

# **New Facilities and Improvements**

LUSAS Version 14.4-1

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](mailto:info@lusas.com)

<http://www.lusas.com>

Distributors Worldwide

Copyright ©1982-2010 LUSAS

All Rights Reserved.

## Contents

<b>Overview .....</b>	<b>1</b>
<b>New section property generators / calculators.....</b>	<b>1</b>
Additions to the standard section property calculators .....	1
Single and multi-cell pre-cast box section property calculator.....	1
Addition to supplied standard section libraries.....	2
<b>Visualisation of section properties .....</b>	<b>3</b>
Tapered arbitrary cross-sections.....	3
Fleshing of arbitrary cross-sections .....	3
<b>Multiple varying sections based upon a distance along a line or path of lines .....</b>	<b>4</b>
Reference paths .....	5
<b>Loading and supports .....</b>	<b>6</b>
Discrete compound loads (Load sets or load trains) .....	6
Editing of discrete loading data .....	6
Oregon vehicle loads .....	6
Load factor assignment / changing .....	7
Loading visualisation.....	7
Stress and Strain loading .....	7
Prestress loading changes.....	8
Deassignment of loading and supports.....	9
<b>Viewing Results .....</b>	<b>9</b>
Results plots.....	9
User Defined Results .....	10
Report generator .....	11
Load combinations: combined Interactive Modal Dynamics (IMD) results and linear results .....	11
Plotting results on slices through the model .....	11
Cracking and Crushing.....	12
Graphing.....	12
Print results wizard.....	12
<b>General user interface changes .....</b>	<b>12</b>
Layer visibility on/off.....	12
Layer names for results layers .....	12
New Utilities layer.....	13
Labels.....	13
Joints .....	13
Colouring surfaces according to their connectivity .....	13
Expressions.....	13
Editing of geometric and material data .....	13
Datatips .....	13
Thermal analysis .....	14
Rate of Internal Heat Generation (RIHG) loading.....	14

Various other user interface changes .....	14
<b>New elements / changes to elements .....</b>	<b>15</b>
New crack tip elements and crack tip modelling attribute .....	15
New composite solid and field elements .....	15
New 3D flat thin nonlinear shell element .....	15
Changes to elements .....	15
<b>Network Licensing .....</b>	<b>16</b>
Improved Network Licensing .....	16
<b>Documentation .....</b>	<b>17</b>
User Manuals .....	17
Changes to Worked Examples .....	17
<b>Retired elements and facilities .....</b>	<b>17</b>

# Overview

LUSAS Version 14.4 is a significant new release with many new features and enhancements. In summary it contains:

- A range of new and improved features for bridge and structural users
- A number of significant general modelling, results viewing and other user interface improvements for all users.
- Various useful and often asked-for development requests
- Improvements to network licensing
- Updates to online documentation and printed manuals

The main features in these new facilities and improvements are grouped according to type and described in the following pages.

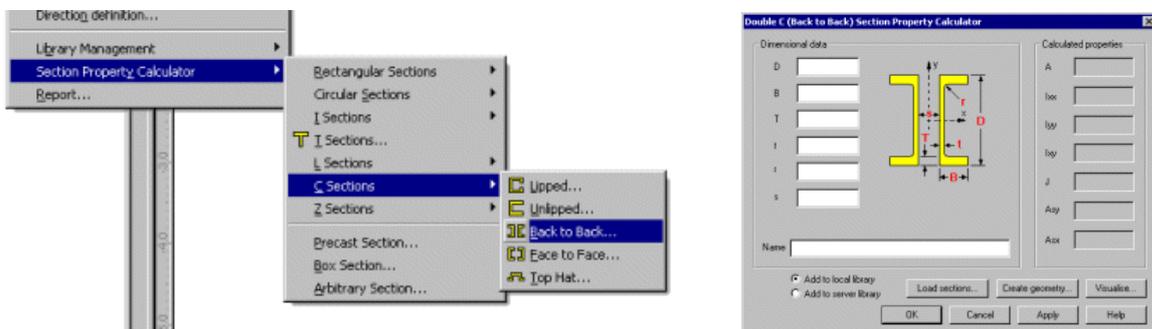
## New section property generators / calculators

### Additions to the standard section property calculators

New **Standard section property calculators** have been provided for:

- an unequal flanged I-section
- an I-section haunch shape
- unequal walled rectangular hollow sections
- back to back L-sections
- C-sections (lipped and unlipped)
- back to back and face to face)
- Z-sections (lipped and unlipped)
- A top-hat section.

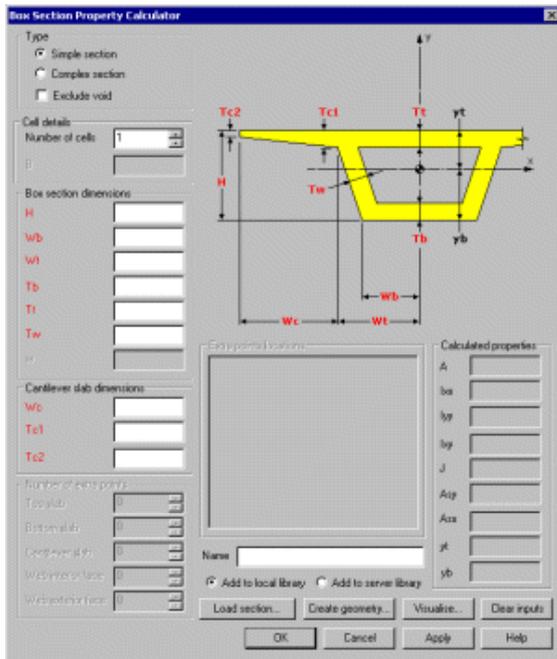
These new sections join the existing standard section property calculators and accessed from the **Utilities> Section Property Calculator** menu item.



