Builder Toolbar

The animation tool builder tool bar can be found using the **View> Toolbars** menu item.

Animation sequences may be edited, or can be created frame by frame, using the animation builder toolbar.

Printing Results

Selected results values may be output to the screen in a tabular listing format for the active loadcase or for selected loadcases. Once listed the results can be printed or saved to a spreadsheet. The results for each loadcase are displayed on a separate tab in the print results window. A model info tab appears in all output windows and provides basic information about the model.

To print results to the screen his use the **Utilities> Print Results Wizard** menu item.

❑ **Active Loadcase** outputs selected results from a single loadcase.

❑ **Load History** outputs selected results for one or more loadcases.

Entity

The **Entity** chosen dictates the type of results printed. When smart combinations or envelopes are present the primary component can be set such that all other values are associated values which occur at the same time as the enveloped component. Only those components applicable for the elements used in the model will be displayed.

The entities and results types available (when applicable) are:

None

❑ **Eigenvalues** - eigenvalue, frequency and error norm.

❑ **Participation Factors -** participation factors in X, Y and Z directions.

❑ **Mass participation factors -** mass participation factors in X, Y and Z directions.

❑ **Sum of mass participation factors -** sum of mass participation factors in X, Y and Z directions. (This enables the % of active mass in each direction to be determined, as the sum of the mass participation factors in each direction should be unity).

Displacement, Residual, Reaction, Reaction Stress, Loading, Potential, Flux, Gradient

❑ **Component -** component results in tabular format.

❑ **Summary** - maximum and minimum visible values and their position on the model.

Stress/Strain

❑ **Component -** component results in tabular format.

❑ **Summary** - maximum and minimum visible values and their position on the model.

❑ **Principal** - principal values in tabular format.

❑ **Wood-Armer components** - Wood-Armer reinforcement moments and forces in tabular format.

❑ **Wood-Armer assessment** - Wood-Armer reinforcement moments and forces in tabular format.

❑ **Fatigue** or **Damage Results** - fatigue (Log Life) and damage results in tabular format.

❑ **Energy** - strain energy and plastic work results.

❑ **State variables** - extra nonlinear material parameters.

Slideline results

❑ **Component -** component results in tabular format.

❑ **Summary** - maximum and minimum visible values and their position on the model.

❑ **System forces** - forces in global or current transformed directions.

❑ **Gap forces** - forces required to reverse nodal penetration.

❑ **Contact forces** -forces generated across contact surfaces.

❑ **Contact stresses** - contact stresses computed as contact force/contact area in normal and tangential directions.

❑ **Section results** - generic contact results e.g. contact state, normal penetration etc.

For more details see **slideline results processing**.

Thermal surfaces results

❑ **Component** - underlying nodal results.

❑ **Summary** - maximum and minimum values (according to the extent specified) and their position on the model.

❑ **Flows -** flow components.

❑ **View factors** - summary of view factor sums across segments.

Transformation of results

Printed results may be **transformed** to be relative to a specified local coordinate, according to element local directions for stresses, relative to the local element material directions, or to a specified angle in the XY plane.

Location

When applicable, printed results can be obtained for the following locations:

❑ **Averaged nodal -** Average nodal (smoothed) results from visible elements. Nodal results are extrapolated from the Gauss point values within each element before averaging.

❑ **Gauss Point** - Gauss point values internal to the visible elements. The most accurate results available from the analysis.

❑ **Element Nodal -** Unaveraged nodal results for visible elements. Nodal results are extrapolated from the Gauss point values within each element.

❑ **Coordinates** When the optional **Coordinates** checkbox is enabled the global X, Y and Z positions of the nodes or gauss points (as appropriate) are included as separate columns in the printed output. If a column heading is double-clicked the results will be sorted in ascending or descending order based upon coordinates. This enables sorting of nodal and gauss results data by coordinate.

Extent

The printed results for selected loadcases are governed by the following extent options:

❑ **Elements showing results** - default option which prints results only for those elements that are displaying them in the Modeller view window.

❑ **Visible model** - prints results all elements that are visible in the Modeller view window.

❑ **Full model** - prints results for all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.

❑ **Specified group** - prints results for a specified named group. Only active if group names have been defined.

Printed results for automatically created slices

When section slices have been defined on a 3D model (with corresponding groups being automatically created in the Groups treeview) the results for just the automatically-defined groups/slices can be printed to the print results window by selecting the **Display for slice(s)** check box. Note that the automatically created group name 'Slices' is a collective name for the automatically created slice group names and does not contain results.

Results display and manipulation

The results for each selected loadcase are displayed on a separate tab in the print results window. A model info tab also appears in all output windows and provides basic information about the model. When the Printed Results window is displayed a context menu can be invoked which allows for the printed results data to be manipulated:

- **Properties** The number of significant figures or decimal places can be changed.
- **Sorting of data** Results data can be sorted in ascending or descending order. In addition, data sorting can be achieved by double-clicking on a header to sort by that column name. A second double-click on the same header will carry-out a reverse sort.
- **Saving to a spreadsheet** The contents of the current tab or all tabs can be saved to a spreadsheet or to a text file.
- **Copying to the clipboard** Selected cells or the whole grid can be copied to the clipboard.
- **Printing** The Print option sends the contents of the active tab to the printer.
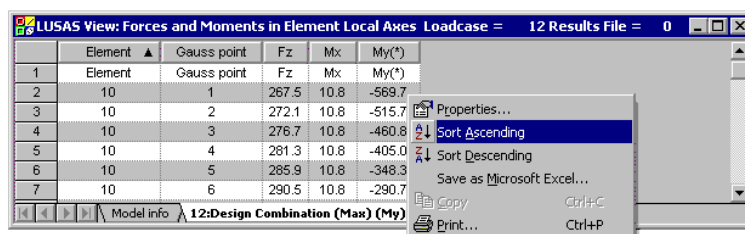
Manipulating printed loadcase results

Printing results for Envelopes or Combinations

When the active loadcase is an envelope or smart combination the results printed will show the primary component (e.g. **Fx**) marked with an asterisk. Additionally, for Envelopes only, the loadcase in which the maximum or minimum value was extracted will be tabulated in the LCID (Loadcase ID) column.

When enveloping on **All** components, the loadcase from which the results are extracted cannot be tabulated because each individual results component may have come from a different loadcase.

	A	B	C	D	E	F
1	Node	Fx(*)	Fy	Mz	LCID	IRES
2	1	38.2381	-252.024	600.284	3	0
3	2	-68.4591	-48.6257	-103.858	3	0
4	3	868.661	0.0	59.092	1	0
5	4	-2.40769	-9.54964	-11.9554	3	0
6	5	839.402	-50.9965	-11.9912	1	0
7	6	-93.4626	-53.8285	-19.4126	3	0
8	7	-38.2381	-247.976	593.811	3	0
9	8	-96.8636	-48.9366	75.7912E-15	3	0
10	9	839.402	50.9965	-42.994	1	0
11	10	84.4685	-50.9805	-18.2945	3	0

Envelope results for primary component Fx showing Loadcase ID

	A	B	C	D	E	F
1	Node	Fx	Fy	Mz	LCID	IRES
2	1	38.2381	71.6582	600.284	0	0
3	2	-68.4591	-45.7704	91.9813	0	0
4	3	868.661	0.0	59.092	0	0
5	4	-2.40769	0.236848E-12	58.6137	0	0
6	5	839.402	-50.9965	20.0294	0	0
7	6	-93.4626	-51.2462	-19.4126	0	0
8	7	-38.2381	-71.6582	593.811	0	0
9	8	-96.8636	45.7704	75.7912E-15	0	0
10	9	839.402	50.9965	19.1477	0	0
11	10	84.4685	51.2462	-10.4339	0	0

Envelope results for primary component All (No Loadcase ID can be shown)

Notes

- Selected slice and slideline data and results may be printed from the group context menu.

- To print results and include model images in a report style format see **Generating Reports**

Printing and Saving Pictures

Views of the LUSAS model in the graphics area may be printed directly to the default printer from the graphics area using the print 🖨 button.

The **File> Print Preview** menu item is useful for visualising the document prior to printing taking place. Using the **File> Print** menu item allows alternative printer settings to be used.

Note. When a model is created the default paper size (for printing use) is now set from the settings of the default printer installed on the local PC. This should help ensure that regional paper sizes are used in preference to otherwise specified sizes.

Saving pictures for use in LUSAS reports and other applications

Views of the LUSAS model can be saved as BMP, JPG, or WMF files using the **File> Picture Save** menu item. BMP and JPEG files are saved to a fixed size of 1800 pixels in width with a height proportional to the size of the graphics window when the file was saved. JPEG files are the most efficient to save in terms of file size. Windows Meta Files now contain bitmaps instead of vectors for the modelling information with correspondingly smaller file sizes. Text and annotation layer information held in a WMF file is vector-based and is therefore scalable.

Note. The contents of the graphics window can also be copied (and subsequently pasted) for use in other applications by clicking the right mouse button and selecting **Copy** from the context menu.

Saving pictures for viewing in Expose

Pictures can also be saved as **LUSAS Picture Files** for viewing only in the LUSAS picture file utility program, Expose. Note that Graphs cannot be saved in LUSAS picture file format.

The contents of the graphics area can also be transferred to the Windows clipboard using the copy 📋 button.

Generating Reports

Report templates (which are created, modified and saved in the 📋Reports Treeview for each model) hold the information required to generate reports of your model or results data. Each time a report is to be viewed, the report details that are specified in the report template are extracted from your model and results files and used to create the report.

You build a report by defining chapters that reference the model and results data that you want to include in the report. The modelling and results data you select can be restricted to particular model geometry, model attributes or loadcases, or for particular results entities, and data can be listed for all of, or just parts of your model. Additional user content such as screen captures, saved images or additional text can also be added. These user content items appear as separate chapters in the report.

Each report template can include any number of chapters that define the model attributes and analysis results to be viewed. Note that the creation of results chapters is only possible when results are loaded on top of a model. Any number of reports templates may be created and saved in the 📋 Reports Treeview of a model. The order of information in a report can be changed by dragging and dropping the chapter names up and down the Report Treeview.

 Reports are viewed in a third-party industry standard report viewer called BusinessObjects Crystal Reports (included with your LUSAS software). Report data may be exported to Microsoft Excel spreadsheets for additional calculations to be carried out as well as being exported in PDF, RTF (for use in Microsoft Word), HTML and other formats.
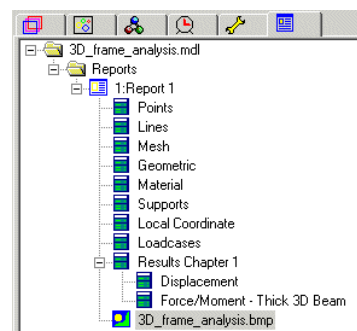
If it is desired that only results values are to be viewed on screen or be selectively printed or output to spreadsheets the **Print Results Wizard** can be used.
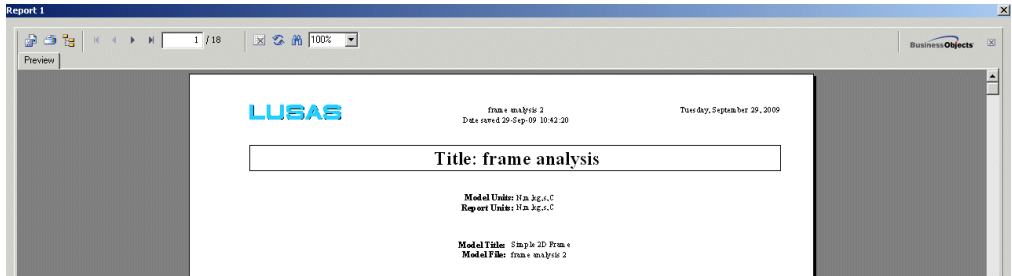
Creating a Report

A new (empty) report template is created by selecting the **Utilties> Report** menu item or from right-clicking **New Report** from the **Reports** context menu in the Report ▦ Treeview. On the Report properties dialog:

- ❏ The report **Title** is optional and is used as a title in the exported report.

- ❏ **Units** for a report are, by default, the same as those of the model. However, it is possible to prepare a report in a different system of units, in which case all values seen in the report will be converted appropriately.

- ❏ It is also possible to control the number of **significant figures** and **decimal places** seen in the report. These can also be specified independently for each chapter

- ❏ If **For all chapters** is selected the values for significant figures or decimal places chosen on this dialog will be used throughout the report. This option overrides any different values set inside each chapter.

- ❏ The **Report** name is the name added to the Report ▦ Treeview. By default reports are named Report 1, Report 2 etc if no name is specified.
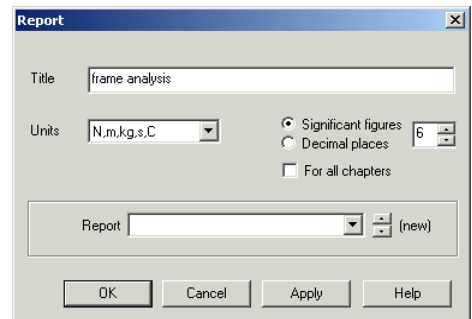
Once a New Report entry has been added to the Report ▦ Treeview, selecting the report name and using its context menu enables the adding of chapters to a report, modifying, viewing, renaming, or deleting of a report. Report templates are saved in the Report ▦ Treeview when a model is saved.

Adding a Report Chapter

Model properties, loadcase and basic combinations, envelopes and smart combination results, eigenvalue results and user information such as images and text may be added to a report in

the form of chapters. Each chapter represents one single aspect of the model. Multiple chapters may be added and re--ordered as necessary in the Report 📧 Treeview.

Chapters are added to a report by selecting the **Add Chapter** menu item from the Report name context menu (accessed by right-clicking the mouse button).

The following chapters can be added:

- ❑ **Model Properties**
- ❑ **Loadcase/Basic Combination results**
- ❑ **Envelope/Smart Combination results**
- ❑ **Eigenvalue results**
- ❑ **User Content**

Add or Edit a Model Properties Chapter
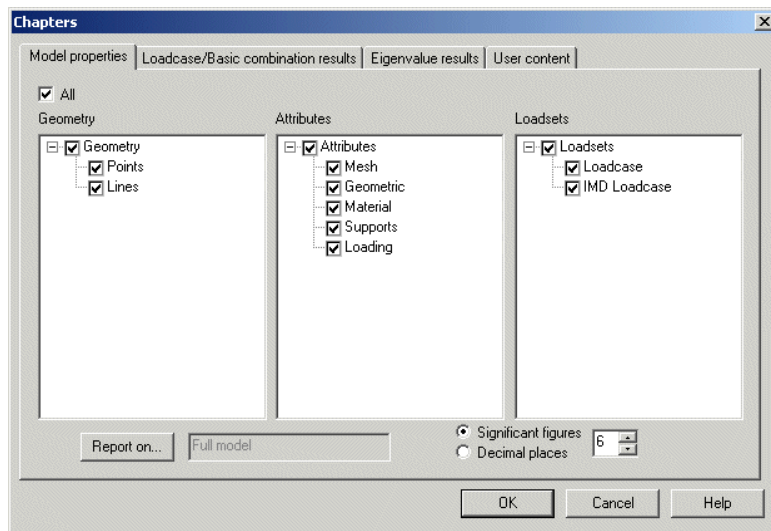
The **Model Properties** tab of the Chapters dialog allows model geometry, attributes and loadcase/IMD loadcase information to be added to the report via the use of tick boxes.

The **Report on…** button displays a dialog which controls the scope of the chapters to be created. By default, the whole model is selected, but a report could also created for just the visible model or for a specified group.

It is also possible to control the number of **significant figures** or **decimal places** for this chapter as presented in the report.
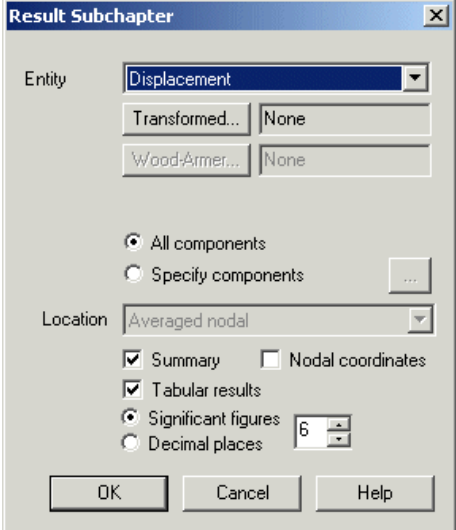
Note: It is possible to visit this dialog several times to create multiple chapters, each of which can have different ordering, scope and loadcase choices. For example you can create one chapter describing the lines in group 1 and subsequently to create a different chapter

describing the lines in group 2. Once created, the order of chapters in the report can be modified at any time by dragging and dropping them up and down the Report ▦ Treeview.

Add or Edit a Results Subchapter (for an Entity)

You can create or modify a section within a results chapter. Select the **Add...** button on the Chapters dialog to display this dialog.

❑ The **Entity** chosen dictates the type of results. The component chosen is only appropriate to Envelopes and Smart Combinations and controls the primary component – equivalent to the component chosen when the Envelope or Combination is set active.

❑ The **Transformed** and **Wood Armer** settings behave as on other dialogs.

❑ Use the **All Components** or **Specify components** and **…** buttons to control which results component values will be added to the report.

❑ The **Location** drop-down specifies whether **Averaged nodel**, **Gauss Point** or **Element Nodal** results should be calculated.

