LUSAS

New Features and Improvements in Version 17.0

New features / enhancements

LUSAS Version 17.0

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

New Facilities and Improvements in Version 17.0

Version 17.0 of LUSAS sees the steel frame design checking and vehicle load optimisation capabilities extended, prestress enhancements, additional creep and shrinkage models, and new facilities for soil-structure interaction modelling. New attribute viewing and editing features simplify the management and editing of assignments, improving general usability and enabling faster editing.

- The Steel Frame Design option now supports EN1993-2: 2006 Eurocode 3: Design of steel structures - Part 2: Steel Bridges. Steel frame design to EN1993-1-1: 2005+A1:2014 has been enhanced to include Class 4 sections and to support shear buckling of I-sections. The implementation for AASHTO LRFD 7th Edition has been updated to allow user-defined input of buckling properties.
- Vehicle load optimisation has been extended to include Finland's LO 24/2014 design code, Norway's NS-EN 1991-2.2003 NA 2010 + NA-rundskriv 07-2015, and the UK's BD21/01 Annexes D and E.
- Prestress modelling has been re-structured and sees the single tendon and multiple tendon prestress wizards, which were used to generate prestress loading, retired. Instead, a tendon loading

attribute, which references tendon profile and tendon property data, can be directly assigned to selected line, surface or volume features in a model, consistent with all other attribute assignments made within LUSAS. New tendon loading assignment options state how the tendon loading is to be applied and, for assignment to lines, mapping and offset options can be used, as well as specifying in which loadcase(s) the tendon loading should apply. Prestress loading is now calculated as a function of solving a model. Prestress data can now be included in a model report by adding a prestress chapter. Graphs showing prestress losses in tendons are now produced using the graph wizard. Editing of tendon loading, property and profile data has been improved and can be done by individual selection of one or more attributes, or by selection of the parent folder containing a set of attributes.

- New material models include Creep and shrinkage for AASHTO LRFD 7th edition, IRC:112-2011, a Duncan-Chang soil material model, and a joint material model that allows definition of symmetrical matrix properties -useful for specifying a joint representing a pilecap sitting on a group of piles in a global (3D) model.
- Initial stresses in soil can now be calculated by providing Ko data for the supported material models that are used in soil analyses. Modelling of the variation of material properties at any location in a model can now be achieved by defining a profile variation. Typical uses include modelling of varying soil properties with depth, as obtained by borehole data
- Section property calculation of a range of riveted or welded section types is now provided in Bridge and Civil & Structural software products. Sections can be defined with reference to sections held in the main section library or from user-defined values. Indian and New Zealand steel sections have been added to the steel section library.
- Target values for stress and strain can now be stated such that a specified value for a selected results component for a feature will be obtained exactly when a solve has taken place. This facility is primarily for use when a specified jacking force is required to be applied to one end of a line representing a cable.

- Editing of attribute assignments is simplified by the new Assignments editor, which allows simultaneous viewing, editing and management of any number of attribute assignments, and is of particular use in editing tendon profile and property assignments. A new Attributes editor similarly enables easier viewing and editing of loading data for a range of structural load types, and is of particular use when editing tendon loading values when carrying out prestress modelling.
- For selections made in the View window, the Properties dialog has been replaced with a new Properties panel, which allows for changing the properties of any number of concurrently selected features/objects, and permits assigning or deassigning attributes to and from those items. A new Hierarchy panel lists the higher and lower order features/objects of the currently selected geometry feature or mesh object.

Note in the following pages, which provide summary information of the changes made, all references to more information relate to topic headings in the online help or appropriate PDF manual.

In summary

Steel design

- Steel design to EN1993-2 (Bridges)
- Steel design to EN1993-1-1
- Steel design to AASHTO LRFD 7th edition

Traffic Load Optimisation

- Finland LO 24/2014 added
- Norway NS EN1991-2:2003/NA:2010 added
- United Kingdom BD2101 Annex D and E added

Prestress

- Prestress modelling restructured and easier to use
- Prestress loading now supported by shells

Material modelling

- AASHTO LRFD 7th edition creep and shrinkage material model
- IRC112-2011 creep and shrinkage material model
- Duncan-Chang soil material model added
- Matrix property definition for joint elements
- Initialisation of ground conditions
- Profile variations in soil

General enhancements

- Section property calculation of plated sections
- Indian steel section library added
- New Zealand steel section library added
- Australian taper flange beam properties updated
- Stress and strain target values for cables
- Improved editing of assignments
- Editing of multiple load attributes now supported
- Discrete load projection over area for shell models
- New Properties panel
- New Hierarchy panel

Element library

- Updated to reflect Modeller enhancements

Documentation

- Generally updated

Updated worked examples

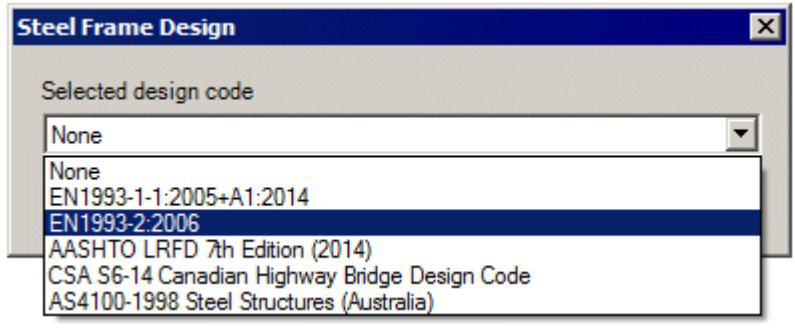
- Linear analysis of a post tensioned bridge
- Segmental construction of a post tensioned bridge
- Drained nonlinear analysis of a retaining wall