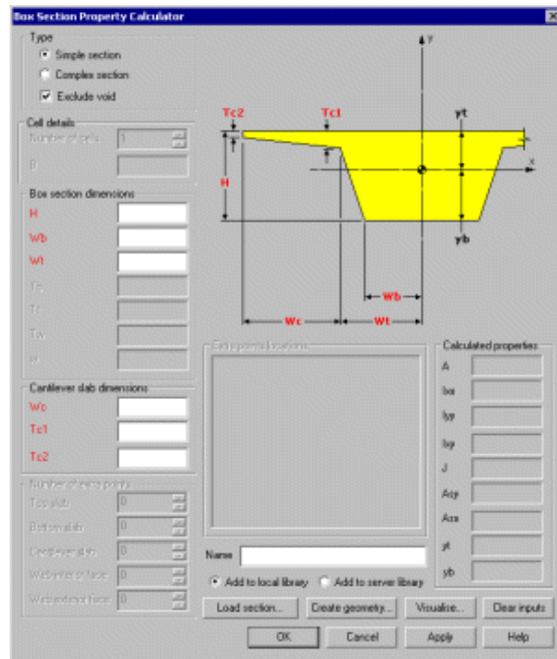
### Single and multi-cell pre-cast box section property calculator

A **Box Section Property Calculator** is now provided in Bridge and Civil & Structural software products. It calculates the section properties of a single or multiple box cross section created from user defined dimensional data. The section shapes can be defined either as a simple cross-section or as complex section created from as many lines as are required to form a suitable representation of

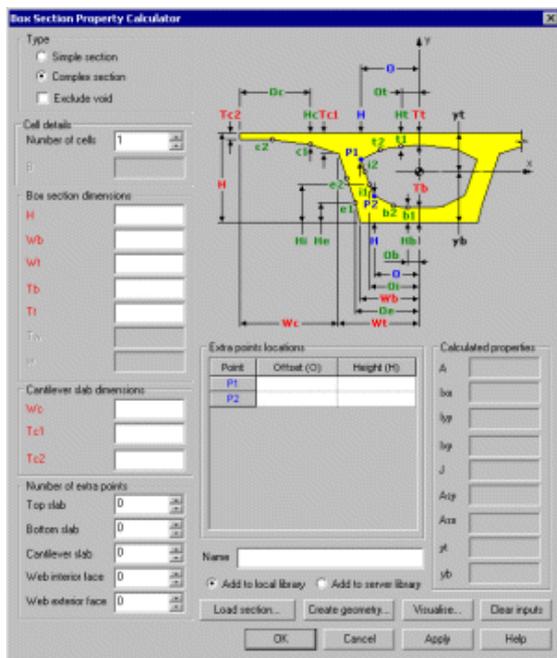
the true cross-sectional shape. A void can be included or excluded from a section. Generated sections can be saved in project section libraries or converted into model geometry for additional manipulation.



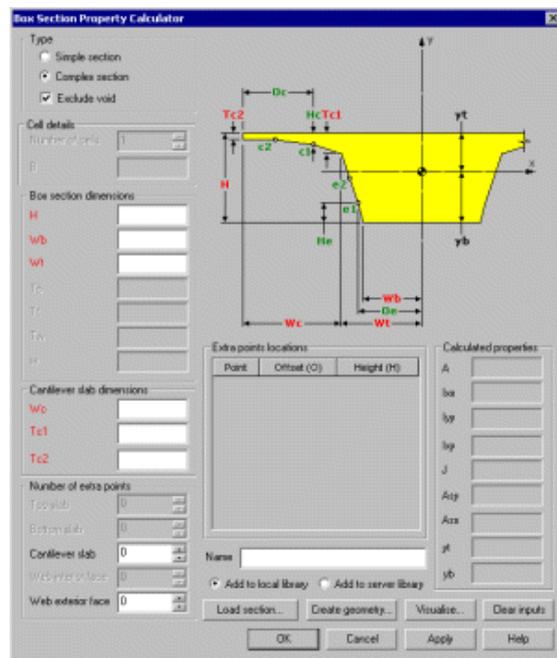
Simple box section with void



Simple box section without void



Complex box section with void



Complex box section without void

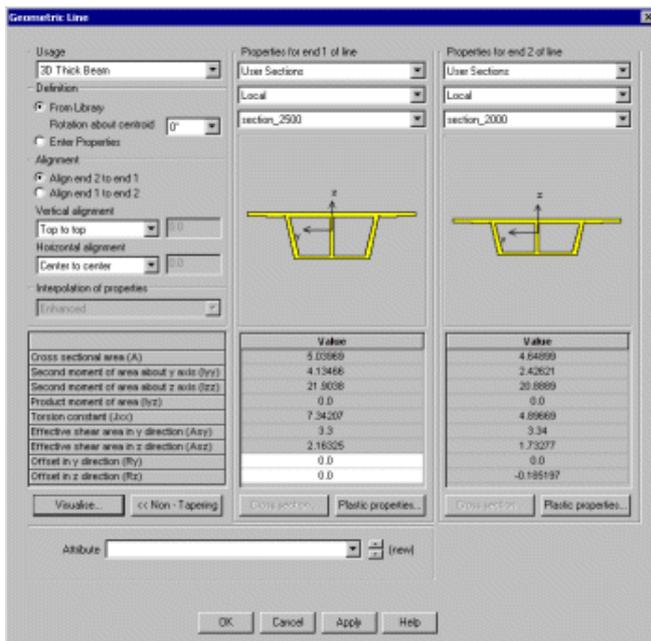
## Addition to supplied standard section libraries

- Chinese standard steel sections have been added.
- A range of precast Northeast Bulb T-sections has been added.

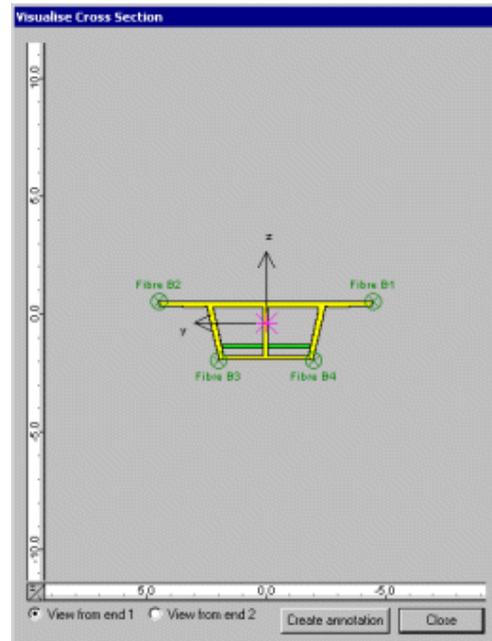
## Visualisation of section properties

### Tapered arbitrary cross-sections

Tapered arbitrary cross-sections can now be defined for supplied or user-defined sections that are saved to a section library. Previously only standard cross-sections could be specified to taper along a beam's length. Tapered sections are defined by using the Attributes > Geometric > Line (From Library) and Attributes > Geometric > Section Library (User section) menu items and specifying section properties for each end of the beam. When the tapering option is chosen, the vertical and horizontal alignment of one end of the beam section from the other can be specified and the method of interpolating the section properties for locations along a beam can also be defined using an enhanced or linear method if an exact calculation cannot be made. Offsets (eccentricities of beam ends from nodal positions) can also be defined.