❑ The **Summary** checkbox chooses whether or not to display a summary for each results component chosen. This summary consists of the maximum and minimum values encountered, along with their location. Note that sub-reports cannot be created from Summary results information when listed in a report.

❑ The **Tabular results** checkbox chooses whether or not to display a table of numerical results. If chosen, a value will be written to the report for each component, for each loadcase, for each node or gauss point chosen. Such tables can be very large. Note that sub-reports can be created from Tabular results information when listed in a report.

❑ Additionally, **Nodal coordinates** can be added to the report. These take the form of additional columns of data in the tabular results, one each for the X, Y, and Z coordinates of each node.

❑ The number of **Significant figures** or **Decimal places** can be specified for values written to the report for this chapter.

Add or Edit a Loadcase Results Chapter
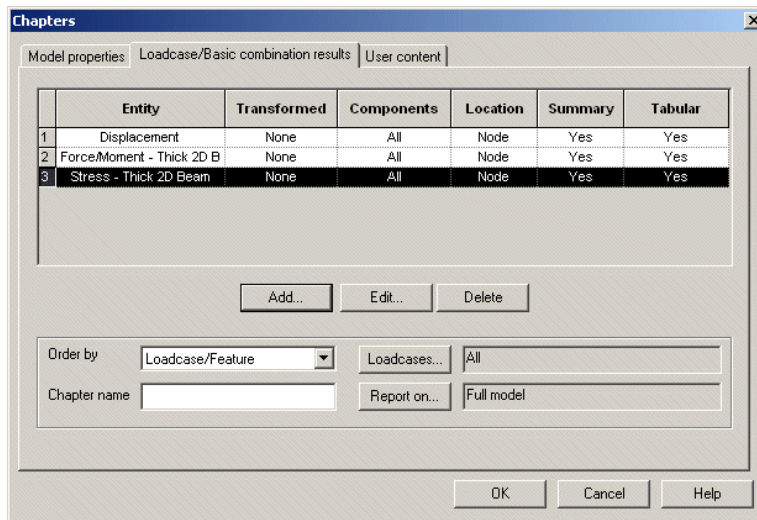
The Loadcase/Basic Combination results tab of the Chapters dialog is only shown if a results file is loaded.

The **Envelope/Smart Combination results** tab (not shown) is only added to the dialog if a model contains envelope or smart combination results.

- Use the **Add...** button to add entities for inclusion in the report as results sub-chapters. When added, results sub-chapters are shown in a grid form at the top of the dialog. By selecting the Entity name in the grid you can subsequently **Edit** or **Delete** existing content.
- Use **Order by...** to dictate the order in which the tabular results sub-headers are presented. This is similar to the concept of sorting data by column header in a spreadsheet program. The order by options are: **Loadcase / Features**, **Features / Loadcase** and **Loadcase / Mesh**. The use of the Order by facility is of particular importance when exporting results to a spreadsheet format where the use of Order by Loadcase / Mesh is recommended because of the reduced number of blank lines it creates in the output file.
- Use the **Loadcases…** button to display a dialog which restricts the chapter to display results for **All**, **Active** or **Specified** loadcases or combinations. By default, all loadcases are selected.
- Use the **Report on...** button to display a dialog which controls the scope of the chapter to be created, for example to restrict the chapter to display results for the **Full model**, the **Visible model** or a **Specified group** only. By default, the whole model is selected.
- **Chapter name** can be edited if the default or previously entered name is to be altered.

Note: It is possible to visit this dialog several times to create multiple chapters, each of which can have different ordering, scope and loadcase choices. For example you can create one chapter describing the displacements for the whole of a model and subsequently to create a different chapter describing the stress in a particular group of elements. Once created, the order of chapters in the report can be modified at any time by dragging and dropping them up and down the Report 📧 Treeview.

Add or Edit an Eigenvalue Results Chapter
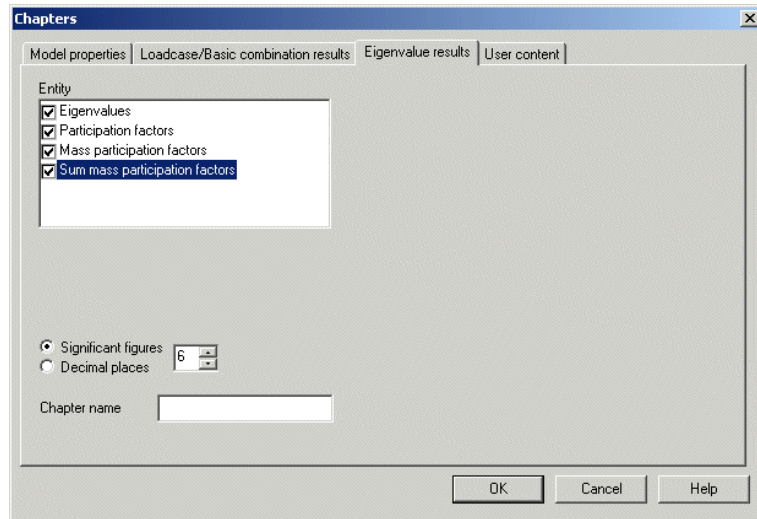
The **Eigenvalue results** tab of the Chapters dialog is only shown if a model contains eigenvalue results. The following eigenvalue results can be selected for listing:

- **Eigenvalues**

- **Participation factors**

- **Mass participation factors**

- **Sum mass participation factors**

The number of **significant figures** or **decimal places** for this chapter can be specified.

The **Chapter name** can be specified or edited if the default or previously entered name is to be changed.

Add or Edit a User Content Chapter
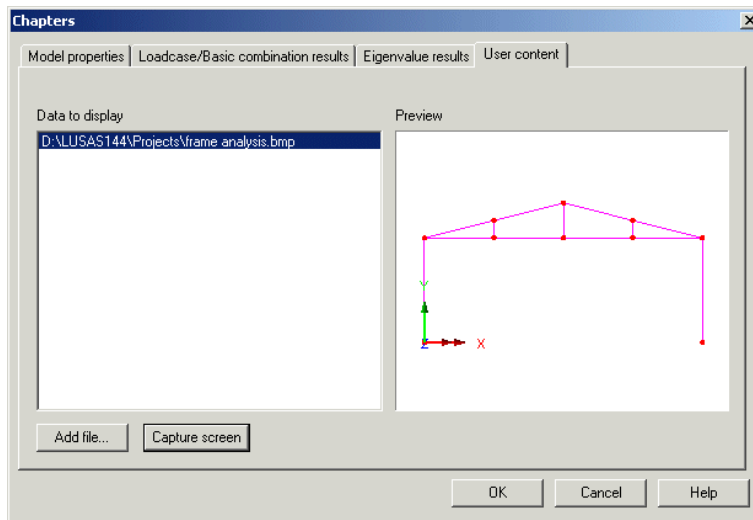
The **User Content** tab of the Chapters dialog allows images or text files to be added to the report using the **Add file...** button.

The **Capture screen** button takes a snap-shot of the current Graphics Area and allows saving it as a fixed-size BMP, JPG or WMF file to the working folder (by default) or to any other specified folder.

Clicking the **OK** button adds this image to the report as a separate chapter. Only BMP files currently appear in the preview pane. Each image added to the report is added as a separate chapter.

Note: It is possible to visit this dialog several times to create multiple chapters. Once created, the order of chapters in the report can be modified at any time by dragging and dropping them up and down the Report ▦ Treeview.

Chapter Extent

The extent of data that the chapter is to report on can be specified by selecting either:

- ❑ **Elements showing results** - prints results only for those elements that are displaying them in the Modeller view window.
- ❑ **Visible model** - prints results all elements that are visible in the Modeller view window.
- ❑ **Full model** - defaults option which prints results for all elements of the model regardless of whether the elements are visible or displaying results in the Modeller view window.
- ❑ **Specified group** - prints results for a specified named group.

For example, if the selected chapter describes Materials, by selecting a Specified group only the Material assignments used in that group will be present in that chapter.

Loadcase Selection

The loadcases, combinations or envelopes that the chapter is to report on can be specified by selecting either:

- **All**
- **Active**
- **Specified**.

If multiple loadcases are specified, multiple entries will appear in the report.

Viewing a Report

Reports are viewed from the Report Treeview by double-clicking on the report name or by choosing the **View Report...** menu item from the report name context menu. After a short delay whilst the report data is assembled and formatted, a report consisting of all the selected chapters will be displayed inside the third-party BusinessObjects Crystal Reports viewer.
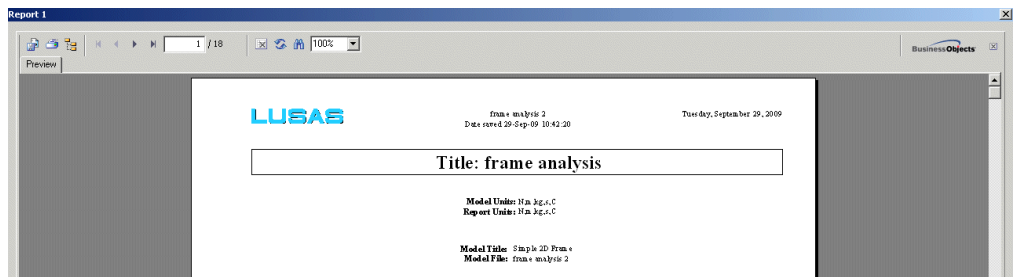
BusinessObjects Crystal Reports viewer

The BusinessObjects Crystal Reports viewer is a linked-in third-party application that is widely used in industry to present and manipulate report data. It has a toolbar that provides the following buttons / facilities for viewing, manipulating, printing and exporting the selected LUSAS model and results data:

Report Viewer Toolbar Buttons

 Export Report permits reports or sub-reports to be exported to a variety of formats to any of the following destinations:

- To a file on your disk (**Disk file**).
- To an **Application** on your computer that will open once the file has been created.
- To a folder in your mail client (Exchange).

Print Report prints the current report or sub-report view only to a specified printer.

Toggle Group Tree permits the viewing of loadcase/feature numbers in sub-reports in a treeview style format

⊲ ◂ ▸ ⊳ [1] / 18 **Page selection** These options provide the means of moving or jumping to a specific page.

⊠ **Stop Loading** stops the loading of large files

↻ **Refresh** refreshes the view contents

🔍 **Search Text** provides the means to find and jump to particular words in the report or sub-report.
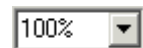
[100% ▼] adjusts the size of the page view

⊠ **Close Current View** closes the sub-report view leaving other views visible until they are closed. To go back to the LUSAS Modeller window the report viewer must be closed by using the report window's main Close button.

Notes

- When viewing a report using the Crystal Reports viewer no changes can be made in LUSAS Modeller. The only way a LUSAS model or the report listing (as held in the LUSAS Report Treeview) can be re-edited is to close the report view. This is to ensure that the report data always matches the model from which it is created.
- When a report is first loaded into the report viewer the Page Down key will not work until the view has acquired focus. Simply click anywhere in the view to set the focus.
- When saving to disk the default export directory is a temporary directory specified by the report viewing software that is used. Browse to your LUSAS project directory if you wish to save your report file with your model. The number of pages to be exported or saved can be specified.

Creating and Viewing Sub-Reports

When viewing a report, selected sections of data may be viewed in a sub-report. This can be particularly useful when wanting to export selected data to another application as, for instance, when exporting report results data to a spreadsheet. It also provides an easy way of visually printing selected pages of a main report.
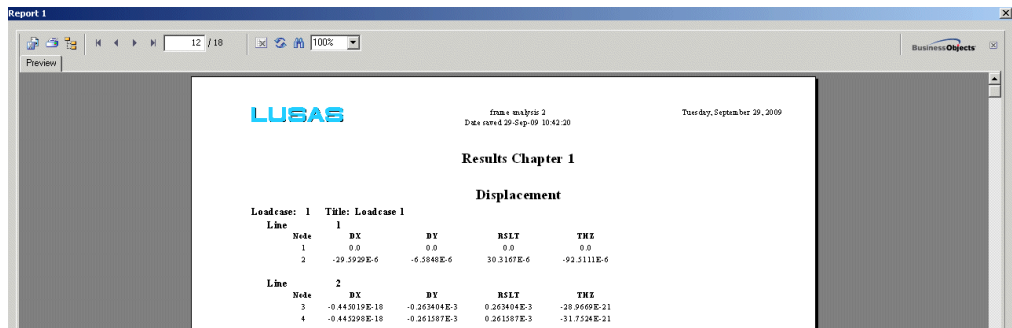
- To create a sub-report from a section of a normal report double-click on the main body of the data where you wish a sub-report to be generated, or double-click on a feature or loadcase name. A new tab will appear in the report viewer next to the Preview tab.

- Selecting the **Toggle Group Tree** button will permit the viewing of the loadcase/feature numbers in the sub-reports in a treeview style format as shown on the following image.
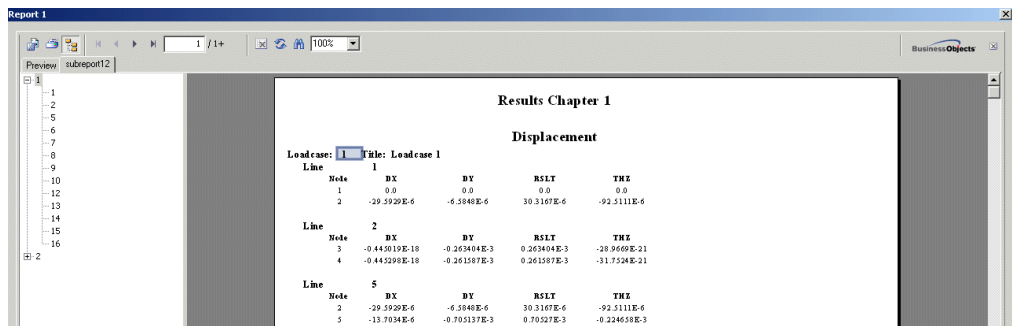
Note that it is also possible to create a sub-report from a sub-report. For example, when a sub-report containing results for a set of loadcases is being viewed, a sub-report for a particular loadcase, or for a particular feature such as a line, surface or volume, could also be created.

- To create a sub-report from a sub-report view double-click on the loadcase name (as shown in the previous image) or feature for which the sub-report should be created. A new tab will appear in the report viewer next to the previous sub-report tab as shown on the following image.

Notes

- Not all sections of a report can be selected to make a sub-report. The cursor will change to a magnifying glass when it is over data that can be selected for a sub-report.

- The toolbar buttons such as Export Report, Print Report, etc., act on the currently selected preview tab (and hence, the currently selected report view). This means that only those pages in the particular report or sub-report that is being viewed will be exported or printed.

- To delete the visible sub-report select the **Close Current View** button at the top-right of the report viewer toolbar.

Exporting Report Data

When viewing a report, report data may be exported to a variety of formats to any of the following destinations by use of the **Export Report** button

- ❑ To an **Application** on your computer that will open once the file has been created
- ❑ To a file on your disk (**Disk file**)
- ❑ To a folder in your mail client (**Exchange**)

In each case the number of pages to be opened or saved can be specified.

Note that when saving to disk the default export directory is a temporary directory specified by the report creation software that is used. Browse to your LUSAS project directory to save your report file with your model if required.

When the number of columns in a report become too large for a portrait view use Page Setup (accessed from the Report Name context menu) to either change the report page margins, or to change the report page orientation to Landscape.

Exporting Report Data to a Spreadsheet

When viewing a report, report data may be exported to a spreadsheet such as Excel by use of

the **Export Report** button

❑ Select a Format of: **Microsoft Excel 97-2000 - Data only (.XLS)**

❑ Select a Destination of: **Application**

❑ Select **Custom** format options

Note that use of the Results chapter **Order by...** option to dictate the order in which the tabular results sub-headers are presented is of particular importance when outputting results data to a spreadsheet. The option to order results data according to **Loadcase / Mesh** is recommended because of the reduced number of blank lines it creates in the output file.
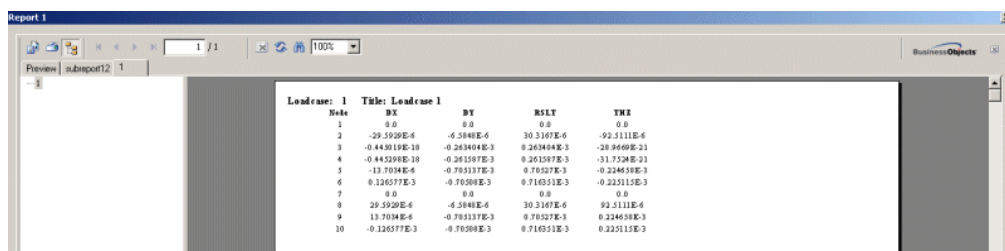
 The images that follow show an example of a sub-report created with Loadcase / Mesh order and the corresponding results exported to an MS Excel spreadsheet.

This **Order by** option should be set prior to exporting the data as follows:

1. In the Report Treeview, select the Results Chapter name and use its context menu to the select **Modify** menu item which, in turn, will display the Edit Chapter dialog.

2. Use the **Order by...** drop-down to select **Loadcase / Mesh** and click **OK**

3. In the Report Treeview, select the report name containing data to be exported and use its context menu to select **View Report**

4. After the report is displayed in the report viewer, find the Results Chapter containing data to be exported and double-click in the body of the data to create a sub-report

5. If results for a particular loadcase is to be exported, double-click on that loadcase data to create a further sub-report

6. Lastly, use the **Export Report** button to allow selection of the Format and Destination of the results data to be created.

Example output

The images that follow show an example of a sub-report created by LUSAS with Loadcase / Mesh order and the corresponding results exported to a Microsoft Excel spreadsheet.
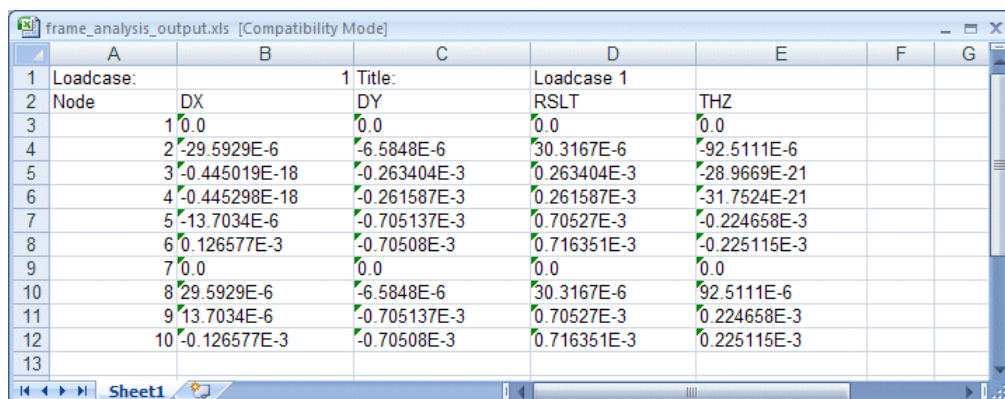
Sub-report listing ordered by Loadcase / Mesh

Sub-report data exported to a Microsoft Excel spreadsheet.

Exporting Report Data to a Word Document

When viewing a report, report data may be exported to a Word Document by use of the

Export Report button ☐ This will require you to:

❑ Select a Format of: **Microsoft Word - Editable (RTF)**

❑ Select a Destination of: **Application**

❑ Select a page range.

Note that the Results chapter **Order by...** option which dictates the order in which the tabular results sub-headers are written is particular use when exporting results data. If you do not wish to group the results per feature (e.g. per line) which is the default, then the option to order by **Loadcase / Mesh** should be used.

This **Order by...** option should be set prior to exporting the data as follows:

1. In the Report ☐ Treeview, select the Results Chapter name and use its context menu to the select **Modify** menu item which, in turn, will display the Edit Chapter dialog.

2. Use the **Order by...** drop-down to select the Order required e.g. **Loadcase / Mesh** and click **OK**

3. In the Report 📧 Treeview, select the report name containing data to be exported and use its context menu to select **View Report**

4. After the report is displayed in the report viewer, find the Results Chapter containing data to be exported and double-click in the body of the data to create a sub-report.

5. If results for a particular loadcase is to be exported, double-click on that loadcase data to create a further sub-report.

6. Lastly, use the **Export Report** button 🗗 to allow selection of the Format and Destination of the results data to be created.

Example output

The images that follow show an example of a report created by LUSAS and the same data exported to an Microsoft Word document.

Report listing in the report viewer

Report data exported to a Microsoft Word document.

Printing Report Data

When viewing a report, report data may be printed by selecting the **Print Report** button 🖨 in the BusinessObjects Crystal Reports toolbar.

Deleting Report Data

To delete the visible report or sub-report select the **Close Current View** ⊠ button at the top-right of the report viewer toolbar.

Appendix A : Smart Combination Examples

Smart Combination Examples

The following examples demonstrate how the different factors and settings can be used in smart combinations. For the purposes of these examples the results at a single node are going to be considered.

- ❑ **Case 1** - Considers a node where long term load effects are all negative.
- ❑ **Case 2** - Considers a node where short-term load effects are of mixed sign.
- ❑ **Case 3** - Considers a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four.
- ❑ **Case 4** - Considers a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four and the "Variable loadcases" set to four.
- ❑ **Case 5** - Considers a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four and the "Variable loadcases" set to one.

Smart Combination - Case 1

Consider a node where long term load effects are all negative.

In this instance the permanent and variable load factors are considered and will be added together based on the nodal result being adverse.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for positive load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Dead load	-20	1.0	0.15	1.0	-20 x 1.0 = -20
Deck surfacing	-10	1.0	0.75	1.0	-10 x 1.0 = -10
Superimposed load	-15	1.0	0.2	1.0	-15 x 1.0 = -15

Smart combination (Max) -45

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Dead load	-20	1.0	0.15	1.0 + 0.15	-20 x 1.15 = -23
Deck surfacing	-10	1.0	0.75	1.0 + 0.75	-10 x 1.75 = -17.5
Superimposed load	-15	1.0	0.2	1.0 + 0.2	-15 x 1.20 = -18

Smart combination (Min) -58.5

Smart Combination - Case 2

Consider a node where short-term load effects are of mixed sign.

In this instance the permanent and variable load factors are considered and will be added together based on the nodal result being adverse. However as the permanent effects have been set to zero, this will only combine the results that are adverse.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for positive load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Temperature	-5	0	1.3	0	-5 x 0 = 0
Wind	-5	0	1.4	0	-5 x 0 = 0
Settlement	-10	0	1.2	0	-10 x 0 = 0
Live load 1	-20	0	1.5	0	-20 x 0 = 0
Live load 2	-15	0	1.5	0	-15 x 0 = 0
Live load 3	10	0	1.5	0 + 1.5	10 x 1.5 = 15
Live load 4	-5	0	1.5	0	-5 x 0 = 0

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results
Temperature	-5	0	1.3	0 + 1.3	-5 x 1.3 = -6.5
Wind	-5	0	1.4	0 + 1.4	-5 x 1.4 = -7
Settlement	-10	0	1.2	0 + 1.2	-10 x 1.2 = -12
Live load 1	-20	0	1.5	0 + 1.5	-20 x 1.5 = -30
Live load 2	-15	0	1.5	0 + 1.5	-15 x 1.5 = -22.5
Live load 3	10	0	1.5	0	10 x 0 = 0
Live load 4	-5	0	1.5	0 + 1.5	-5 x 1.5 = -5

Smart combination (Min) = -83

Within the smart combination facility there are also two check boxes marked "Loadcases to consider" and Variable loadcases". These additional options are used for a number of bridge design codes that require the loadcases in the combination to be filtered.

Smart Combination - Case 3

Consider a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four.

In this instance the permanent and variable load factors are considered and will be added together based on the nodal result being adverse. With the number of "Loadcases to consider" set to four, only the four most positive resultants will be combined for the Max combination and the four most negative resultants will be combined for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for positive load effects. The number of load effects summed is restricted to the number of loadcases specified. The loadcases used are the most adverse, for example the most positive for max combination and all other load effects assembled are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	0.8	0.7	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used

Smart combination (Max) = 4.5

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects. The number of load effects summed is restricted to the number of loadcases specified. The loadcases used are the most adverse, for example the most negative for min combination and all other load effects assembled are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Used
Wind	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Not used
Settlement	-10	0.7	0.8	0.7 + 0.8	-10 x 1.5 = -15	Used
Live load 1	-20	0.7	0.8	0.7 + 0.8	-20 x 1.5 = -30	Used
Live load 2	-15	0.7	0.8	0.7 + 0.8	-15 x 1.5 = -22.5	Used
Live load 3	10	0.7	0.8	0.7	10 x 0.7 = 7	Not used
Live load 4	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Not used

Smart combination (Min) = -75

Smart Combination - Case 4

Consider a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four and the "Variable loadcases" set to four.

In this instance the permanent and variable load factors will only be considered for the number of loadcases specified as the number of Variable loadcases to consider. The factors will be added together based on the nodal result being adverse. The remaining loadcases are considered using the permanent factor. With the number of "loadcases to consider" set to four, only the four most positive resultants will be combined for the Max combination and the four most negative resultants will be combined for the Min combination. However by setting the "variable loadcases" to four, only positive results will be considered for the Max combination and negative results for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for the *number of positive load effects* specified by the *number of Variable loadcases* to consider. The remaining positive load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are discarded. The loadcases used are the most adverse, for example, the most positive are used for a maximum combination and all other load effects assembled are discarded. Also with the variable loadcases set to four the max combination will include only positive load effects, all negative load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	0.8	0.7	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects for number of negative load effects specified by the number of Variable loadcases to consider the remaining negative load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are also discarded. The loadcases used are the most adverse, for example the most negative for min combination and all other load effects assembled are discarded. Also with the variable loadcases set to four the min combination will include only negative load effects, all positive load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7 + 0.8	-5 x 1.5 = -7.5	Used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -.5	Not used
Settlement	-10	0.7	0.8	0.7 + 0.8	-10 x 1.5 = -15	Used
Live load 1	-20	0.7	0.8	0.7 + 0.8	-20 x 1.5 = -30	Used
Live load 2	-15	0.7	0.8	0.7 + 0.8	-15 x 1.5 = -22.5	Used
Live load 3	10	0.7	0.8	0.7	10 x 0.7 = 7	Not used
Live load 4	-5	0.7	0.8	0.7	-5 x 1.5 = -3.5	Not used

Smart combination (Min) = -75

Smart Combination - Case 5

Consider a node where short-term load effects are of mixed sign with the "Loadcases to consider" set to four and the "Variable loadcases" set to one. The permanent and variable load factors will only be considered for the number of loadcases specified as the number of Variable loadcases to consider. The factors will be added together based on the nodal result being adverse. The remaining three loadcases are considered using the permanent factor. With the number of "loadcases to consider" set to four, only the four most positive resultants will be combined for the Max combination and the four most negative resultants will be combined for the Min combination. However by setting the "variable loadcases" only positive results will be considered for the Max combination and negative results for the Min combination.

Smart combination (Max) will assemble results from the loadcases using just the permanent factors given for negative load effects and using permanent + variable factors for number of positive load effects specified by the number of Variable loadcases to consider the remaining positive load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are also the discarded. The loadcases used are the most adverse, for example the most positive for max combination and all other load effects assembled are discarded. Also with the variable loadcases set to one the max combination will include only positive load effects, all negative load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Not used
Live load 1	-20	0.7	0.8	0.7	-20 x 0.7 = -14	Not used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Not used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used

Smart combination (Max) = 15

Smart combination (Min) will assemble results from the loadcases using just the permanent factors given for positive load effects and using permanent + variable factors for negative load effects for number of negative load effects specified by the number of Variable loadcases to consider the remaining negative load effects will only use the permanent factor. The number of load effects summed is restricted to the number of loadcases specified and the other loads are discarded. The loadcases used are the most adverse, for example the most negative for min combination and all other load effects assembled are discarded. Also with the variable loadcases set to one the min combination will include only negative load effects, all positive load effects are discarded.

Loadcase	Nodal result	Permanent factor	Variable factor	Factor used for maximum combination	Factored nodal results	
Temperature	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Used
Wind	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used
Settlement	-10	0.7	0.8	0.7	-10 x 0.7 = -7	Used
Live load 1	-20	0.7	0.8	0.7 + 0.8	-20 x 1.5 = -30	Used
Live load 2	-15	0.7	0.8	0.7	-15 x 0.7 = -10.5	Used
Live load 3	10	0.7	0.8	0.7 + 0.8	10 x 1.5 = 15	Not used
Live load 4	-5	0.7	0.8	0.7	-5 x 0.7 = -3.5	Not used

Smart combination (Min) = -51

Appendix B : LUSAS Solver Trouble Shooting

LUSAS Solver Troubleshooting

During an analysis, warning and error messages may appear in the LUSAS Solver output file. An error message will terminate the solution immediately. A warning message will attempt to continue the analysis. The most common warning and error messages are:

- ❑ **Negative Jacobian** (Error)
- ❑ **Diagonal Decay** (Warning)
- ❑ **Small Pivot** (Warning)
- ❑ **Negative Pivot** (Warning)
- ❑ **Zero Pivot** (Error)

A description of these Warning and Error messages follows.

Negative Jacobian Errors

A Jacobian determinant is a measure used to give an accurate value of the current area or volume of an element. A magnitude of less than or equal to zero will automatically invoke this message and may be a result of one of the following:

- **Incorrect definition of the 2D continuum and plate elements**

  By design, LUSAS requires these element types to have an anti-clockwise node numbering sequence. The order is controlled from the underlying surface feature in which the element resides. If this message is output, the solution is to **reverse the ordering** of the surfaces for the elements having these warning messages output. Do this in LUSAS Modeller with the **Geometry> Surface> Reverse** menu item.

Note that, in LUSAS Modeller, the **local axis system of the surface may be viewed** prior to tabulation - the xy axis system displayed on each surface represents a right handed axis system, from which the anti-clockwise (or positive qz definition may be checked).

❑ **Too large a loading increment causing massive deformation of one or more elements**

This means that the elements are inverting. Note that this is only applicable for nonlinear analyses

Diagonal Decay Warnings

The stiffness matrix is a crucial component in a finite element analysis, but it can be poorly conditioned. Poor conditioning may result in round-off error, which is a loss of accuracy in the evaluation of the terms during the reduction process of the solution. This in turn leads to inaccuracies in the predicted displacements and stresses.

LUSAS monitors the round-off error by evaluating the amount of diagonal decay present during the Gaussian reduction process. This criterion is based on the assumption that initially large diagonal terms accumulate errors proportional to their size. As reduction progresses, the diagonal term is reduced, amplifying the errors until they become a maximum when the diagonal term is the pivot. An indication of probable errors may be obtained by examining the change in magnitude of the diagonal term.

The tolerance threshold above which a diagonal decay warning is output is actually quite conservative (controlled by a system variable DECAYL, default = 0.1E5). Although a check would always be recommended for any Warning of this description, significant effects are not generally expected until the decay reaches a value of 0.1E8 or greater.

Poor conditioning of the stiffness matrix occurs because of large variations in the magnitude of diagonal stiffness terms and may be due to:

❑ **Large stiff elements being connected to small less stiff elements**. An example may be where a stiff beam element is being used to transfer load into the structure. The stiffness of the beam would need to be reduced - typically, the beam would only need to be 1000 times the stiffness of the local elements.

❑ **Elements with highly disparate stiffnesses**, e.g. a beam element may have a bending stiffness that is orders of magnitude less than it's axial stiffness.

For instance, the cantilever beam problem is notoriously problematic with respect to ill-conditioning because of the potential for large differences between the axial and shear/rotational stiffness components. A typical stiffness matrix might be

$$K = \begin{bmatrix} EA/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 12EI/L^3 \end{bmatrix} \begin{matrix} (u) \\ (v) \\ (\theta) \end{matrix}$$

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 are greater than the recommended limits (see the corresponding element section in the *Element Reference Manual* for further information). An ideal value is 1:1. however, values up to 1:10 are reasonable. Depending on the results required, this value may be increased still further (a test run would be recommended first). This problem is indicated by the WARNING message: **"Unreasonably distorted element... "** The only exception are explicit dynamic elements which really do require aspect ratios of 1:1.

- Some element shapes are too distorted (see the corresponding element section in the *Element Reference Manual* for further information).

❑ **Geometric properties -** typical mistakes:

- Omission of values for any shear area parameters in the geometric properties for beams

- Omission of values for other important properties, such as the torsional constant or thickness

- Defining incompatible 1st and 2nd moment section properties for beams

❑ **Material properties** - typical mistakes:

- Different units used to define the nodal coordinates and the material properties.

- Incorrect nonlinear material parameters (yield stress and hardening values particularly)

- Inconsistent units throughout the model. This would only be of concern for dynamic analyses, where SI units are recommended.

- Incorrect definition of orthotropic properties. The inequalities given in the appropriate element section of the theory manual need to be adhered to. Numerical instabilities may result when the material characterisations approach their limits (see **Notes on material properties orthotropic** for a list of these inequalities).

❑ **Support nodes -** typical mistakes:

- The structure has not been restrained against free body translation and rotation.

Each of the above suggestions are of interest because they make a contribution to the stiffness matrix.

A further possibility is that the LUSAS Modeller model geometry is invalid because the element mesh contains gaps or has discontinuities in the connection of the elements. Such modelling problems may be found in LUSAS Modeller by:

- Using the Mesh layer to view only the outline of the mesh. The view will show lines wherever a discontinuity occurs.

- Using the Labels layer to draw the node numbers onto the mesh to see if any node numbering is overwriting at any point (indicating two nodes at the same point). Correction would normally require either a **merging** or an **equivalencing** operation.

The diagonal decay message is closely related to the small pivot WARNING message (see below). See also the additional notes in the *Theory Manual* regarding the Gaussian solution method.

Small Pivot Warnings

See the section titled **Diagonal decay warnings**.

Negative Pivot Warnings And Errors

A negative pivot could be the result of poor conditioning so make sure you have seen the section titled **Diagonal decay warnings**. However, a well conditioned stiffness matrix can produce a negative pivot if:

❑ **The system is unstable**- an unstable structure could be passing through a bifurcation or limit point, as shown in the following diagram:
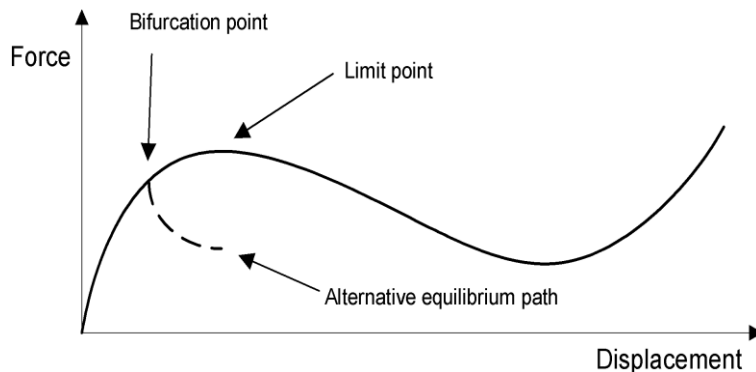
Such a bifurcation point could permit another, non-physical, solution path to be followed because, numerically, it requires less energy.