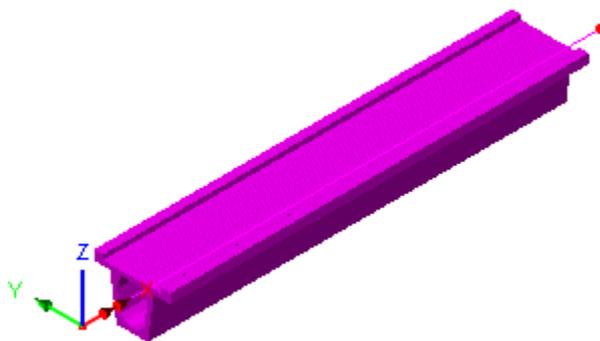
Selecting cross-sections to defined a tapering section



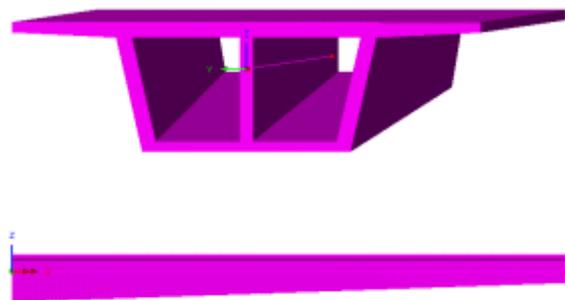
Alignment visualisation of tapering cross-section

### Fleshing of arbitrary cross-sections

- Fleshing of arbitrary and also tapered arbitrary sections is now provided.



Fleshing of a constant arbitrary section on a beam element



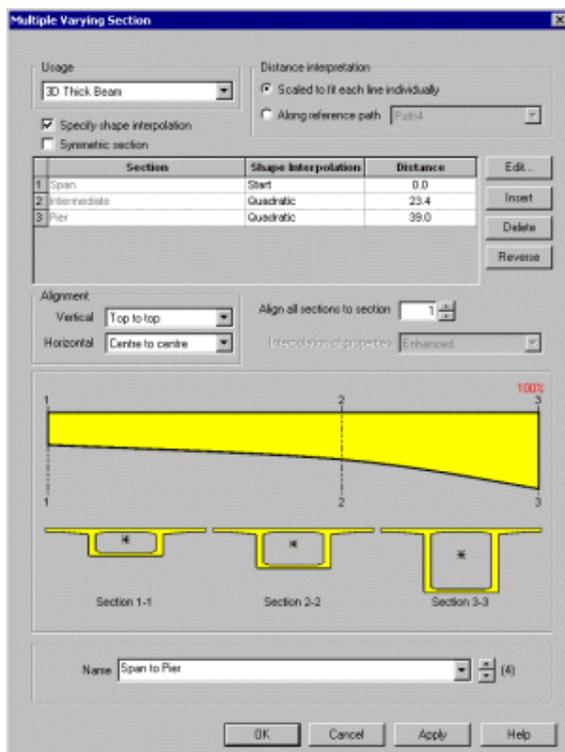
Fleshing of a tapering arbitrary section on a beam element

## Multiple varying sections based upon a distance along a line or path of lines

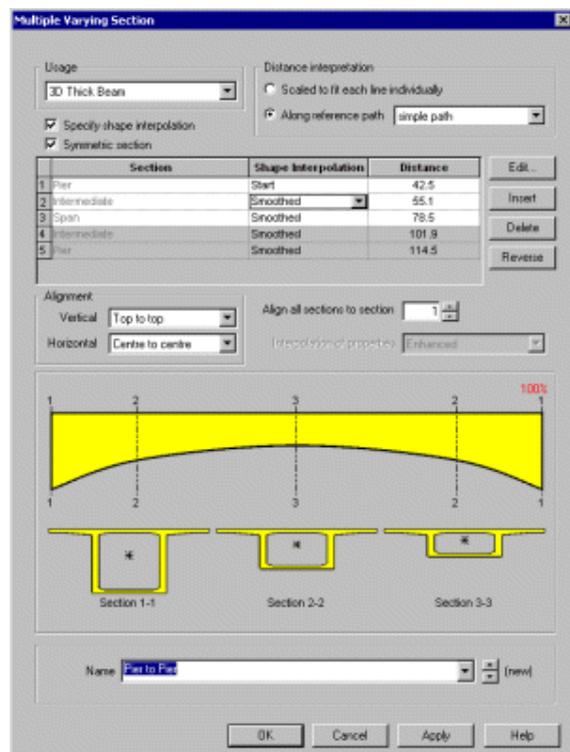
A new **multiple varying section** facility enables pre-defined cross-sections to be specified at distances for subsequent assignment to a single line or to a series of lines with reference to a pre-defined **reference path**.

On the dialog, 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. As the multiple varying section is defined in the table, a visualisation of the longitudinal vertical alignment of the cross-section shapes used is displayed on the same dialog.

The multiple varying section dialog is accessed by using the **Attributes > Geometric > Multiple Varying Section** menu item.

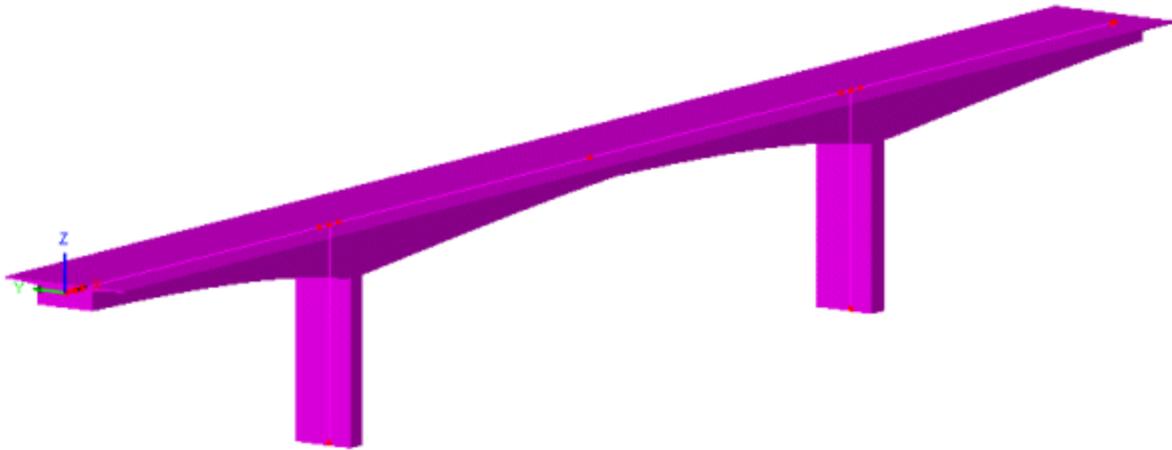


Varying sections defined for assignment to a single line



Varying sections defined to follow a reference path

Use of the new multiple varying section facility will enable simple preliminary or 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.