Every negative pivot warning occurring in the LUSAS output file represents a bifurcation point. A negative CSTIF value, together with a negative PIVMIN value corresponds to a limit point but a positive CSTIF and a negative PIVMIN correspond to a bifurcation point (although this is only the first one located in each case since limit points are detected by a **change** in sign of the slope of the force displacement curve). See **The nonlinear logfile**.

A negative pivots sometimes occurs during the iterative solution (indicating that the load step may be too big) but disappear when the solution has converged. If negative pivots occur and the solution will not converge then first try reducing the load step.

If the solution still does not converge, a limit or bifurcation point may have been encountered and the solution procedure may need to be changed. Running the problem under arc-length control gives the best chance of negotiating a limit or bifurcation point. A load limit point can also be overcome by using prescribed displacement loading.

❑ **The system is not adequately restrained** - for example when using a 3D beam in a 2D analysis.

❑ **Mechanism has been excited** - This is a further possibility when reduced integration is used. The use of Option 18 will normally solve this problem. If the problem persists, continue with the use of the option but refine the mesh further.

A count of the number of negative pivots is given in the LUSAS log file (parameter NSCH). Initially NSCH = 0 since, initially, a stable path is assumed. When NSCH = 1, an unstable point (limit or bifurcation) has been reached, PIVMN will give the value of the minimum pivot at this point.

Notes

- The use of LUSAS Option 62 is not recommended until all other checks have been carried out to ensure model integrity.

- Before modifying the solution procedure to arc-length, the checklist given in the section above on small pivots should be checked.

Zero Pivot Errors

LUSAS uses a Gaussian reduction solution technique to solve the finite element equations. This technique requires the structure stiffness matrix to be non-singular. This means that for static analyses the structure, or any components of the structure, must not permit any rigid body displacements or rotations. Failure to comply with this criterion will result in a zero pivot message.

The error message includes the node and variable number that may be affected by the poor conditioning - these variables should be investigated in the model. Typical mistakes can include:

- Omission of a support condition in one or more of the rigid body directions for the structure.

- Insufficient additional restraint when connecting a beam element to a continuum element. In this case a rigid body torsional spin about the axis of the beam may occur.

- Six degrees of freedom have been specified for a thick shell element, but the drilling rotation has not been correspondingly restrained.

- Insufficiently large slideline interface stiffness coefficients allowing the two bodies to pass through each other as rigid bodies. The load increment may also be too large.

- Incorrect nonlinear material parameters, such as a zero yield stress.

- Joint elements may require investigation as the stiffnesses operate in local directions and can be easily defined incorrectly - as a result, the joint stiffnesses will not be providing support in the required directions.

- There may be totally or partially unconnected elements within the structure as a result of incomplete merging or equivalencing of the model.

Other Warnings

Other warnings that may be found in the LUSAS output file include:

❑ **Aspect ratios warnings** - See the appendix on element restrictions in the *Element Reference Manual* for more information.

❑ **Excessive curvature for beams warnings** - See the appendix on element restrictions in the *Element Reference Manual* for more information.

Notes On Orthotropic Material Properties

For orthotropic material models the D matrix must be symmetric and a number of further relations must also be satisfied:

Material properties orthotropic (e.g., QPM4)

To maintain symmetry:

$$nyx = nxy * Ey/Ex$$

and to obtain a valid material:

$$nxy < (Ex/Ey)1/2$$

This applies to Fourier elements as a special case to simulate a bladed structure.

Material properties orthotropic plane strain (e.g. QPN4).

$$Ey * (nxy*Ez + nyz*nxz*Ex) = Ex * (nxy*Ez + nxz*nyz*Ey)$$

Material properties orthotropic axisymmetric (e.g. QAX4).

To maintain symmetry:

$nyx = nxy * Ey/Ex$

$nzx = nzx * Ez/Ex$

$nzy = nyz * Ez/Ey$

and to obtain a valid material:

$nxy < (Ex/Ey)^{1/2}$

$nxz < (Ex/Ez)^{1/2}$

$nyz < (Ey/Ez)^{1/2}$

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:

$nxy < (Ex/Ey)^{1/2}$

$nxz < (Ex/Ez)^{1/2}$

$nyz < (Ey/Ez)^{1/2}$

Material properties orthotropic thick (e.g. QSC4).

To maintain symmetry:

$nyx = nxy * Ey/Ex$

and to obtain a valid material:

$nxy < (Ex/Ey)^{1/2}$

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.

When An Eigenvalue Analysis Goes Wrong

It is good practice to perform a linear static analysis prior to the eigenvalue analysis. This eliminates the added complexities of the dynamic variables and will enable a check on the basic stiffness matrix for the structure. Any warning or error messages in the LUSAS output file (such as zero, negative or small pivots) should be investigated.

In the event of problems occurring after completing the linear static and the eigenvalue analysis consider some of the more common queries and their typical solutions as listed below.

Eigenvalues are missing

A Sturm sequence check is performed by default to indicate the number of modes which may be missing.

❑ **Check for Warnings or Errors** Check the output file for any other warnings or errors. There may be diagonal decay or pivot warnings that will indicate node and element numbers and help to identify any suspect areas of the mesh.

❑ **Iteration Vectors** Increase the number of starting iteration vectors. If any of the modes are close together, the default magnitude for this parameter may not be sufficient to allow accurate resolution in their extraction.

❑ **Convergence Tolerance** Tighten the convergence tolerance since, again, some modes may be close together. This would normally also require an increase in the number of iterations permitted.

❑ **Convergence Achieved** Ensure the solution converged correctly. If not, then increase the number of iterations permitted.

❑ **Mesh Refinement** Increase the mesh refinement of the model in order to increase the number of degrees of freedom in the structure to simulate all the modes expected.

❑ **Symmetry** Taking advantage of symmetry in an eigenvalue analysis may cause the inadvertent omission of several eigenvalues as a result of the corresponding symmetry supports restraining certain non-symmetric eigenmodes.

❑ **Increase Shift** If a shift has been used to eliminate rigid body motions when analysing unsupported structures, then it may be that the value used is insufficient. The solution is typically not overly sensitive to changes in this parameter and, therefore, any changes tried should be in terms of orders of magnitude.

❑ **Constraint Equations** If constraint equations have been defined in the problem the Sturm sequence check may prove unreliable. This is a limitation of the Lagrange multiplier technique used in LUSAS.

The Solution Did Not Converge

❑ **Check for Warnings or Errors** Check the output file for any other warnings or errors. There may be diagonal decay or pivot warnings that will contain node and element numbers, and help identify any suspect areas of the mesh.

❑ **Iteration Vectors** Increase the number of starting iteration vectors. If any of the modes are close together, the default magnitude for this parameter may not be sufficient to allow accurate resolution in their extraction. Increasing this parameter is also essential if only requesting a small number of eigenvalues (1-2). Ten iteration vectors would be a reasonable starting value for such a situation. This parameter is not used for Lanczos extraction.

❑ **Convergence Tolerance** The convergence tolerance criteria may be too tight - try slackening this criteria. This would normally also require an increase in the number of iterations permitted.

❑ **Increase Shift** If a shift has been used to eliminate rigid body problems when analysing unsupported structures, then it may be that the value used is not sufficient. The solution is typically not overly sensitive to changes in this parameter and, therefore, any changes tried may be in terms of orders of magnitude.

Negative Eigenvalues Are Calculated

❑ **Check for Warnings or Errors** Check the output file for any other warnings or errors. There may be diagonal decay or pivot warnings that will indicate node and element numbers and help to identify any suspect areas of the mesh.

❑ **Iteration Vectors** Increase the number of starting iteration vectors. If any of the modes are close together the default magnitude for this parameter may not be sufficient to allow accurate resolution in their extraction. Increasing this parameter is also essential if only requesting a small number of eigenvalues (1-2). Ten iteration vectors would be a reasonable starting value for such a situation.

❑ **Convergence Achieved** Ensure that the solution converged correctly. If not, then increase the number of iterations permitted.

❑ **Convergence Tolerance** Tighten the convergence tolerance, since some modes may be close together and require greater numerical resolution. This may also require an increase in the number of iterations permitted.

❑ **Reduce Load Level** Reduce the load applied to ensure that it is below the lowest expected buckling mode of the structure. A negative eigenvalue in a buckling analysis could also simply mean that the applied loading is in the opposite direction to that which would cause buckling; e.g. a strut subjected to tensile load instead of compression.

❑ **Alternative Buckling** QSL8 elements have given negative eigenvalues for thin structures and resulted in negative projected mass errors. The remedy is to use the alternative buckling algorithm where positive eigenvalues should be obtained. This is always the case except when the buckling load factor is less than unity. Adjust the load level to ensure that all the load factors are greater than 1 if this occurs. Additionally, use Option 18 (fine integration rule for the element) to overcome the excitation of any element mechanisms.

Why Loading is Ignored in an Eigenvalue Analysis

In a standard eigenvalue analysis, the loading will be ignored completely. For a linear analysis, once the eigen-pairs have been obtained, the stress distribution s for each mode shape F is evaluated using:

s = D * B * F

where D and B are the elastic constitutive and strain-displacement matrices respectively.

The reason that the forces have no effect on the vibrational behaviour is because the loading conditions in a linear analysis do not affect the D or B matrices. To include their effects, a nonlinear D and B matrix must be evaluated prior to the eigen analysis. This is achieved by performing a static nonlinear analysis (with a geometrically nonlinear option) followed by an eigenvalue analysis. In this way the effects of the loads are included via the updated B matrix (and D matrix if a nonlinear material has been specified).

Appendix C : Keyboard Shortcuts

Keyboard Shortcuts

Selecting Model Features

Features displayed in the graphics window may be selected using either specific cursors or by using normal cursor mode in conjunction with specific keys:

Feature / Mesh Object Selection Options

Hold the key shown down when using the left mouse button.

Specific cursor	**= Normal cursor + key**
All geometry selection	= + **G** key
Point selection	= + **P** key
Line selection	= + **L** key
Surface selection	= + **S** key
Volume selection	= + **V** key
Mesh selection	= + **M** key
Node selection	= + **N** key
Edge selection	= + **B** key

[icon] Face selection = [icon] + **F** key

[icon] Element selection = [icon] + **E** key

[icon] Annotation selection = [icon] + **A** key

Note. Key shortcuts can be used to override specific cursor selections.

Area Selection Options

Rectangular, circular, or polygonal areas can be selected by using specific area toolbar buttons or by using normal cursor mode with a specific key:

Specific cursor	= Normal cursor + key
[icon] Click and drag the cursor to the opposite diagonal corner.	= [icon]
[icon] Click the centre of the circle and drag the cursor to the required radius.	= [icon] + **C** key
[icon] Click each corner of a polygon and either double click to close the polygon or select Close Polygon from the context menu.	= [icon] + **X** key

Memory shortcuts

After features have been selected:

Ctrl + **M** key = Set selected items into Selection Memory

Selection Modifiers for All Cursors

Features displayed in the graphics window may be added to, or removed from, any initial selection using these selection modifiers:

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

Browsing Selected Features

Items in the current selection may be viewed in the Browse Selection window which can be displayed from the **View> Browse Selection** menu item. This window can also be triggered by a right mouse button click in the **Selected** area of the status bar at the bottom of the graphics area, or by right-clicking in a graphics window. Selected features can be deselected and reselected as necessary from those listed.

Model Viewing Shortcuts

The model can be rotated, zoomed, panned, and viewed at predefined orthogonal and non-orthogonal views using specific cursors or view buttons, or by using normal cursor mode in conjunction with specific keys. Rotation, zoom and pan can also be carried out in other cursor input modes such as when defining lines by cursor or section slicing for example.

Dynamic Pan (Drag)

Specific cursor	**= Normal cursor + key**
Hold down the left mouse button to pan the model.	= + **D** key
	= + **Middle** mouse button

Note. Hold down the key(s) to restrain the pan for either specific or normal cursor mode about the axis stated:

X or **Shift** key = Restrain in the screen X axis

Y or **Ctrl** key = Restrain in the screen Y axis

Dynamic Rotation

Specific cursor	**= Normal cursor + key**
Rotates the model around various multiple axes.	= + **D** key
	= + **Middle** + **Left** or **Right** mouse button

Note. The model is rotated about its centre unless any part of the model is selected in which case the model is rotated about the centre of the selection. Hold down the key(s) to restrain rotation about the axis stated:

X or **Shift** key = Restrain in the screen X axis

Y or **Ctrl** + **Shift** keys = Restrain in the screen Y axis

Z or **Ctrl** key = Restrain in the screen Z axis

Dynamic Zoom

Specific cursor	= Normal cursor + key

 Hold down the left mouse button and move the mouse.

= + **Scroll mouse wheel**

= + **Z** key

Note. If any part of the model is selected it is used as the centre of the zoom.
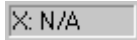
Zoom

 Drag a box around the region to be enlarged or click the left mouse button to zoom in progressively with each click.

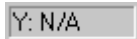**Ctrl** key = Zoom out (when held at same time)
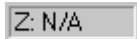
Note. The cursor position dictates the centre of the zoom.

Orthogonal Model Views

These buttons are located in the status bar.

 = View along the +X axis towards the origin

 = View along the +Y axis towards the origin

 = View along the +Z axis towards the origin

Any of the views along these axes can be modified by using these key sequences which select an alternative view orientation:

Shift key + = View along the -X axis towards the origin

Ctrl key + = View along the -Y axis towards the origin

Ctrl + **Shift** key + = View along the -Z axis towards the origin

Equivalent toolbar buttons for these view shortcuts can be found on the customisable toolbar dialog or by clicking the right mouse button on the orthogonal model view buttons.

Non-Orthogonal Model Views

 = Isometric view
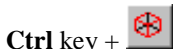
Ctrl key + = Reverse isometric view

 = Dimetric view
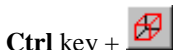
Ctrl key + = Reverse dimetric view

 = Trimetric view

Ctrl key + = Reverse trimetric view

Useful Windows Shortcuts

The following standard Windows shortcuts are useful when creating and printing models in LUSAS Modeller:

Ctrl key + **N** key = New

Ctrl key + **O** key = Open

Ctrl key + **S** key = Save

Ctrl key + **P** key = Print

Ctrl key + **A** key = Select all items

Ctrl key + **C** key = Copy

Ctrl key + **X** key = Cut (for text only)

Ctrl key + **V** key = Paste

Ctrl key + **Z** key = Undo

F2 key = Rename (when name is selected)

F5 key = Redraw

Appendix D : Tip of the Day

Tip of the Day

When starting LUSAS Modeller useful tips can be optionally displayed. This is done by selecting the **Help > Tip of the Day...** menu item and ensuring that **Show tips at Startup** is selected. Next and Previous tips can be browsed.

The following is the list of tips supplied in the current release version:

- If you haven't already done so, please read the 'Getting Started' leaflet.
- The 'Keyboard Shortcut Guide' will help you use LUSAS in an efficient manner
- If you lose unsaved edits because of a power, hardware or software failure, restart LUSAS, open the model again and chose 'yes' when prompted to recover.
- Saving your work frequently prevents data loss and makes undo/redo work faster.
- Help is available even for greyed out buttons and menus. First click on the Help button on the main toolbar, then click on any menu item or toolbar button.
- Right-clicking in a window displays a context menu relevant to both that window and any selection within it.
- Toolbar buttons with a small triangle to the right hand side are menu buttons. Press the left hand side to use the top button, or the right hand side to display other buttons.
- Holding down the Shift key while selecting will add to the current selection.
- Holding down the Ctrl key while selecting will toggle an object's selection state.
- Holding down the Alt key while selecting will additionally select items that intersect (cross) the selection perimeter.
- Ctrl-A selects everything visible in the current window.
- You may filter your selection to include only Volumes, Surfaces, Lines, Points, Elements, Nodes or Annotation by respectively holding down the V, S, L, P, E, N or A key while selecting.
- Alt-Enter displays the properties of the current selection.
- If you the hover the cursor over an object, a data tip will appear providing details of that object.
- Pressing Enter whilst a data tip is showing will select the object described.

- To Zoom, Drag or Rotate the model hold the Z, D or R key down whilst moving the mouse.
- Using the dynamic rotation while holding down the Ctrl key rotates the model in the plane of the screen.
- Using the dynamic rotation while holding down the Shift key rotates the model about the screen X axis.
- Using the dynamic rotation while holding down the Ctrl & Shift keys rotates the model about the screen Y axis.
- Using the drag tool while holding down the Ctrl key drags the model up and down.
- Using the drag tool while holding down the Shift key drags the model left and right.
- If you have a 3 button wheel mouse, the wheel will zoom, holding the middle button will drag and holding the middle button and either of the other two buttons will rotate.
- To change the rotation increment, visit the 'View' tab on the Window Properties dialog (right click in the current view)
- To change the style of the XYZ axes arrows, visit the 'View Axes' tab on the Window Properties dialog (right click in the current view)
- To change the selection or background colours, visit the 'General' tab on the Window Properties dialog (right click in the current view)
- Most context menu functionality for attributes is also available for groups of attributes. This helps you check your model.
- Dragging an attribute from the treeview and dropping it on a window assigns it to anything selected in that window.
- Dragging and dropping layers in the layers treeview changes the order in which they are drawn. This is useful if, for example, your solid model is eclipsing your labels.
- Dragging and dropping loadcases in the loadcase treeview changes the order in which they are analysed.
- Dragging and dropping controls, attributes, or groups of attributes, in the loadcase treeview assigns them to a different loadcase.
- The 'Selected' box in the status bar shows the number of objects are currently selected.
- Pressing 'Tab' or clicking in the 'Selected' box in the status bar cycles through all the objects which could have been selected at the last mouse click.
- Right-clicking in the 'Selected' box in the status bar allows access to a menu of selected objects. Choose 'Previous' when you have cycled the selection once too many!
- The 'browse selection' window gives useful feedback on which items are currently selected.
- The 'browse cyclable items' window shows all items which could possibly be selected at the chosen position.
- To select an item with a known name, use the 'Advanced Selection' dialog.
- To show the location of a named item, select it using 'Advanced Selection', right click on it in the 'Browse Selection' window, and chose 'find' from the menu.
- If a message in the text output window refers to a particular object by name, double click on the message for help finding that object.

- Clicking in the 'Z' box in the status bar views the model from the positive end of the Z axis - and similarly with X and Y. (Hold down Ctrl while clicking to view from the reverse side).
- Clicking the 'advanced' button on the LPI command bar lets you define macros for frequently used commands
- The members of a group can be examined from the group's property page
- The 'jump to' button on the 'hierarchy' tab of an object's property page can be used to select and show the properties of connected objects.
- The 'assigned in active loadcase' option on an object's property page allows you to view either assignments in the active loadcase, or in any loadcase.
- Treeviews can be moved to different tree frames by drag and drop. This is useful if you prefer to see more than one treeview at the same time.
- Attributes can be assigned to the contents of groups by copying the attribute and pasting it onto a group in the treeview.
- By saving a view using the Window menu, and naming it 'default', every new window will use the saved view.
- Pressing the Ctrl and Break keys together will interrupt the current process.
- For larger models save time by setting the manual redraw option (Window menu) and manually redraw (F5 key) at any time.
- For larger models save time by setting the manual resize option (Window menu) and manually resize at any time.
- For larger models save time by locking the mesh whilst making several geometry or mesh changes. Either manually remesh, or unlock once finished to reinstate automatic meshing
- It is usually much quicker to undo several events all at once than to undo them individually
- Many additional toolbar buttons are available via the View>Toolbars>Customise menu item
- Selection is possible using several different criteria (e.g. connectivity, element type) - see the 'Advanced Selection' dialog
- To retain a record of the commands used in a session, use the File>Script>Start Recording menu item.
- You may use the Esc key to close any dialog.
- Holding down the Shift and Control keys while selecting will remove from the current selection. For example, the current selection could be trimmed from a perpendicular view orientation to achieve a 3D selection.
- A selection may be filtered to include only Geometry, Volumes, Surfaces, Lines, Points, Mesh, Elements, Nodes or Annotation by respectively holding down the G, V, S, L, P, M, E, N or A key while selecting.
- Element faces may be selected by holding down the F key while selecting (LUSAS HPM users only)
- LUSAS Solver can be paused by pressing the Pause key on your keyboard to temporarily free up PC resources if required. Press the Esc key to resume Solver.

- Errors and warnings reported in the Text Output window can be double-clicked to open the Identify Object facility which can be used to locate the referenced items.

Editing and adding your own tips

You can add your own tips by editing the text file named **tips.txt** which is held in the <LUSAS installation folder>\Programs\Config directory.

Appendix E : Real Numbers and Expressions in LUSAS

Input and Output of Real Numbers in LUSAS

The precision of user-entered real numbers is preserved both internally within LUSAS Modeller, and when re-displayed on dialogs.

All real numbers are displayed in engineering-style notation (that is in 3,6,9 etc, powers of 10; as in 89.6E3) rather than 8.96E4 or 0.896E5. This applies to all text entry fields, including the grids used in many places throughout the Modeller user interface.

For cosmetic numbers that are system generated, like the current zoom factor, only a sensible number of significant figures is shown. This does not alter the precision stored, which will always be the maximum allowed by the operating system.

Expressions and Functions Supported

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

The following functions are also supported:

Arithmetic

```
(A), -A, A+B, A-B, A*B, A/B, A^B, ceil(A), floor(A), abs(A),
max(A, B), min(A, B), pow(val, exp), mod(val, div)
```

Trigonometric

```
sin, cos, tan, asin, acos, atan, sinh, cosh, tanh, sind,
cosd, tand, asind, acosd, atand, atan2, atan2d
```

Mathematical

```
exp, log, log10, sqrt
```

Logical

```
A > B, A < B, A = B, not(A), and(A, B), or(A, B), boolEq(A,
B), boolNE(A, B), gt(A, B), ge(A, B), lt(A, B), le(A, B),
eq(A, B), ne(A, B), if (condition, then, else)
```

Other

```
Radians(angle) Converts an angle entered in degrees into an
angle in radians
```

```
Degrees(angle) Converts an angle entered in radians into an
angle in degrees
```

Glossary

abscissa

The x axis of a graph. *See also* **ordinate**, **graph dataset**.

acceleration

Second derivative of displacement with respect to time. *See also* **velocity**.

acronym

a word formed by the initial letters of words or by initial letters plus parts of several words. For example, FORTRAN is an acronym for FORmula TRANslator.

access

The process of seeking, reading, or writing data on a storage unit.

active load case

Each graphics window used in a LUSAS Modeller has an active load case set. The active load case is marked with a dot in the Load case Treeview. To set a new active load case, click on another load case, right-click and choose Set Active from the shortcut menu.

- In a model file (.mdl) attributes that have been assigned to load cases, such as loading, may only be visualised for the active load case.

- In a results file (.mys) results are displayed for the active load case.

adaptive analysis

A series of analyses in which subsequent meshes are refined to reduce the overall errors in the solution.

analytical surface

A Surface that can be represented by an analytical expression, thus defining its internal geometry. LUSAS supports cylindrical, conical and spherical analytical Surfaces. *See also* **ruled surface**, **regular surface**, **irregular surface**.

angular acceleration

A measure of the rate of change of rotational speed expressed in radians per second per second.

angular velocity

A measure of rotational speed expressed in radians per second.

animation

Screens of information displayed in rapid succession to give the impression of movement.

anisotropic

Material allowing different material properties to be specified in arbitrary (non-orthotropic) directions by direct specification of the modulus matrix. *See also* **orthotropic**.

annotation

Information added to the screen to clarify a plot. This may take the form of text, lines, symbols, results summaries etc.

application

A computer program designed to meet specific user needs. This term is interchangeable with **program**.

apply

See **apply button**.

apply button

This push button will carry out the actions signified by the chosen form subject to the parameters entered, then re-display the form allowing you to enter further data.

arc-length method

A generalised form of displacement control where the load level does not remain constant during each load increment so that convergence near limit points may be achieved. A benefit of the arc-length method is that it can stabilise the iterative process. *See also* **constant load level**, **line search**. *Also known as* arc-length control.

arrow keys

On your computer keyboard, the keys you use to navigate around your screen. Each key is marked with an arrow and is named for the direction in which the arrow points. There is an UP ARROW, DOWN ARROW, LEFT ARROW and RIGHT ARROW key. *Also known as* direction keys or cursor keys.

ASCII

Acronym for American Standard Code for Information Interchange. ASCII is a standardised 8-bit code used by most computers for interfacing.

aspect ratio

The ratio of longest side to shortest side of a finite element. It is important to take the aspect ratio into account as the quality of results is affected when it becomes too large.

assign

To match up a feature with a previously defined attribute, you assign the attribute to the feature.

association

See **associativity**.

associative plasticity

Plasticity formulation where the direction of plastic straining is normal to the yield surface. This is a common finite element implementation for non granular materials.

associativity

All features in LUSAS are defined in terms of lower order features, that is they are associative. When a point is moved all lines, surfaces and volumes are updated to reflect the new position.

assumed strain

A strain field used in the formulation of the general shell element to avoid shear locking' when it is used as a thin shell.

attribute

See **attribute dataset**.

attribute dataset

Describes the properties of features to which it is assigned, for example materials, mesh, loading, etc.

automatic incrementation

Analysis scheme where successive levels of applied load are applied by specifying multiplying factors for a given load case. Successive load levels can be altered based on the previous convergence history.

automatic masters

A facility within a LUSAS Guyan reduction in which LUSAS automatically selects the master freedoms.

averaged nodal results

Most results are calculated at the Gauss points of an element and then extrapolated to the nodes using the element shape functions. When the results from all elements joined at a node are added together and divided by the number of contributing elements these are known as averaged nodal results. *See also* **unaveraged nodal results**, **Gauss point results**, **shape functions**.

axial force

The longitudinal force along the axis of a beam. *See also* **force diagram**.

axisymmetry

A structure that has circular symmetry of geometry, material, supports and loading, such that a radian slice can be analysed.

b matrix

Matrix of element shape function derivative terms describing the relationship between strain and displacement. It is used in the formation of the element stiffness matrix. *See also* **stiffness matrix**, **d ~matrix**.

background grid

A collection of triangular or tetrahedral shapes which are used to specify the element edge length when meshing surfaces automatically.

backward Euler

An implicit plasticity formulation in which the plastic strain normality condition is formulated at the trial stress position. *See also* **forward Euler**.

bandwidth

See **frontwidth**.

bar element

A straight or curved isoparametric line element without any rotational degrees of freedom used to model strut or truss structures.

batch file

A text file that contains MS-DOS commands. When you run the batch program, MS-DOS carries out the commands in the file as if you had typed them at the MS-DOS prompt. Used to set up an environment and run a program. A batch file has the extension bat. *See also* **script file**. *Also known as* batch program.

beam element

A straight or curved line element with rotational degrees of freedom used to model frame structures. *See also* **explicitly integrated beam**, **numerically integrated beam**

bending moment diagram

A graphical representation of the bending moment results from a beam element drawn along the element. Maximum or minimum and end values are accessible. *See also* **force diagram**.

beta angle

The angle between the element axes and the line axes.

biaxial

A description of a material model in which two axes are considered to determine the material response. For instance, the biaxial concrete model uses the two principal stress directions. *See also* **uniaxial**, **triaxial**.

bicubic patch

A surface formed of cubic splines in which the internal geometry is defined as well as the boundary.

bifurcation point

A point at which an alternative load-displacement path can be taken. *See also* **limit point**.

binary file

A random access file containing information that is in machine-readable form; it can be read only by an application.

body force loading

See **constant body force**.

boot

To start or restart your computer, loading the operating system from your hard disk or floppy disk. *See also* **hard disk**, **floppy disk** .

boundary conditions

Loading, constraints and supports applied to a finite element model. Boundary conditions are sometimes termed nonlinear where the model changes in contact, such as with a lift-off support or a contact analysis. *See also* **constraint equation**, **supports**.

bracketing

The accurate computation of a critical point which may be either a limit point or a **bifurcation point**.

branching

The tracing of an alternative solution path from a **bifurcation point**.

buckling

See **eigenvalue buckling**.

buffer

A temporary area in which data is stored.

Cartesian axis set

A standard set of axes based on the global XYZ axis set but not necessarily in the same position or orientation. *See also* **local coordinate system**, **cylindrical axis set**.

cascade

The action of bringing down a secondary menu by selecting an entry on an upper menu level. Usually designated by a right facing arrow.

central processor (CPU)

The heart of the computer system. The CPU interprets and executes programs, routes information, controls input and output and temporarily stores data.

checking

See **data check**.

checklist

A list of items through which to work to establish the integrity of a model or its results.

classical yield

Plasticity formulation where yielding is independent of the sign (tension/compression) of the stress causing it (e.g. von Mises, Hill).

close contact detection

A facility that automatically inserts a small spring element between opposing surfaces of a slide pair when they are within a specified tolerance of any section of the opposing surface. The spring controls possible mechanisms in the initial response. *See also* **contact**.

click

To press and release a mouse button quickly.

co-rotational

Form of geometric nonlinearity in which large displacement effects are related to a set of axes that follow and rotate with the element.

coarse integration

A method of element integration using an alternate selection of a smaller number of Gauss points. *See also* **fine integration**.

cohesion

A material property of granular materials, such as soils or rocks, describing the degree of granular bond and a measure of the shear strength.

colour fill

Solid block colours used to represent stress patterns or different mesh views.

combination

See **load combination**.

combined line

A LUSAS feature made up from a combination of individual connected line datasets. It can be used to simplify meshing.

command bar

Part of the LUSAS interface where commands may be typed. Displayed from the View menu. *Also known as* command line.

command file

Commonly used sequences of commands may be grouped in a file and replayed at any time. *See also* **sub-session file**.

command line

See **command bar**.

comments

Notes attached to a data or command file for informational purposes only.

compatible

A description of data, programs or equipment that can be used between different kinds of computers.

compiler

A computer program that translates a program written in a programming language into a set of instructions in the language of the computer.

component

See **group**.

compound load

a collection or set of defined discrete loads that are assigned to a model as a single loading attribute

composite properties

Define how individual materials are layered in a composite material.

concatenation

Joining two or more strings together to form a single string.

conduction

The process of energy exchange by kinetic motion or direct impact of molecules (fluids) or the drift of electrons (metals). *See also* **convection**, **radiation**.

concrete model

A plane stress material model that mimics the nonlinear cracking of concrete. *See also* **orthogonal cracking**, **non-orthogonal cracking**, **cracking**.

conjugate gradient solver

A solver that iterates towards a solution, judging convergence by a user-defined tolerance. This is in contrast to a direct solver, such as the frontal solver. Because the solution is inexact, very sensitive to the matrix conditioning and repeated for every load case, it is best suited to large, solid, well-conditioned, single load case analyses. *See also* **pre-conditioned conjugate gradient solver**.

constant body force

A load type that applies a force loading per unit volume. This load type can also be used to apply acceleration loading. *Also known as* **body force loading**.

consistent mass

Distributed between the degrees of freedom. The mass matrix is not diagonal. *See also* **lumped mass**.

constant load level

When the load is applied in fixed increments and the chosen iterative algorithm is utilised to obtain convergence at each load level. *See also* **line search**, **arc-length method**.

constitutive relationship

The empirical relationship between stress and strain. May include thermal effects and be nonlinear with irreversible plastic deformation.

constraint equation

An additional relationship between different freedoms of a structure. *Also known as* multi-point constraints.

contact

The action of bringing one portion of a finite element mesh into a position where it impacts on another part of the mesh. *See also* **impact**.

contact node

Any node assigned for use on a slideline surface. *Also known as* a slideline node.

contact zone

The area of a model that is considered to be touching another part. *See also* **zonal detection**.

contacted node

A slideline node currently considered to be in contact with the opposing slideline surface.

context-sensitive help

Available by clicking on the Help button on the main toolbar, or by pressing Shift F1 on the keyboard. This puts the cursor into help mode, next click on any toolbar button, menu entry or other part of the LUSAS interface for context-sensitive help. A*lso know as* pop-up help. *See also* **help**.

continuum element

Finite element formulation based on classical continuum mechanics principles. The output from these elements is in terms of continuum stresses rather than stress resultants.

contour plot

A graphical representation of a results pattern across the surface of the model. *See also* **colour fill**.

control dataset

A dataset containing LUSAS analysis control information. The parameters stored in these datasets control the progress of the analysis. A control dataset is assigned to a load ID. *See also* **assign**.

convection

The process of transmission of heat through liquids or gases at a body surface by means of the motion of the fluid. *See also* **conduction**, **radiation**.

convergence

When using incremental iterative solution algorithms, a measure of the convergence of the solution is required to define whether equilibrium has been achieved. *See also* **increment**, **Iteration**, **displacement norm**, **residual norm**, **work norm**.

coprocessor

A microprocessor device connected to the central processor that performs specialised floating point computations more efficiently than the CPU alone. *See also* **central processor**.

coupled analysis

The interaction between a thermal and structural analysis may be studied by performing a coupled analysis. *Also known as* **thermo-mechanical coupling**. *See also* **fully coupled**, **semi-coupled**.

copy

To create a new feature by copying an existing one using a transformation dataset. *See also* **move**, **transformation dataset**.

CPU time

A measure of actual processing time, usually expressed in seconds. *See also* **elapsed time**.