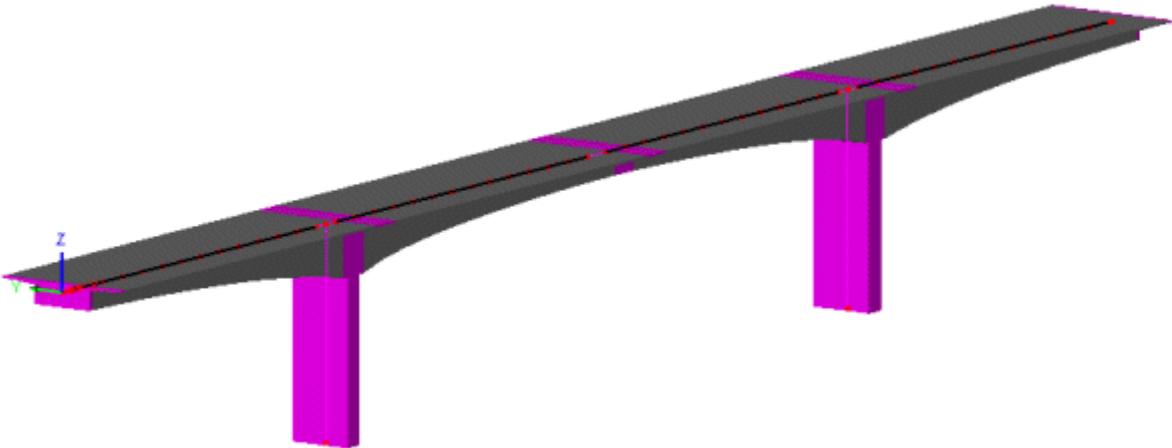


Varying section line attributes assigned to single lines on a model

## Reference paths

**Reference paths** are a new concept introduced in this release. 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 specified on the Path definition dialog, 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.

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 and the actual profile of the longitudinal members as a whole may vary along the longitudinal beam's length.



Varying section line attributes (shown selected) assigned to multiple lines on a model with reference to a path

Use of the new multiple varying section with the reference path facility will enable complete 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. When used to create more detailed line beam models for use with activation and deactivation facilities,

it is possible to create models for staged construction analysis much quicker and far easier than previously.



A new worked example, '3-Span Concrete Box Beam Bridge of Varying Section' has been added to the Application Examples Manual (Bridge, Civil and Structural). This examples illustrates the use of the new multiple varying sections and reference path facilities.

## Loading and supports

### Discrete compound loads (Load sets or load trains)

**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.

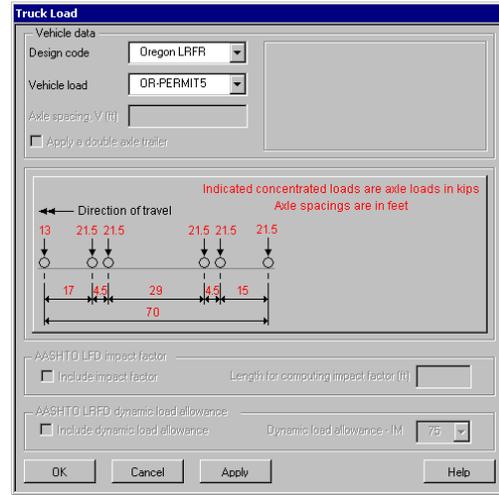
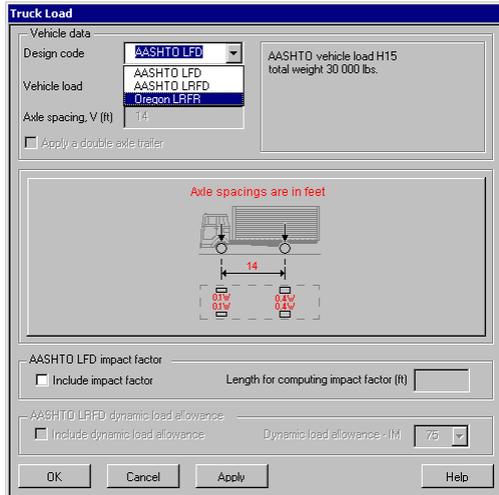
A compound discrete load attribute may be defined from any combination of existing point, patch and compound discrete loads. Creating a compound discrete load attribute is similar to the way in which a load combination is created, namely, predefined discrete load types are dragged from the Available panel of the Compound dialog into the Included panel. When created, compound discrete loads are held in the Loading section of the Attributes  Treeview in their own section named Compound. Compound loads enable complex loading patterns to be defined using far fewer actual loading attributes than were required previously.

### Editing of discrete loading data

Editing of pre-defined discrete loading data now 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. After the creation of a vehicle load, the resulting attribute in the Attributes  Treeview now has a special context menu entry named Edit Definition in addition to the existing Edit Attribute context menu entry. These menus can be seen by right clicking on the attribute.

### Oregon vehicle loads

- Oregon Department of Transportation (ODOT) vehicle loadings with reference to the ODOT LRFR Load Rating Manual (updated 06/10/2008) have been included in this release. They include the following types:
  - ODOT Continuous Trip Permit (CTP) Trucks
  - ODOT Legal Trucks (LEG)
  - ODOT Permit Load (PERMIT) Trucks
  - ODOT Single Trip Permit (STP) Trucks
  - Specialized Hauling Vehicles (Denoted as SU Trucks)



## Load factor assignment / changing

- The load factor that is defined during assignment of loading attributes is now appended to the loading name in the Loadcases Treeview. This displays as, for example, (x1.0) or (x2.0). Where previously only one item represented all load factor assignments, now each loading entry represents all those assignments having the same load factor. This makes for easier checking and changing of assigned load factors.
- Load factors of assigned loads can be changed by viewing the Loadcases Treeview and right-clicking on the loading name and selecting the Change load factor menu item from the context menu.