CQC method

A method used in spectral response analysis which gives a combination formula that includes all cross modal terms. Stands for complete quadratic combination. *See also* **SRSS method**, **seismic analysis**.

crack tip elements

A non-standard continuum element family, in which the mid-side nodes on 2 of the element edges are moved to the quarter point. This enables an accurate resolution of stresses to be obtained at a sharp notch or crack.

cracking

Material nonlinearity resulting from material failure. Most directly associated with the concrete (ductile fracture) material model in LUSAS.

creep

The time-dependent inelastic deformation of a material which causes a permanent change in the material once the loading is removed. Creep can occur as soon as the body is stressed and is independent of a yield criterion.

critical damping

The damping value for which the impulse response is just oscillatory.

critical point

Either a limit point or a **bifurcation point**.

critical time step

The time step in an explicit dynamic analysis that satisfies the conditional stability criteria.

critically damped system

The dividing line between under-damped and over-damped systems where the equation of motion has a damping value that is equal to the critical damping.

current group

The group which will contain all newly defined geometry features.

current stiffness parameter

A measure of the current, overall structural stiffness. Originally unity, a lower value indicates a general softening whilst a higher value indicates a stiffening of the structure.

cursor

The device used to pick items from the graphics area and select menu entries. The hardware device used to control the position of the cursor is normally a mouse. *See also* **cursor selection**.

cursor picking

See **cursor selection**.

cursor keys

See **direction keys**.

cursor selection

To select an item with the cursor. *Also known as* cursor picking.

curve

See **load curve**.

cycle

Feature definitions can be rotated, or cycled, by one or more steps. For example, a surface may have the definition order of the lines cycled to change the local element axes. *See also* **reverse**.

cylindrical axis set

An axis set based on a cylindrical form. *See also* **Cartesian axis set**, **local coordinate system**.

d matrix

Matrix of material constants which describe the stress/strain or constitutive relationship for a material. *See also* **constitutive relationship**.

damage

The damage done to a structure is used as a measure of how many loading cycles the structure will withstand before failure. Failure occurs when the damage reaches unity. *See also* **fatigue**, **log life**.

damage material model

A nonlinear material model in which the value of Young's modulus degrades with progressive loading.

damage threshold

The initial stress limit below which no material damage is assumed to occur when using the nonlinear damage material model. It is the linear elastic limit point.

damping

Any mechanism that dissipates energy. *See also* **mechanism**.

data check

A LUSAS analysis which progresses as far as the pre-solution stage. The data file is checked for errors and a plot file is created for visual checking in the post-processor. Frontal optimisation is carried out if the default solution order commands are included.

data file

The LUSAS database must be formatted before solution. The formatted analysis file is known as the data file.

data transfer file

A file used to store information during a coupled analysis for access by the secondary analysis. *See also* **coupled analysis**.

default

An action or value that a computer or program automatically assumes, unless a different value or instruction is given.

define

To create a dataset. Datasets are used to store information about features, attributes, utilities and results. *See also* **edit**, **feature**, **attribute**.

deformation gradient

A tensor used in continuum mechanics for defining certain strain measures.

deformed mesh

The mesh displayed in a deformed state when subject to an external load or when a mode shape is being displayed. *See also* **undeformed mesh**.

degrees of freedom

Express the manner in which the nodes are free to displace.

delamination

The separation of bonded plys in a laminated composite. It is often referred to as the edge effect because it is most common at the free edges of the structure.

delimiter

A character that marks the beginning or end of a unit of data.

density

A measure of the mass per unit volume of a structure. *Also known as* mass density. *See also* **mesh density**.

departure from linearity

Achieved by using a hierarchical degree of freedom at a central node which represents the relative displacement of the two end nodes.

device

A piece of hardware that performs a specific function. For example, a printer.

dialog box

A box displayed on the computer screen to accept data input from the user. *See also* **OK button**, **apply button**.

diagonal decay

Is an indication of poor conditioning. It indicates that round-off error during the solution has become significant which may lead to inaccuracies in predicted 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 stiffnesses.

diffuse radiation

Radiative heat emitted equally in all directions from the source point.

direct integration

The equations of motion (including mass and damping effects) are integrated in the time domain. The solution is progressed at discrete time points - the time interval between each being the time step. *Also known as* step-by step-dynamics.

direction keys

See **arrow keys**.

directory

Part of a structure for organising your files on a disk. A directory can contain files and other directories (called sub-directories). The structure of directories and sub-directories on a disk is called a directory tree.

discrete point and patch loads

May be used to distribute a given loading pattern over a number of features. Point loads apply a series of discrete point loads and patch loads apply an interpolated loading patch based on specified apex values. *Also known as* HA/HB loading, General loading. *See also* **highway loads**.

discretisation

Sub-division of features into finite elements.

disk drive

A device used to read from and write to disks. *See also* **hard disk**, **floppy disk**.

disk usage

The amount of hard disk space used during an analysis. *See also* **hard disk**

diskette

See **floppy disk**.

displacement

The amount by which a node moves during loading is known as the displacement.

displacement norm

A LUSAS convergence parameter which expresses the limit for the sum of the squares of the iterative displacements as a percentage of the sum of the squares of the total displacements. *See also* **residual norm**, **work norm**, **convergence**.

displacement vectors

Arrows drawn on the screen to represent the direction and magnitude of resultant displacements for a model. *See also* **vector plotting**, **principal stress vectors**.

distributed element loads

Form of element load in which a load variation can be expressed along an element in terms of local or global axes.

domain

A term used for the external boundaries of a finite element problem.

double-click

To rapidly press and release a mouse button twice without moving the mouse. Double-clicking carries out an action, such as starting an application.

double-precision

The use of two computer words to represent each number. Allows the use of twice as many digits as normal to represent a number and allows for extra precision in calculations. LUSAS is a double-precision application.

draping

The laying of a composite fabric over a surface

draw

The action of placing an entity onto the graphics screen.

Drucker-Prager stress

The scalar stress state obtained by combining the individual component stresses at a point according to the classical Drucker-Prager failure criterion. Typically used in the failure of soil or granular type materials. *See also* **Tresca stress**.

ductile fracture

A nonlinear material process by which tensile stresses are progressively released from a cracked Gauss point. *See also* **concrete model**.

DXF

An acronym for AutoCAD Drawing eXchange Format, which can be read into LUSAS Graphics to form the basis of a model's features.

dynamic analysis

Analyses in which the effects of mass are included. This may be in the time domain using direct integration methods or in the frequency domain using eigenvalue extraction methods. *Also known as* natural frequency analysis. *See also* **direct integration**, **eigenvalue analysis**, **explicit dynamics**.

eccentricity

A geometric property of a thin beam, plate or shell that allows an offset of properties from the nodal line such as any applied supports or loading. For thick beams an eccentricity is defined as an **offset**.

edit

To change the contents of an existing dataset. Properties from the edited dataset will automatically be used when the model is next tabulated. *See also* **define**.

edge collapsing

Edge collapsing is the process of removing elements with very short sides or acute angles by merging them with neighbouring elements. This is particularly useful when meshing models imported from CAD.

eigen mode

The natural frequencies of a structure are calculated using an eigenvector analysis. The mode shape is known as the eigen mode. *See also* **eigenvector**, **mode shape**.

eigenvalue

The frequency or load factor magnitude corresponding to the displacement shape resulting from an eigenvalue analysis. Eigenvalues can also be described as the roots of the characteristic equation of the system. The number of eigenvalues relates directly to the number of equations of motion of a system. The square root of the eigenvalues are the resonant or natural frequencies.

eigenvalue analysis

Extraction of natural modes of vibration of a structure. *Also known as* natural frequency analysis, **dynamic analysis**.

eigenvalue buckling

The procedure used by LUSAS to predict the buckling load of a structure. *Also known as* linear buckling.

eigenvector

A deformed shape resulting from an eigenvalue analysis. If a structure is excited at a resonant frequency then the shape that it adopts is the mode shape corresponding to the eigenvalue. *See also* **eigen mode**, **mode shape**.

elapsed time

A measure of total time spent during an analysis or some other computer activity. This time can be greater than the CPU time as it includes effects of other system activity, such as disk access, file transfer, multiple users, etc. *See also* **CPU time**.

elasticity

The property of a material to recover its original form on the removal of the loads.

elasto-plastic

Material behaviour that spans both linear and nonlinear regimes. *See also* **constitutive relationship**.

element

See **finite element**.

element normal

A vector that is orthogonal to the face of an element. Can be displayed graphically to check element face orientations.

element results

Results at nodes within a finite element that are unaveraged with nodal results from surrounding elements. *See* **unaveraged nodal results**.

emissivity

Ratio of the energy emitted by a real surface to the theoretical maximum possible at the same temperature. Emissivity is always in the range 0 - 1.0 (LUSAS assumes reflectivity = 1 - emmisivity).

end-of-file mark (EOF)

A machine or machine equivalent that indicates that the last record of a file has been read.

engineering strain

A strain measure which has its reference as the current length as opposed to the undeformed length.

enhanced strain

A strain field used to modify standard continuum elements to improve accuracy in which the internal degrees of freedom are eliminated at the element level before assembly of the stiffness matrix for the structure.

enthalpy

A physical property of a material which governs how much heat is liberated or absorbed during a phase change.

envelope

Defines the maximum and minimum variation of results across specified load cases, combinations or other envelopes.

environmental node

A means of distributing heat which is transferred to the medium separating a number of thermal surfaces.

environmental temperature

A load type which describes the temperature of the medium surrounding a body.

equilibrium

A state where applied external loads on a model are balanced by the generated internal forces.

equivalencing

Nodes existing at a similar point in space may be replaced by a single node.

equivalent stress

See **von Mises stress**.

error message

A visual indication of hardware or software malfunction or of an illegal data entry attempt or command sequence.

Eulerian geometric nonlinearity

A strain formulation which has its reference as the current configuration. *See also* **updated Lagrangian**, **total Lagrangian**.

excitation

The forcing of a structure by an external influence. Excitation is often associated with dynamic loading.

execute

To carry out an instruction or perform a task.

explicit dynamics

High velocity dynamics, where the rate of loading is high compared to the natural frequencies of the structure being analysed. Explicit problems are classed as wave propagation problems where the behaviour of the wave front, dominated by high frequency components, is of engineering importance. *See also* **impact**, **implicit dynamics**.

explicitly integrated beam

A beam element based on engineers' theory of simple bending. *Also known as* engineering beam.

exploded view

Mesh plots may be viewed with the element vertices moved towards the centroid of the element by a given amount (exaggeration factor).

extension

A three character set that follows a filename. The extension clarifies the filename to the application. It is separated from the filename by a full stop.

extrapolation

A process whereby values outside the limits of a field are extrapolated from values within the field. *See also* **isoparametric mapping**.

face load

A load type that converts a pressure over an element face in 2D or 3D into equivalent nodal loads. *See also* **uniformly distributed load**.

fatigue

When a material is subjected to repeated tensile stresses within the elastic range, the material tires and fractures after a large but finite number of repetitions of stress. This is termed fatigue. *See also* **damage**, **log life**.

feature

Defines the spatial geometry of the structure. Uses an associative hierarchy. *See also* **associativity**.

feature loads

Loads that are applied to a pre-processing feature. They can be varied using variation functions.

fibre location(s)

The position(s) defined on a beam cross-section that can be used to calculate beam stress results for plotting force and moment diagrams.

field analysis

Corresponds to the solution of the general 'quasi-harmonic' equation. A wide variety of problem types are possible, including thermal, seepage, incompressible flow and elastic torsion.

field elements

Finite elements used in a field analysis. *See also* **field analysis**.

file privilege

The settings that can be given to a file or directory to control who on a system has access to read from, write to or execute the file.

file protection

The devices or procedures that prevent unintentional access or erasure of computer files or directories.

file selector

External files are chosen using a file selector box which is machine-dependent. Generally, the filename can be entered in an edit box or chosen with the cursor from a presented list. *See also* **dialog box**.

fine integration

A method of element integration using an alternate selection of a larger number of Gauss points. Can sometimes have the effect of stiffening the mesh. *See also* **coarse integration**.

finite element

A sub-region of the overall model for which the behaviour can be represented mathematically.

first moment of area

A standard section property used to determine the position of the centroid. The first moment of area about the x axis is defined as the integral of x with respect to the area of the section. First moments of area are zero for symmetrical sections. *See also* **second moment of area**, **product moment of area**.

fixed disk

See **hard disk**.

fleshing

The display of a bar or beam element in its true cross-sectional form.

floppy disk

A disk that can be inserted in and removed from a floppy disk drive. *Also known as* diskette. *See also* **hard disk**, **disk drive**.

flow rule

The process by which the direction of plastic straining is calculated. *See also* **associative plasticity**, **non-associative plasticity**.

flux

The rate of flow of heat, usually expressed per unit area.

foam model

Also known as crushable foam *See* **volumetric crushing model**.

force diagram

The graphical display of axial or shear forces on the graphics screen. *See also* **axial force**, **shear force**, **bending moment diagram**.

follower force

See **non-conservative loading**.

forced response

See **harmonic response**.

forward Euler

An explicit plasticity formulation whereby the direction of plastic straining is computed as the yield surface is intersected. *See also* **backward Euler**.

Fourier Transform

A method for finding the frequency content of a time varying signal.

Fourier analysis

Analysis of an axisymmetric structure with non-axisymmetric loading. Achieved by representing the circumferential variations of applied loads and structural displacements as the sum of harmonics of a Fourier series.

fracture

Any form of failure in which stress release occurs. *See also* **ductile fracture**.

frame diagram

This is a graphical representation of the forces and moments in bar and beam structures.

frequency domain

When the forcing function and response of a structure are defined in terms of their frequency content. The inverse Fourier transform of the frequency domain gives the corresponding quantity in the time domain. *See also* **time domain**.

freedom

See **degrees of freedom**.

FRF

Denotes a frequency response function. This expresses the transfer function used in modal dynamics calculations to transfer from the global to the modal domain and allows frequency and time-domain response calculations to be carried out from the results of an eigenvalue analysis.

friction angle

Material property of granular properties, such as cohesive soils and rocks.

frontal method

A method of solving simultaneous equations based on Gaussian elimination.

frontal optimiser

The part of the code that controls optimisation. The order in which the elements are presented for solution makes a significant difference in the time taken and the space required for the solution. The frontal optimiser attempts to order the elements in an optimum manner. *See also* **optimisation**.

frontwidth

The maximum number of unknowns at one time in the solution front is known as the frontwidth of the problem. *See also* **optimisation**.

fully coupled

A coupled analysis in which the structural and thermal files are run at the same time and communicate with each other via a data transfer file. *See also* **coupled analysis**, **semi-coupled**.

fundamental mode

The lowest dynamic natural frequency or buckling load factor of a structure. It is often associated with the simplest mode shape.

gap

A specified distance through which a structure must move before additional external forces come into play. Modelled in LUSAS using joint elements.

Gauss point

Position where numerical integration takes place within an element, hence giving the greatest accuracy of results.

For more information see Gauss Point Quadrature Rules in the appendix of *LUSAS Theory Manual 2*

Gauss point results

Results at the element Gauss point position. *See also* **Gauss point**, **averaged nodal results**, **unaveraged nodal results**.

general loads

See **discrete point and patch loads**.

generalised freedoms

Degrees of freedom specified in a modal dynamic analysis which are used in the reduction of the global system of equations. The accuracy of this method is determined by the selection of appropriate freedoms.

generalised stress

See **von Mises stress**.

geometric nonlinearity

Arises from significant changes in the structural geometry during loading, for example snap-through of a hemispherical cap.

geometric properties

Define the geometrical information about the feature to which they are assigned which has not been specified, for example the thickness of a shell.

global

A term indicating that an entity is expressed relative to the global Cartesian axis set. *See also* **local**.

gradient

See **temperature gradient**.

granular

A characteristic of compound materials such as soils and rocks.

graph dataset

A dataset describing the x axis and y axis data required to construct and display a graph.

graphics area

See **graphics screen**.

graphics screen

The LUSAS main window consists of the areas used to control the software and a central graphics area which displays graphical information.

group

A collection of features given a reference name and used for manipulation of the model.

Guyan reduction

A finite element approximation to low frequency natural vibrations may be obtained by considering only those freedoms (known as master freedoms) whose contribution is of most significance to the oscillatory structural behaviour. The effective selection of master freedoms is central to the accuracy of the simulated structural response.

grillage

A finite element mesh made up of interlocking gril elements. *See also* **beam element**.

HA/HB loading

See **discrete point and patch loads**.

hard copy

A printed copy of computer output in a readable form, such as tabulated results, contour or graph plots.

hard disk

A data storage device usually fixed to a computer to act as the main data storage area. Data is read from and written to the disk using a disk drive. *See also* **disk drive**, **floppy disk**.

hollow volume

A number of surfaces which are not perfectly merged together but are considered by Modeller to form a volume.

hardening

The post yield behaviour of a material.

harmonic response

The steady state response of a structure subject to one or more periodic or vibrating loads of constant frequency. *Also known as* forced response. *See also* **interactive modal dynamics**.

heat fraction coefficient

The fraction of plastic work which is available to be converted into heat energy. This is used in temperature dependent and coupled analyses.

help

Available on‑line by choosing *Help topics* from the Help menu or clicking on the Help push button on any dialog box. *See also* **context-sensitive help**, **dialog box**.

highway loads

A subset of discrete point and patch loads, usually in 2D, used to apply loads to bridges or similar structures. *See* **discrete point and patch loads**.

Hilber dynamics

This is an accurate and efficient direct integration scheme which uses equations based on assumptions made by Hilber, Hughes & Taylor. *See also* **direct integration**.

Hill criterion

A yield criterion which caters for anisotropic plasticity. *See also* **Hoffman criterion**.

Hoffman criterion

A yield criterion catering for pressure dependent anisotropic plasticity. *See also* **Hill criterion**.

hour-glassing

An zero energy mode of deformation induced in an under-integrated element. *See also* **mechanism**.

IGES

International graphical exchange system. *See also* **DXF**, **interface files**.