## Loading visualisation

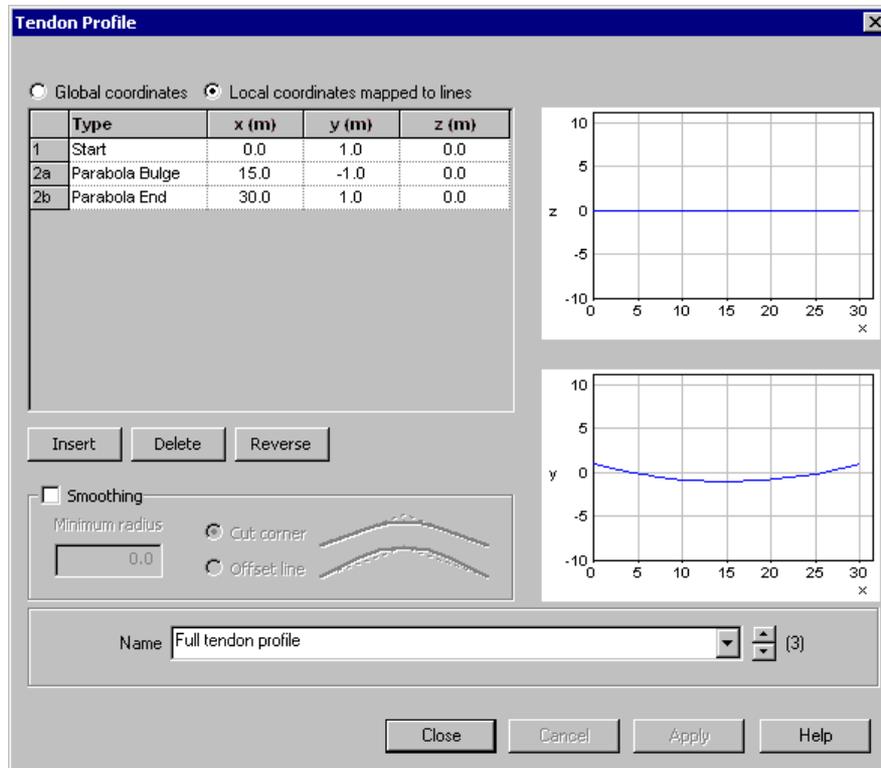
- The default setting for the visualisation of loading has changed from 'by definition' to 'by effect on mesh'.

## Stress and Strain loading

- A much improved dialog for the definition of initial stress and strain loading has been provided. The input values required for each element are now listed according to the selected element type.

## Prestress loading changes

Improved tendon profile definition is provided for multi-tendon use. The tendon profile dialog allows tendon profiles to be defined or copied and pasted from a spreadsheet. It also shows tendon profile geometry and allows specification of line segment connection type when a tendon profile has been generated by defining lines, arcs or splines in Modeller. Tendon profiles are now stored in a new Utilities layer.



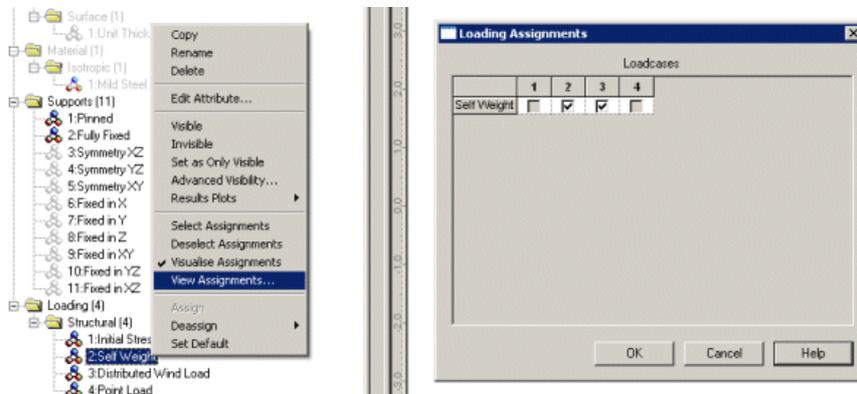
- Options for defining tendon segments and their control points are: Straight, Arc Bulge, Arc Centre, Spline, Parabola Bulge, or Spline Continued.
- Smoothing options allow the definition of a minimum radius around or through which two intersecting tendon segments will be shaped.

In addition the following tendon related changes have been made:

- Tendon diameter has been replaced with tendon area on all prestress dialogs
- Deassignment and deletion of any single tendon or all multi-tendon prestress wizard created loading from the Attributes Treeview is now permitted.
- Tendon integrity checks are now made. If a model is altered, you are reminded that the prestress loads (which were calculated by the prestress wizard) will need to be recalculated.

## Deassignment of loading and supports

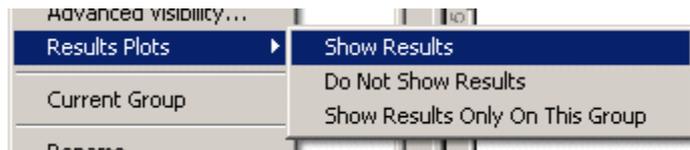
It is now possible to quickly deassign attributes such as supports and loading from assigned loadcases by deselecting the items listed in the View assignments dialog.



## Viewing Results

### Results plots

The method of showing results on groups, section slices and slidelines as provided originally in Version 13 of LUSAS has been re-instated.



Three new menu entries, Show Results, Do Not Show Results, and Show Results Only On This Group have been added to the context menu for group names, section slice names and slideline names. These are all accessed from within the Results Plots menu item.

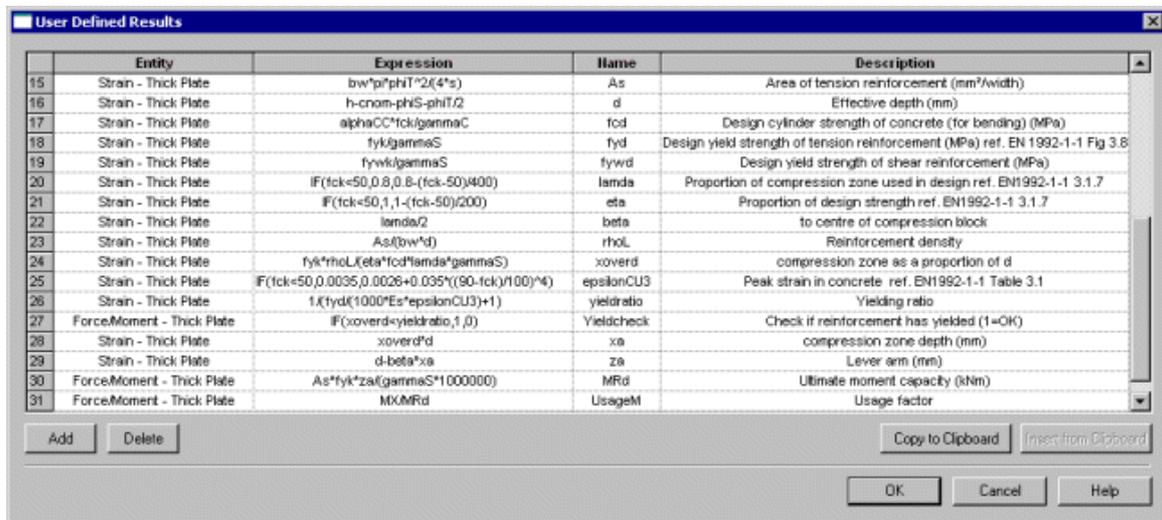
The same functionality has also been added to the context menus of all assigned attributes in the Treeview and for selected items in the View window by using the context menu for the item selected. This means that:

- Isolated results for assigned material types, geometry, mesh types, supports or loadings can be easily plotted without having to define individual groups for these items.
- Regions of the model can be selected in the View Window either by cursor or by using the Advanced Selection dialog and results can be plotted directly on these selected features without having to pre-define groups first.

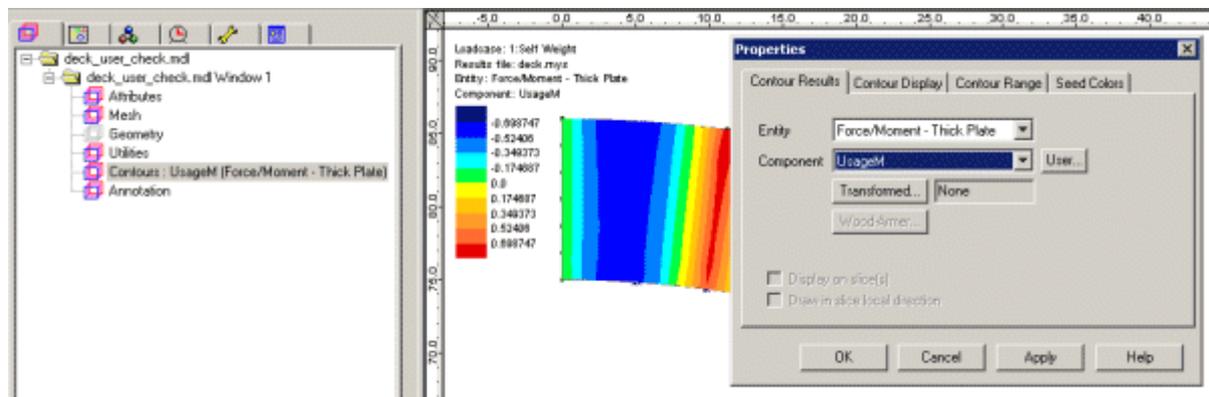
## User Defined Results

A new **User Defined Results** facility allows results components to be defined by creating arithmetic expressions based upon 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.

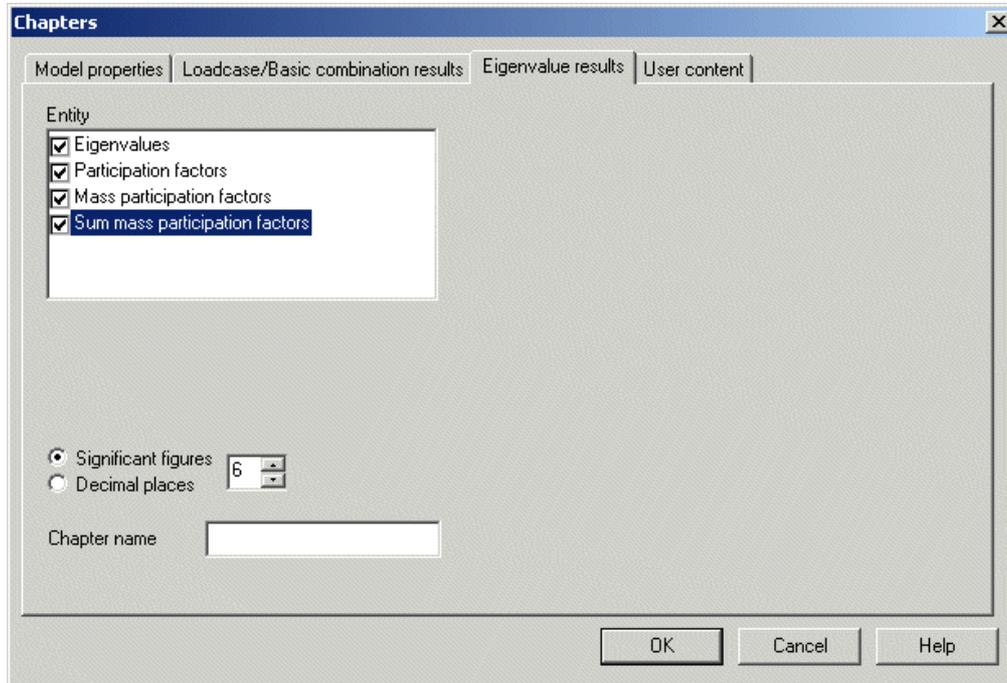


After definition, the created user results components can be selected by name from the Component drop-down list on the Contours, Values, and Diagrams layers properties dialogs. Graphing of user defined results is also possible.



## Report generator

- IMD loadcases can now be added to a report. These are selected on the Model properties chapter dialog.
- Eigenvalues results can now be added to a report. These are selected on a new Eigenvalues chapter dialog.



- It is now possible to specify different numbers of significant figures or decimal places for each report chapter inside a report. For example, a report can contain a results subchapter that lists displacements with a precision of 3 decimal places and have another results subchapter that lists Forces / Moments to 6 significant figures. It is also possible to override this by setting a value to be used for all chapters on the main Report properties dialog.
- Documentation describing the use of the report generation facility has been extensively updated to detail all aspects of its use.

## Load combinations: combined Interactive Modal Dynamics (IMD) results and linear results

- IMD loadcases (this includes Spectral Analysis) can now 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 another combination including a dead/live load with a load factor of 1 and a spectral loadcase with a load factor of -1.

## Plotting results on slices through the model

- When cutting a slice section through a 3D model an option to create an annotation polygon is now provided. This annotation polygon effectively defines the orientation of the cutting plane and gets saved with the model. This annotation polygon may be re-selected if a model is reloaded at a later date in order to generate and view results at the same location.

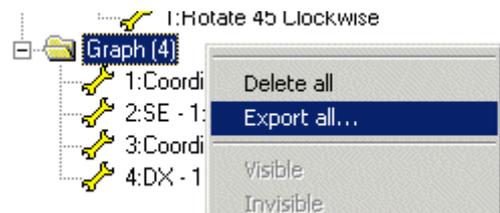
- As an alternative to using annotation polygons, slice section results can also now be plotted using surfaces to indicate and slice section locations. These surfaces do not have to surround a model, they must simply be defined in the orientation of the cutting plane required.

## Cracking and Crushing

- A new component **Crack/Crush** has been added to the Values dialog to allow plotting of cracking and crushing symbols on concrete material models. Choosing Crack / Crush will produce the fullest failure plot possible, including crack planes and/or crush locations, dependent upon the material model in use. The Crack option previously on the Vector properties dialog has been removed.

## Graphing

- It is now possible to export either 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.



## Print results wizard

- The Print Results wizard now remembers the number of significant figures set when the wizard is used until Modeller is closed and restarted.