IMD

See **interactive modal dynamics**.

ill-conditioning

The state of the problem equations when they are numerically unstable. Can be caused by vastly different stiffnesses, or a badly supported structure.

impact

An analysis where one body hits another. *See also* **contact**, **explicit dynamics**.

implicit dynamics

Used for inertial problems where the response is dominated by low frequency components. *See also* **explicit dynamics**.

incompatible elements

Elements using incompatible shape functions to achieve an enhanced stress resolution performance. *See also* **shape functions**.

increment

A step in a nonlinear analysis where a portion of the total load is applied. Results can be obtained at each converged increment level.

incremental loading

Application of a loading scheme in steps (or increments). Used in a nonlinear analysis when full load will not converge.

initial stiffness method

A modified Newton Raphson nonlinear solution method whereby the initial stiffness matrix is used exclusively throughout the analysis.

inertia

A measure of the forces that would be generated in a structure under dynamic excitation as a result of the mass of the structure.

initial stress/strain

Application of initial stress/strain without balancing forces will result in deformation. This is used as a load type in LUSAS.

input/output (I/O)

A general term for devices that communicate with a computer or for communication via those devices.

integer

A complete natural number having no fractional part. For example 32 is an integer while 32.1 is not.

integration point

See **Gauss point**.

interactive modal dynamics

The use of results from an eigenvalue analysis to interactively calculate the frequency/time response of a structure. *Also known as* IMD.

interface files

Used to transfer CAD data into and out of LUSAS Graphics. *See also* **DXF**, **IGES**.

interface stiffness parameter

A property of a slideline surface which controls the penetration of contacted surfaces.

irregular mesh

A mesh applied to a surface that has more than 4 sides.

irregular surface

A surface defined by more than 4 lines which may only be meshed using irregular meshing.

isobeam membrane

A 2D membrane element which bends in-plane as a beam.

isoflex plate

A 2D thin plate which ignores through thickness shears. *See also* **Mindlin plate**.

isoparametric elements

Finite elements formulated using an isoparametric mapping technique. *See also* **isoparametric mapping**, **mapping**.

isoparametric mapping

Describes the geometry in exactly the same way as the displacement field, using the same shape functions. *See also* **shape functions**, **mapping**.

isotropic

A term indicating that the material properties are the same in all directions. *See also* **orthotropic**, **anisotropic**.

iteration

A step within a load increment where the analysis solver attempts to converge to an acceptable solution. *See also* **increment**.

joint element

An element with translational (and rotational) stiffness, which may have initial gaps, contact properties and other nonlinear behaviour. *Also known as* a spring element.

Kbyte (K)

Signifies a quantity memory equal to 2 raised to the power 10 bytes, which is equal to 1024.

kinematic

Plasticity hardening formulation associated with translation, as opposed to expansion, of the yield surface.

Kirchhoff beam

Thin beam which ignores shear deformation. *See also* engineering beam, **numerically integrated beam**.

knife edge loads

A special case of discrete patch load with two or three defining points to model a straight or curved knife edge load. *See also* **discrete point and patch loads**.

label

A label can be drawn on the screen to show feature dataset numbers or the dataset numbers of attributes assigned to them. *See also* **feature**, **attribute**, **assign**.

laminate

An individual layer of a composite material. *See also* **layer**, **composite properties**.

large rotation

A geometrically nonlinear analysis for beam, plate or shell elements where the rotational degrees of freedom undergo large incremental or total change.

large displacement

Common term for geometrically nonlinear analysis where the stiffness of the structure is not constant but becomes a function of displacement.

latent heat

The additional heat gain or release required to obtain a material thermal phase change. Phase change modelling requires a nonlinear thermal analysis.

layer

Certain finite elements, such as shells, are based on a layered formulation. *See also* **composite properties**.

lift-off

A change in a support condition that allows movement away from a support in a certain direction. Used where a structure rests on a foundation but there is no restraint to stop the structure from moving upwards if loading conditions dictate.

limit point

A local maximum point on the load displacement curve. *See also* **bifurcation point**.

line

Feature defining a structural edge, passing through any number of points.

line search

A technique used to improve the convergence rate of Newton iteration by minimising potential energy. *See also* **constant load level**, **arc-length method**.

line section

See **section line**.

linear analysis

An analysis where the relationship between stress and strain is a straight line.

linear buckling

See **eigenvalue buckling**.

link element

A thermal line element, similar to structural joint, which transmits temperature along its length by conduction, convection, and/or radiation.

load combination

The results from several load cases may be combined and viewed as an additional load case. This additional load case is called a combination. Combining loads is only valid for a linear elastic analysis.

load curve

A graphical representation of the variation of a load during an analysis. Can be expressed as a load factor in terms of increment or analysis time.

loading dataset

The loading applied to a structure is defined in a loading dataset then assigned to the required feature.

local

A term indicating that an entity is expressed relative to local element axes. *See also* **global**.

local axes

The axis set used to define an element sign convention.

local coordinate system

May be used to specify the coordinates of a feature or a node in terms of a local Cartesian, cylindrical or spherical axis set. *See also* **transformed freedom**.

local node/segment

The node/element edge on a slideline surface which is geometrically closest to the node currently being considered in the slideline calculations.

log life

Log plot of cycles to failure over a structure. *See also* **fatigue**, **damage**.

lumped mass

A diagonalised form of the mass matrix where the mass is lumped at the nodal positions. *See also* **consistent mass**.

mapping

The numerical process by which quantities, for example shape, results or stiffness, are transferred from one position to another. Often associated with isoparametric elements. *See also* **isoparametric elements**.

mass matrix

A matrix containing the mass terms in a finite element analysis. *See also* **consistent mass**, **lumped mass**.

mass normalisation

Normalisation of eigenvectors with respect to mass. Must be carried out when eigenvalue results will be used to calculate harmonic or spectral response. *See also* **normalisation**, **harmonic response**, **spectral response**.

master freedom

Freedoms at which the mass contribution of the inertia effect is considered to be significant. *See also* **slave freedom**, **Guyan reduction**.

master index table

The master index table is an area of memory that stores an overhead associated with every load case, increment, or time-step dump to a plot or restart file.

master/slave

Terms used to signify paired sets of attributes where the action of one item is subservient to the action of the other. This term is used in the assignment of slides, joint interface meshes and constraint equations.

material properties

Describe the stress-strain relationship and/or thermal properties of an element.

matrix properties

A material model used only for joint elements that allows direct definition of linear properties of stiffness, mass or damping.

mechanism

A deformation mode which has no associated energy (and is not a rigid body deformation) and is, therefore, unrestrained. Usually obvious in elastic problems but can have more subtle manifestations when using material or geometric nonlinearity.

memory-resident program

A program that is loaded into memory and is available even when another application is active. *Also known as* a TSR.

menu bar

The area at the top of the main LUSAS window where pull-down menus are accessed.

merging

Similar features may be 'merged' to give common definitions. Whether a feature is successfully merged with another will depend on the closeness of both members and the attributes assigned to them.

mesh

The discretisation of a body into small finite elements, whose individual behaviour can be predicted.

mesh dataset

A dataset containing details of element discretisation.

mesh density

A term used to describe the relative numbers of elements in a finite element mesh. A dense mesh has more elements than a less dense or coarse mesh. *Also known as* mesh refinement.

mesh lock

The act of preventing a mesh from being updated until it is unlocked. Locking a mesh stops commands such as visualisation of loads and supports from forcing a re-mesh.

mesh objects

Nodes, edges and faces of an element (which itself is a mesh object too).

mesh only modelling

Models comprised of nodes and elements only with no geometric feature types such as points, lines, surfaces or volumes.

message window

Messages are displayed in a scrolling message window.

mirror

A form of transformation in which the model can be duplicated across a mirror plane.

Mindlin plate

These are thick plates which include shear deformation effects. *See also* **isoflex plate**.

modal damping

Damping associated with the displacements defined by the eigenvectors. Its value has no physical significance since the eigenvector contains an arbitrary normalising factor.

modal domain

The solution of a dynamics problem by utilising the system eigenvectors.

modal synthesis

A sub-structuring or superelement technique involving multiple Guyan reductions for very large natural frequency analyses.

mode shape

The displaced shape of a structure due to its natural frequency. *See also* **eigenvector**, **eigen mode**, **eigenvalue analysis**.

model file

The LUSAS database is stored in a binary file called a model file.

modified Newton-Raphson

Modifications to the standard Newton-Raphson nonlinear solution method whereby the stiffness matrix is updated less frequently (common forms are referred to as KT1, KT2, Ko). More iterations are generally required, but the method can be computationally economical. *See also* **Newton-Raphson**.

modem

An acronym for modulator demodulator. A modem converts data from a computer to analogue signals that can be transmitted down a telephone line. It also converts signals from a telephone line into a form the computer can use.

modulus matrix

A matrix of material constants that relates stress to strain. *See also* **rigidity matrix**.

move

To alter the position of a feature using a transformation dataset. *See also* **copy**, **transformation dataset**.

MS-DOS

Acronym for Microsoft Disk Operating System.

multi-point constraint

See **constraint equation**.

natural frequency analysis

See **eigenvalue analysis**.

negative pivot

A well-conditioned matrix could produce a negative pivot if the system is unstable, i.e. passing through a bifurcation or limit point, or if the system is not adequately restrained, i.e. a 3D beam in a 2D analysis. *See also* **pivot**, **small pivot**.

Newton-Cotes

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.

Newton-Raphson

A nonlinear solution method in which a solution is obtained using an iterative approach based on successive recalculation of the stiffness matrix. *See also* **modified Newton-Raphson**.

nodal equivalencing

See **equivalencing**.

nodal results

Results at node positions. Displacements are calculated at nodes, whereas stress results are calculated at Gauss points and extrapolated to the nodal positions. *See also* **averaged nodal results**, **unaveraged nodal results**, **Gauss point**.

node

Finite Elements have their vertices defined by nodes. The nodes have degrees of freedom which are the unknowns at the solution stage.

non-associative plasticity

A plasticity formulation for granular materials in which the direction of plastic straining is not normal to the yield surface. *See also* **associative plasticity**, **plasticity**.

non-conservative loading

When an applied force changes direction during an analysis to match the deformed shape of the model. *Also known as* follower force.

non-orthogonal cracking

Formation of a second (subsequent) crack which is not normal to the first crack direction. That is a cracking formulation which allows re-computation of the principal stress directions following an initial cracked direction formation. *See also* **cracking**, **concrete model**, **ductile fracture**, **orthogonal cracking**.

non-planar

Not lying in a flat plane. *See* **planar**.

nonlinear

A term encompassing all effects that do not have a basis in linear stress analysis. For example, geometric and material nonlinear effects and nonlinear boundary conditions.

nonlinear elasticity

A material which follows a nonlinear (i.e. not straight) path on loading but exhibits complete recovery on unloading.

norm

A measure used in convergence. *See* **convergence**.

normalisation

A process where the maximum displacement of an eigenvalue analysis is expressed relative to unity or to the total mass of the problem.

null

A line mesh containing no elements which is used purely to define mesh spacing is termed a null mesh. A null slideline type allows definition of a slideline in a problem which introduces no contact effects. Slideline

types can be changed during an analysis, so contact can be introduced at a specified point by changing the slide type.

numerically integrated beam

A beam finite element based on numerical integration rather than engineers' simple bending theory. *See also* **explicitly integrated beam**, **beam element**.

object

A **LUSAS object** is a data entity, which is anything listed in the Treeview, or a geometric feature such as a Point or Line. LUSAS objects can be either edited or have their properties displayed via the shortcut menu (right mouse button). *See also* **OLE object**.

OK button

Will apply the dialog box with the given parameters. *See also* **apply button**, **dialog box**.

on-line help

See **help**.

OLE object

A Windows application file that can be linked or embedded into a LUSAS model file. OLE is an acronym for Object Linking and Embedding.

optimisation

The process of reducing the frontwidth of a model to the minimum to increase speed of solution and reduce storage requirements. *See also* **frontal optimiser**.

options

LUSAS options control certain aspects of the analysis. They are set in the data file. *See also* **data file**.

options box

Some LUSAS parameters have only a limited number of possible responses. These are gathered into lists and presented as options boxes.

ordinate

The y axis of a graph. *See also* **graph dataset**, **abscissa**.

orthogonal cracking

Spontaneous or subsequent double cracking whereby both principal stresses violate the cracking criteria and two orthogonal cracking planes develop. *See also* **concrete model**, **ductile fracture**, **non-orthogonal cracking**, **cracking**.

orthotropic

An orthotropic material assumes that the material constants in the two or three principal directions may differ. *See also* **anisotropic**.

oscillation

Repeated numerical variation often associated with out of balance forces (residuals) in a nonlinear analysis.

output file

ASCII text file containing the results of a LUSAS analysis run. It contains warning or error messages from the analysis. It can optionally contain stress, strain, displacement and reaction results.

over damped system

A system which has an equation of motion where the damping exceeds the critical value. It has an exponentially decaying, non-oscillating impulse response.

page format

The layout of a page in a print file may be controlled so as to include headers, footers and titles.

parameter

A variable that is given a value for use by a program or command. Could also be known as a definable characteristic.

parasitic shear

See **shear locking**.

parametric language

Facility for using variables and a control structure as tools during program operation.

patch load

See **discrete point and patch loads**.

patch test

A test in which a convenient shape, for example a rectangle, has its interior divided into a non-symmetrical pattern of the element to be tested. The patch test then assumes a uniform strain over the rectangle, and applying the relevant boundary displacements checks that the finite element solution delivers a constant stress everywhere.

peak value plot

A post-processing plot that shows the largest maximum or minimum (or both) values of a results entity on the screen. The peak values can be shown as a symbol or as a value.

penalty function method

A method of monitoring the interaction of two contact surfaces which, upon contact, imposes interaction forces to oppose penetration.

penetration

A term used in a contact analysis to define the relative overlap of meshes on separate contact surfaces. *Also known as* interpenetration.

perfectly plastic

A material that will not sustain additional load after yielding. The stress strain curve is effectively flat above the yield stress.

peripheral

An external input/output, printing or storage device.

perspective view

Takes into account the distance from the viewpoint to the object when displaying the plot.

phase angle

The ratio of the in phase component of a signal to its out of phase component gives the tangent of the phase angle of the signal.

phase change

The transformation of a material between phase, usually being accompanied by a liberation or absorption of heat.

picking

See **cursor selection**.

picture files

Used to store LUSAS pictures in a neutral plotting format.

pivot

A pivotal term in a matrix that is being solved using Gaussian elimination. *See also* **small pivot**, **negative pivot**, **diagonal decay**.

planar

Lying in a flat plane. *See* **non-planar**.

plastic hinge

The formation of a position in a structure where the applied moment has exceeded the plastic moment capacity resulting in a loss of rotational stiffness or hinge.

plasticity

A nonlinear material description associated with yielding of material (e.g. metals, plastics, soils).

plot

See **contour plot**.

plot file

The binary results file written by the LUSAS solver which is used to pass results information to the post-processor.

plot scale

The ratio of screen coordinates to model coordinates is known as the plot scale.

point

Defines the coordinates of the vertices of the structure.

point load

See **discrete point and patch loads**.

poisson's ratio

A material property value indicating the amount of a longitudinal strain that is translated into a contraction due to lateral strain in a Hookean material.

polar decomposition

Method of decomposing a deformation gradient into a stretch and rotation tensor. *See also* **deformation gradient**.

polar second moment of area

For a plane area in the XY-plane, the polar second moment of area is defined as the integral of X squared plus Y squared over the area of the section. The polar second moment of area thus equates to the second moment of area about the x axis plus the second moment of area about the y axis. For circular sections the second polar moment of area is equal to twice the second moment of area about the x or y axis. *Also known as* torsional moment of inertia.

post-processing

The process of viewing the analysis results.

post-solution

Any part of the finite element solution process following the computation of the displacements, for example strain, stress, moment recovery and output phases.

power law

A creep formulation using three material parameters which include primary, secondary and tertiary regions of creep behaviour. *See also* **strain hardening**.

pre-conditioned conjugate gradient solver

Iterative solution using a conjugate gradient solver is very sensitive to the matrix conditioning. For this reason the matrix is usually pre-conditioned before solution.

pre-processing

The process of building the model ready for analysis.

pre-solution

Any part of the finite element solution before the solution of the displacements. It is usually associated with the formation of elemental stiffness matrices and load vectors.

prescribed variable

The specification of the value of a nodal displacement or of a differential of nodal displacement with respect to time. Chiefly used to prescribe displacement values.

pressure-dependent yield

Plastic deformation where the amount of plasticity is dependent on the sign of stress (tension/compression). For example, Hoffman, modified von Mises material models.

pressure loading

Loading expressed in per unit area or unit length terms.

primary creep

The initial nonlinear part of the creep curve.

principal stress vectors

Vector plot showing maximum and minimum principal stress directions and relative magnitudes. In LUSAS the vectors are colour coded to indicate sign. *See also* **vector plotting**, **displacement vectors**.

print file

A text file to which information can be printed from LUSAS.

product moment of area

See also **first moment of area**, **second moment of area**.

program

A series of instructions in a form acceptable to a computer. *Also known as* **application**.

pseudo-nodes

Dummy nodes created at element edges when a slice is cut in LUSAS. Results calculated at these pseudo-nodes are used to contour the slice.

pseudo-time

A numerical time quantity used in a creep analysis.

pull-down menus

Cascading menus may be 'pulled down' from the main menu bar to pass instructions to LUSAS.

quadrature

Numerical (usually approximate) evaluation of an integral, for example Gauss, Newton-Cotes.

quarternion

A quantity consisting of a scalar and vector component which defines a rotation in 3D space. A normalised quarternion is sometimes referred to as an Euler parameter.

quasi-harmonic

A name for the governing field equations used in thermal or seepage analyses.

radiation

The energy emitted by bodies as a consequence of their temperature. *See also* **convection**, **conduction**.

radio button

When several parameters are mutually exclusive, they may be presented in a dialog box as a series of radio buttons. They are signified by diamonds or circles, and when one is clicked all others in that group will be released. *See also* **dialog box**.

rate dependent

Time dependent material properties.

Rayleigh damping

A form of damping which is constructed from user specified proportions of the stiffness and mass matrix terms. *See also* **structural damping**, **viscous damping**.

reaction

A force or moment calculated at a support position to counteract any applied loading.

real number

An ordinary number, either rational or irrational.

reference path

A route (or path) through the model that provides a concept of distance from an origin to each point in the model.

regular mesh

A mesh lying on a regular surface consisting of the same numbers of divisions on opposing lines.

regular surface

A surface defined by 3 or 4 lines, hence may be meshed using any meshing technique. *See also* **ruled surface**.

release

The act of freeing a beam end rotation. *Also known as* moment end release.

renumber

To change the numbers of features. Usually used to rationalise feature numbers into consecutive ascending order.

residual force

A measure of the unbalanced force present in the structure after solution.

residual norm

A LUSAS convergence parameter which expresses the limit of the sum of the squares of all residual forces as a percentage of the sum of the squares of all external forces. *See also* **convergence**, **displacement norm**, **work norm**.

residual stress

Stresses that are locked into the structure in its initial configuration.

resonance

The amplification of displacements occurring when the frequency of applied loading and natural vibrations coincide or are similar.

restart file

A file containing the relevant parts of a LUSAS analysis database that will allow an analysis to be restarted from a previously converged situation.

restaint

See Supports.

resultant

See **stress resultant**.

results calculator

See **calculator**.

results dataset

See **graph dataset**.

results file

See **plot file**.

retained freedom

A term used to describe master and slave freedoms used in a Guyan reduced eigenvalue extraction.

reverse

To reverse the orientation of a line, surface or volume feature. *See also* **cycle**.

ribbed plate

A 2D high performance incompatible membrane element. Takes into account both membrane and flexural deformations. Transverse shearing effects are ignored. Designed to be used in conjunction with ribbed plate beam elements.

rigidity matrix

A matrix of material constants that relate stress resultants to strain. *See also* **modulus matrix**.

round-off

See **diagonal decay**.

RU/RL loading

Light/standard rail loading similar to HA/HB.

rubber

A hyper-elastic nonlinear material description with (approximately) constant volume characteristics.

ruled surface

A surface which has at least two opposite lines that are straight. The special case of spherical surfaces are also considered to be ruled surfaces. Some surface operations will only work with ruled surfaces. For example, splitting, intersecting, filleting. *See also* **analytical surface**, **regular surface**, **irregular surface**.

s-n curve

A curve used to express damage done to a structure for a given number of loading cycles. Standard curves are available in BS5400. *See also* **fatigue**, **damage**.

sampling point

Sampling points are the locations along a tendon profile at which calculated equivalent tendon loads will be applied to the model.

See also **Gauss point**.

screen annotation

See **annotation**.

screen coordinates

The coordinate system used to position annotations on the graphics area.

script file

A name given to a collection of UNIX commands used to set up an environment and to run a program*. See also* **batch file**, **parametric language**, **command file**.

search area

The area over which a general point or patch load is applied may be limited using a search area.

secondary creep

Commonly used linear region of the creep curve.

second moment of area

The second moment of area of a plane area with respect to the x axis is defined by the integral of x squared over the area. *Also known as* moment of inertia of area*. See also* **first moment of area**, **polar second moment of area**, **product moment of area**.

section line

A section along a line, from which results can be taken and placed in graph datasets*. See also* **graph dataset**, **section slice**.

section slice

A 2D slice through a 3D model on which contours can be displayed. The slice can be further interrogated using a section line to obtain a graph dataset*. See also* **section line**, **graph dataset**.

segment

A section of a contact surface. On a slideline a segment is an element edge, and on a slide surface a segment is an element face. In addition, an individual screen of graphics information processed during a LUSAS animation is termed a segment.

seismic analysis

An analysis of the effect of an earthquake on a structure*. See also* **spectral response**.

semi-coupled

A coupled analysis in which the structural and thermal analyses are run separately and one before the other. *See also* **coupled analysis**, **fully coupled**.

semi-loof beam

A curved line element based on the semi-loof formulation. *See also* **semi-loof shell**.

semi-loof shell

A 3D plane stress thin shell element based on the semi-loof formulation where the element hinge (or loof) rotations are lumped together at the edge mid-side nodes.

session file

All commands issued during a LUSAS session will be recorded in a session file. This may be replayed to recover any work lost after a system crash.

shear area

The cross-sectional area of a structure that can be said to react transverse shear loading.

shape functions

Mathematical mapping functions used to define the variation of displacement or other variables within an element. *See also* **isoparametric mapping**.

shear force

Transverse force. *See also* **axial force**, **stress resultant**.

shear lag

A delay or slow response in developing shear flow reactions to applied loads.

shear locking

Causes over stiff performance in bending and occurs as a result of the bending deformation mode requiring strain energy storage by shear strain as well as by normal strain. *Also known as* parasitic shear.

shear retention factor

The factor by which the in-plane shear modulus is reduced following formation of a crack plane in the biaxial concrete model. *See also* **concrete model**, **ductile fracture**.

shell elements

A form of plate element that considers both membrane and bending behaviour.

shifting

A procedure used in eigenvalue extraction where the stiffness matrix is shifted to overcome numerical problems when rigid body modes are present.

SI units

Units of the Système Internationale d'Unités. The most relevant to finite element analysis are as follows. Length: metre (m); mass: kilogramme (kg); time: second (s); temperature: Kelvin (K); force: Newton (N); frequency: Hertz (Hz).

singular

See **singularity**.

skew angle

A measure indicating a departure from orthogonality for a bridge deck. Also used to specify the angle between reinforcement directions for Wood‑Armer calculations.

singularity

A theoretical infinite stress, or a condition found during solution which indicates that insufficient information has been provided to solve the stiffness matrix. Often occurs due to insufficient specification of boundary conditions.

slave freedom

Freedoms in a model where the mass contribution of the inertia effect can be considered to be relatively insignificant. *See also* **master freedom**, **Guyan reduction**, **retained freedom**.

slide surface

A 3D contact surface made up of surface boundaries. *See also* **slideline**.

slideline

A 2D contact surface made up of line boundaries. *See also* **slide surface**.

slides

Contact and impact problems may be analysed with the use of slide lines or slide surfaces on the contacting features. The LUSAS contact facility uses the penalty function approach. *See also* **slideline**, **slide surface**, **penalty function**.

smoothing

The process of creating a smoother transition between stresses in adjacent elements by using a calculated nodal average value at common nodes. *See also* **averaged nodal results**, **unaveraged nodal results**.

small pivot

Indicates a poorly conditioned stiffness matrix. *See also* **diagonal decay**, **negative pivot**.

snap through

Large deflection nonlinear effect whereby a structure converts membrane to bending energy and adopts a stable but usually inverted configuration.

solidification

See **phase change**.

spawn

The action of creating a sub-process to carry out a specified task. LUSAS Graphics can spawn a process to run LUSAS Solver on UNIX machines.

spacing ratios

Used by line mesh datasets to express the variation in segment length when meshing a line.

specific heat

A thermal material property associated with the resistance to temperature change. It is the equivalent of inertia in a structural analysis.

spectral curve

Input curve for a response spectrum.

spectral response

An analysis in which a 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.

spherical path

See **arc-length method**.

spline

A type of line defined between points, using a mathematical expression.

spring element

See **joint element**.

SRSS method

A method used in spectral response analysis, where design displacements, stresses and forces are evaluated as the square root of the sum of the squares. *See also* **CQC method**.

start-up file

A file used to convey system set-up information to an application on start-up or at pre-defined points during its execution. The LUSAS start-up file can contain system parameters and valid LUSAS commands. These files are run when the program is initiated and when a new model is loaded or initialised.

starting iteration vectors

The number of terms used as initial guesses in an eigenvalue extraction. *See also* **Iteration**.

static

An analysis which assumes that the response to the load is immediate because inertial effects are ignored.

steady state response

The response of a system when all of the transient components have become insignificant. *See also* **transient response**.

steepest descent method

Implicit plasticity algorithm by which inadmissible elastic trial stresses are returned to the yield surface.

step-by-step dynamics

See **direct integration**, **dynamic analysis**.

stiffness matrix

The matrix containing terms that relate the displacement of a structure to the applied forces. *See also* **b ~matrix**, **d ~matrix**.

strain energy

The work done by an external force in displacing a structure is stored as strain energy. It may be released on unloading for elastic analyses or cause permanent deformation in elastic-plastic analyses.

strain hardening

The ability of a material to continue to accept stress following NL yielding, usually at a reduced rate.

strain softening

Post-yielding or cracking behaviour whereby continued loading results in a release of accumulated stress and a negative stress-strain slope. It is often called tension-stiffening with reference to the biaxial concrete model.

stress recovery

The process of calculating the stress results from the deformed shape and stiffness matrix.

stress vectors

See **principal stress vectors**.

stress resultant

Generally useful in civil design and defined as stress per unit length (or width). Conversion to a stress is achieved by dividing by the thickness.

string

A character-based variable.

structural damping

Specifies the damping of each system mode of vibration via the hysteretic damping factor. *See also* **viscous damping**, **Rayleigh damping**.

Sturm sequence check

A stable procedure which indicates the number of eigenvalues below a given eigenvalue and is utilised to test for missing values. *See also* **eigenvalue**.

sub-session file

Commands written to the LUSAS session file may also be selectively logged to a duplicate file. This is useful for creating a short command file of one or two commands during a LUSAS session. *See also* **command file**.

subspace iteration

Evaluation of the lowest eigenvalues and corresponding eigenvectors is achieved by performing a simultaneous inverse iteration using a set of iteration vectors followed by a projection of the problem matrices onto the subspace formed by these vectors. *See also* **Iteration**.

substructure

See **superelements**.

superelements

Permit the finite element model to be divided into smaller, more manageable components which are then added together to form the complete structure. Individual components are known as superelements or substructures.

superposition

For a linear problem the solution is the same if it is found by adding together two or more individual loads or forcing functions and then solving or by solving for the separate loads or forcing functions and then combining the results or responses. The second method is called superposition. *See also* **load combination**.

support surface

Four-sided surface used to define the internal geometry of an irregular non-planar surface.

supports

Describe how the model is restrained at its boundaries, or may be used to infer symmetry conditions on a model.

surface

Feature defining the faces of the structure. Defined as being bounded by lines. *See also* **regular surface**, **irregular surface**, **ruled surface**.

system parameters

The LUSAS Graphics and LUSAS Solver databases may be tuned on initialisation using system parameters in the start-up file.

tabulation

The process of creating an analysis data file from the pre-processing model file.

tangent stiffness

The formation of the stiffness matrix at a given equilibrium position.

temperature gradient

The variation of temperature through the thickness of an element. This distribution induces bending.

termination

Completion point of an analysis. Termination is controlled using the termination line in a nonlinear or dynamic control.

text output window

A separate window is created to display information in a text format requested by the user. *See also* **message window**, **graphics screen**.

thermal analysis

See **field analysis**.

thermal gap

A space between two bodies in a thermal analysis through which heat will flow.

thermo-mechanical coupling

See **coupled analysis**.

thick element formulation

An element formulation that includes the effect of transverse shear deformation.

thickness update

A procedure used in plane stress situations where the thickness is updated due to the effect of Poisson's ratio.

thin element formulation

An element formulation that ignores the effect of transverse shear deformation.

tied slideline

A type of slideline setting that effectively ties two disconnected meshes together to move as if connected. *See also* **slides**.

time domain

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. *See also* **frequency domain**.

time step

The period over which the time is integrated in a dynamic analysis. *See also* **dynamic analysis**.

toggle

The act of turning on or off a function.

toggle button

A button on a dialog which represents a parameter which may take only one of two values (for example yes/no).

total Lagrangian

A strain formulation which has its reference as the initial configuration at the start of the analysis. *See also* **updated Lagrangian**, **Eulerian geometric nonlinearity**.

transformation dataset

Describes a transformation in space, for example a translation, mirror, etc. which may be used to generate, move or copy features. *See also* **move**, **copy**.

transformed freedom

The transformation of the local axis system of a node from the global system.

transient analysis

The field equivalent to a structural dynamic analysis in which the effects of thermal inertia are included (i.e. specific heat).

transient response

System response that is not periodic.

transition mesh

A mesh generated on a regular surface where the number of divisions on any two opposite boundary lines are unequal.

Tresca stress

The scalar stress state obtained by combining the individual component stresses at a point according to the classical Tresca failure criterion. Typically used in the failure of soil or granular type materials. *See also* **Drucker-Prager stress**.

triaxial

A description of a material model in which three axes are considered to determine the material response. *See also* **biaxial**, **uniaxial**.

TSR

An acronym for terminate-and-stay-resident program. *See also* **memory-resident program**.

unaveraged nodal results

Most results are calculated at the Gauss points of an element and then extrapolated to the nodes. When the results from the elements joined at a node are extrapolated to the nodes but not averaged together these are known as unaveraged nodal results. *See also* **averaged nodal results**, **smoothing**.

undeformed mesh

A mesh displayed in its original shape without any imposed deflections. *See also* **deformed mesh**.

under damped system

A system which has an equation of motion where the damping is less than critical. It has an oscillating impulse response.

unified creep

A creep formulation whereby creep and plasticity are combined.

uniaxial

A description of a material model in which one axis is considered to determine the material response. *See also* **biaxial**, **triaxial**.

uniformly distributed load

A load type which specifies a load per unit length along a line element or a load per unit area for a surface element.

units

A system of dimensions in which the model and associated properties and loads are defined. *See also* **SI units**.

unzoom

The action of reverting to an unmagnified view of the structural model. *See also* **zoom**.

updated Lagrangian

A strain formulation which has its reference as the end of the last converged increment. *See also* **Eulerian geometric nonlinearity**, **total Lagrangian**.

upgrade

To expand a system by installing options or using revised or updated software.

utility dataset

A utility dataset stores items in the database that are not assigned directly to features. *See also* **transformation dataset**, **control dataset**.

variable

A quantity that can assume any set of values as a result of processing data.

variation

A dataset used to vary an attribute over a feature. A variation dataset can be described in terms of field variables, interpolation values and functions.

vector plotting

Results with both a magnitude and direction may be represented by a vector whose length is proportional to the magnitude. *See also* **displacement vectors**, **principal stress vectors**.

velocity

First derivative of displacement with respect to time. *See also* **acceleration**.

view factor

A measure used in a thermal radiation analysis to indicate how much exposure a face has to the radiation emitted from another face. A view factor will vary between 0 for no exposure and 1.0 for full exposure.

virtual work

The principle of virtual work states that, for any small, virtual displacement imposed on a body, the total internal work must equal the total external work for equilibrium to be maintained.

visco-elastic

A nonlinear time-dependent model in which strain is totally recoverable.

visco-plastic

A nonlinear time-dependent model in which there is a permanent set after unloading.

viscous damping

Damping can be said to be viscous when the damping force is proportional to the velocity. *See also* **structural damping**.

volume

Feature defining the solid nature of a structure. Defined as being bounded by surfaces.

volumetric crushing model

Represents the behaviour of crushable foams and certain soils which exhibit significant volumetric strain. It is applicable to plane strain, axisymmetric and 3D stress states only.

von Mises stress

The scalar stress state obtained by combining the individual component stresses at a point according to the classical von Mises failure criterion. Typically used in the failure of metals. *Also known as* generalised stress, equivalent stress.

warping

The degree to which the nodes of a shell element do not lie in the same plane.

wdl file

A windows definition language file used by LUSAS GRAPHICS to define the main menu bar and associated pull-down menus.

wildcard

A character that represents one or more other characters. For example, *.exe represents all files that end with the .exe extension.

window

A component screen area within the LUSAS GRAPHICS work area. For example, graphics window, text window.

Wood-Armer

Method of calculating the moments of resistance required in reinforcement that are placed in any two directions of a linear solution.

word

The set of bits comprising the largest unit that the computer can handle in a single operation.

work norm

A LUSAS convergence parameter which expresses the limit for the work done by residual forces based on iterative displacements as a percentage of the work done by the loads in the first iteration of a nonlinear problem. *See also* **convergence**, **displacement norm**, **residual norm**.

write-protect

An action that can be taken with a diskette to prevent data being written to it. There is a switch in the corner of the diskette which when moved into the open position write-protects the diskette.

XMS memory

Extended memory that can be accessed by using the eXtended Memory Specification from Lotus, Intel, Microsoft and AST. LUSAS applications work with an extended memory manager which makes sure that only one application is using a portion of memory at any one time.

yield criterion

The condition used to establish whether a material has reached its ultimate elastic condition. *See also* **yield stress**.

yield flags

Indicate the spread of plasticity in a nonlinear material by plotting yielded Gauss points.

yield stress

The level of stress at which a material is said to start unrecoverable or plastic behaviour. *See also* **Tresca stress**, **von Mises stress**.

Young's modulus

The linear relationship between stress and strain for a Hookean material.

zonal detection

The area around each node on a slide surface that is used to search for contacting nodes on another surface.

zoom

The action of increasing the display magnification of a specified portion of the graphics screen or structural model.

Index

M

T

U

V

W

Y

Z