## General user interface changes

### Layer visibility on/off

- Treeview layers can now be turned on/off to prevent having to delete them from the Layers Treeview in order to stop layer data from being displayed in the model view. This is done by selecting the On/Off menu item from the layer's context menu. Turning a layer off retains the layer in the Layers Treeview but removes the display of that layer from the Graphics Area. A symbol adjacent to each layer name in the Layers Treeview shows the visibility and status of each layer:

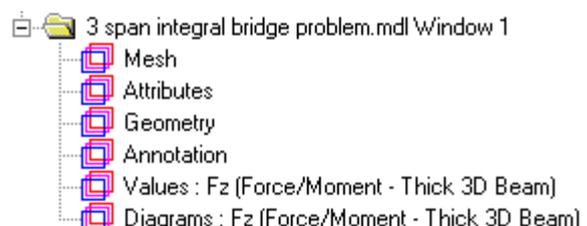
 A coloured layer symbol indicates that the display of the layer has been turned 'on'.

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

 No results are loaded or currently available for this layer, or inappropriate settings are currently set.

### Layer names for results layers

Contours, Diagrams, Vectors and Values results layer names as added to and seen in the Layers Treeview now have the component (such as Mx, My or Fz, for example), appended to the layer name followed by the entity type such as (Force/Moment - Thick 3D Beam). If different component and entities are selected at a later time



the layer name in the Layers  Treeview is updated to reflect the chosen selection.

## New Utilities layer

A new Utilities layer has been introduced. This controls visualisation of Tendon Profile, Reference Path, Background Grid and any other software product-specific utility data.

## Labels

- The Labels properties dialog now allows selection and labelling of Annotation features and of labelling the Description of selected Line, Surface, Volume and Element features. When labelling these features line length, radius, surface area, volume size and element type are also included.

## Joints

- A cosmetic change to the joint dialog allows the selection of a joint type prior to defining the joint properties.

## Colouring surfaces according to their connectivity

- The ability to colour surfaces by their connectivity to other features is provided. This works in a similar manner to the way in which the option Colour by Line Connectivity works. Features are drawn in different colours according to the number of higher order features that are connected to them. To use, go to the Layers  Treeview and right-click on the Geometry layer name, select Properties from the context menu and choose to colour by Surface Connectivity, clicking Set to filter the displayed surfaces based upon their connectivity status. 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 could be used to correct and connect any unmerged and isolated features.

## Expressions

- A new **if (condition, then, else)** logical condition has been added to all existing expressions that can be used when inputting values on text entry fields and grids throughout the Modeller user interface. A full list of expressions and functions available has been included in Appendix E of the *Modeller User Manual*.

## Editing of geometric and material data

Geometric line and material data added to the Treeview as a result of selecting an item from the section or material library will now display the original library dialog as opposed to a generic section or material input dialog when the section or material attribute in the Treeview is edited. To change material properties to add, for example, plastic properties to an isotropic material right-click on an attribute name and choose Edit Attribute.

## Datatips

- The data tip (seen when hovering your cursor over a model feature) now also reports the underlying geometric feature type e.g. straight line, arc, planar surface as well as provide dimensional data for that feature e.g. length of line, radius of arc, area of surface etc.

## Thermal analysis

- Thermal bar elements can now be accessed from the drop-down list when selecting the **Attributes > Geometric > Section Library** menu item when using a Thermal user interface.

## Rate of Internal Heat Generation (RIHG) loading

New field material models have been included which incorporate the concrete heat of hydration and HPM resin cure kinetics properties. As such, the requirement to define concrete and resin cure models via the assignment of a separate internal rate of heat generation loading to selected features is no longer required.

- Input of concrete heat of hydration properties is now achieved by entering data on the Thermal tab of either the isotropic or orthotropic material dialogs.
- Generalised user defined RIHG loading can still be defined.
- For more details refer to the *Element Reference Manual*.

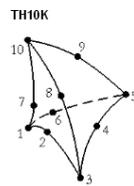
## Various other user interface changes

- The number of decimal places can be specified in addition to significant figures when plotting numeric output.
- A new radio button permitting the saving of a model to the current folder has been added to the New Model dialog.
- A keyboard shortcut (**Ctrl +Shift** key) has been added to allow deselection of already selected items.
- After features have been selected, using the **Ctrl + M** key sets the selected items into Selection Memory.
- A notes section has been added to the Model Properties dialog to allow entering of model-specific notes.
- It is now mandatory to save a model file prior to solving to prevent results being out of sync with models.
- All annotation data added to the Annotation layer is now given an identifying name. This is done automatically if no user-defined name is given. One example of use is for plotting results on slices through the model where previously defined annotation polygons defining a slice through a model may be re-selected if a model is reloaded at a later date in order to generate and view results at the same location.
- 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.

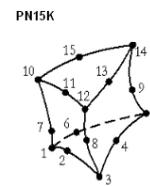
## New elements / changes to elements

### New crack tip elements and crack tip modelling attribute

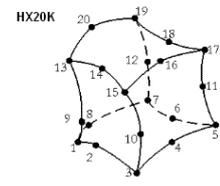
- The family of crack tip elements has been extended to include 20 node hexahedral (HX20K), 15 node pentahedral (PN15K) and 10 node tetrahedral (TH10K) 3D quadratic continuum crack tip elements. These elements are specifically designed to be applied to fracture mechanics problems and may be used to model the singularities that occur at the crack tip. See the *Element Reference Manual* for details.



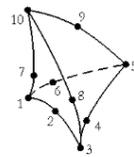
Crack specified at Node 1



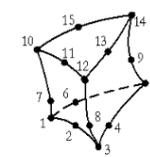
Crack specified at Node 1



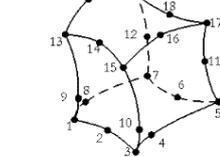
Crack specified at Node 1



Crack specified along edge 1-2-3



Crack specified along edge 1-2-3

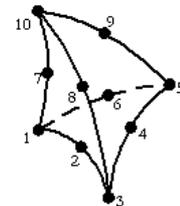


Crack specified along edge 1-2-3

- When modelling, a **crack tip attribute** can now be defined and assigned to any point in a 2D model or at any point or line in a 3D model. After assignment the mid-point nodes of any quadratic continuum 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.

### New composite solid and field elements

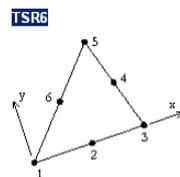
- A 3D solid laminated composite tetrahedral element (TH10S) and a 3D solid composite tetrahedral field element (TF10S) has been released for use by composites users. Laminated tetrahedral elements allow meshing of composite components to be carried out using standard tetrahedral meshing software and their use can give large savings in modelling time when compared to using hexahedral and pentahedral elements for complex geometry problems. See the *Element Reference Manual* for details.



- XML interface allows import of drape information allowing designers to define drape using FiberSIM or Advanced Composite Modeller.
- Layer top/bottom and mid surface results can be examined to determine stresses, strains and composite failure criteria.

### New 3D flat thin nonlinear shell element

- A 3D flat thin laminated nonlinear shell element (TSR6) has been released. See the *Element Reference Manual* for details.



### Changes to elements

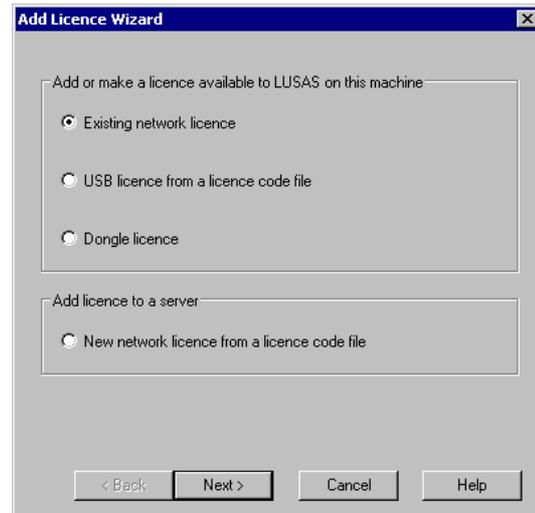
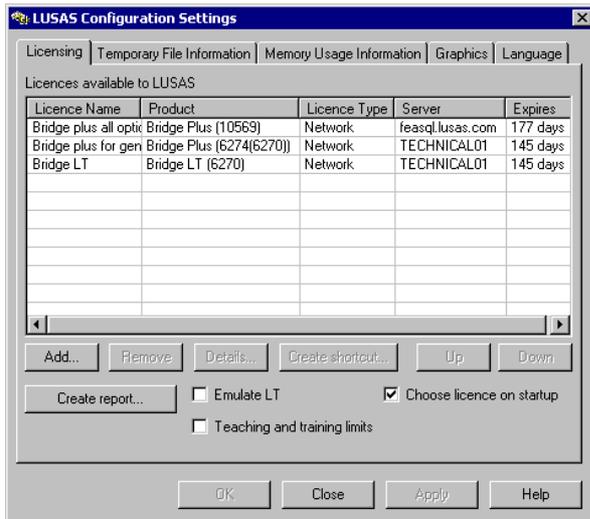
- 3D Thick Shell elements (TTS3, TTS6, QTS4, QTS8) permit the setting of a new option (option 396) which invokes an improved transverse shear calculation. This new option is set 'on' by default for models created by version 14.4 and above, and set 'off' - for models created by previous versions.
- The Flat Thin Box Shell Element (SHI4) has been retired. QSI4 should be used in place of this element.

## Network Licensing

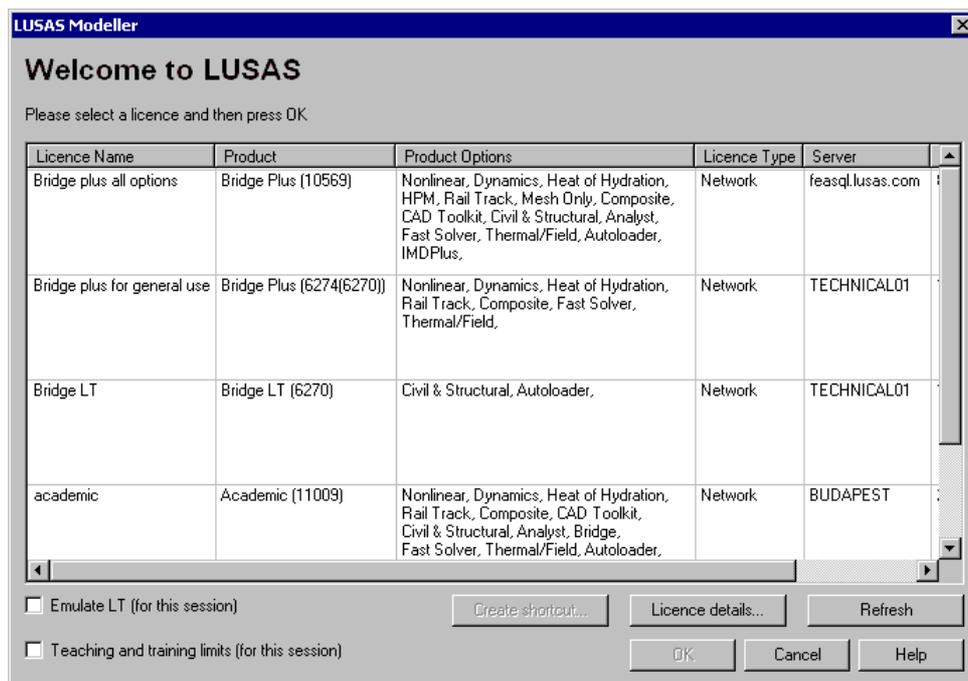
### Improved Network Licensing

Changes have been made to make the use of LUSAS licences simpler and to ensure that licences are only used as appropriate.

- An updated Configuration Utility Licensing page now lists all available licences and provides the means to add and administer all Network, USB and Dongle licences through a single dialog.



- A new **licence selection dialog** can be optionally displayed to allow users to select a particular LUSAS licence. It lists all available licences (including all Network, USB and Dongle licences listed in the FEA.KEY file), along with details of the licenced product options, licence type, server name, number of available licences and the number of days left until expiry.



- If the licence selection dialog is turned off Modeller will now work its way through the list of licences on the Configuration Utility Licensing page until a valid and available licence is found.
- If an analysis is run, LUSAS Modeller now passes all details of the licence it is running with to LUSAS Solver, including the minimum set of licence options required to solve the job. Solver then tries to use the same licence as the Modeller licence and if not possible re-orders all suitable Solver licences and internally lists them such that the least functional Solver licence that is still able to solve the job is attempted first.
- If Modeller is running with Teaching and Training limits Solver also runs with the same limits (effectively not using a Solver licence)
- Shortcuts to Modeller and Solver executables can be created that also specify a particular licence to be used.

## Documentation

### User Manuals

All relevant documentation has been updated for this new release and is provided in PDF format as part of any V14.4 software download file or on the V14.4 software CD. Particular changes of note include:

- The use and creation of toolbars and toolbar buttons has now been documented in the Modeller User Manual
- Documentation describing the use of the report generation facility has been extensively updated in the Modeller User Manual to detail all aspects of its use including the creation of sub-reports and exporting of data to spreadsheets.



### Changes to Worked Examples

- All existing worked examples have been reviewed and updated as necessary for this new release.
- A new worked example, '3-Span Concrete Box Beam Bridge of Varying Section' has been added to the Application Examples Manual (Bridge, Civil and Structural). This example illustrates the use of the new multiple varying sections and reference path facilities.

### Retired elements and facilities

- The PATRAN export facility is no longer supported and has been withdrawn.