Steel design improvements

Steel design to EN1993-2: 2006 Eurocode 3

Steel frame design now is now supported to EN1993-2: 2006 Eurocode 3: Design of steel structures - Part 2: Steel Bridges. Eurocode National Annexes currently supported are:

- Eurocode Recommended Values (EN 1993-2:2006)
- Finland (SFS-EN 1993-2/NA:2010)
- Ireland (I.S. EN 1993-2/NA:2010)
- United Kingdom (BS EN 1993-2:2006/NA+A1:2012)

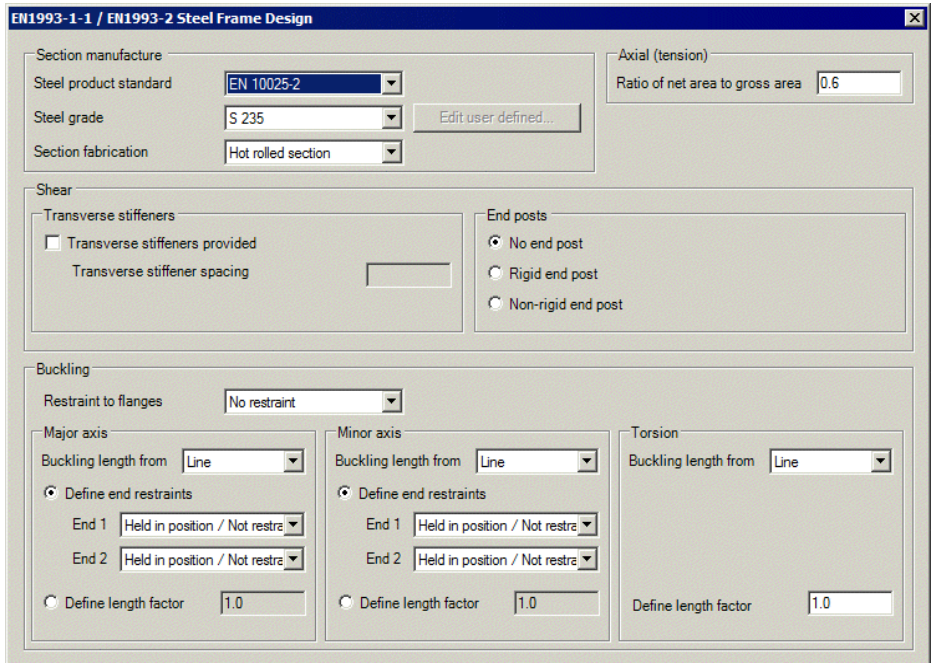


For more information see **Steel Design Code Checking**

Steel design to EN1993-1-1

Steel frame design to EN1993-1-1: 2005+A1:2014- EN 1993-1-1:2005 Eurocode 3: Design of steel structures – Part 1-1: General rules and rules for buildings has been enhanced.

- Class 4 sections are now supported.
- Shear buckling of I-sections is now supported.
- The reduced design plastic resistance moment allowing for shear force as given by expression (6.30) of EN1993-1-1 for I-sections with equal flanges has been extended to include class 3 sections.



For information about this design code see **Steel Frame Design Attribute for EN1993-1-1 and EN1993-2**

Steel design to AASHTO LFRD 7th edition

The Steel Frame Design attribute dialog for the AASHTO LRFD 7th Edition code has been updated to separate the input of buckling properties into those for compression buckling and those for flexural buckling. As a result it is now possible to specify a user-defined unbraced / buckling length for compression buckling, and a user-defined unbraced length for flexural buckling.

For more information see **Steel Frame Design Attribute for AASHTO LRFD 7th**

LUSAS Traffic load optimisation improvements

Finland LO 24/2014

When the country name 'Finland' and Design Code 'LO 24/2014' is selected on the Vehicle Load Optimisation dialog, traffic loading can now be generated according to Finland standard:

- **LO 24/2014 - Application of the Eurocode - Bridge loads and design criteria - NCCI 1, Liikenneviraston (Finnish Transport Agency), Helsinki.**

For more information see **LO 24/2014 Optional Code Settings**

Norway - NS EN1991-2.2004 NA 2010 + NA-rundskriv 07-2015

When the country name 'Norway' and Design Code 'EN1991-2 Norway 2010 + NA-rundskriv 07-2015' is selected on the Vehicle Load Optimisation dialog, traffic loading can now be generated according to the following:

- **NS-EN 1991-2.2003 NA 2010 Eurokode 1: Laster på konstruksjoner - Del 2: Trafikklast på bruer**
- **NS EN 1990-2002 A1 2005 NA 2016 Eurokode: Grunnlag for prosjektering av konstruksjoner**
- **NA Circular 07/2015 Traffic Load in manual N400 Bruprosjektering State Highways Authority of Norway**

For more information see **EN1991-2 Norway 2010 + NA-rundskriv 07-2015**

United Kingdom - BD 21/01 Annexes D and E added

When country name 'UK' and Design code 'BD21/01 Annexes D and E' is selected on the Vehicle Load Optimisation dialog, main road traffic loading can now be generated to:

- **BD21/01 The Assessment Of Highway Bridges And Structures Design Manual for Roads and Bridges, Volume 3, Section 4, Part 3, Highways Agency, May 2001**

For more information see **UK BD21/01 Annexes D and E Optional Code Settings**

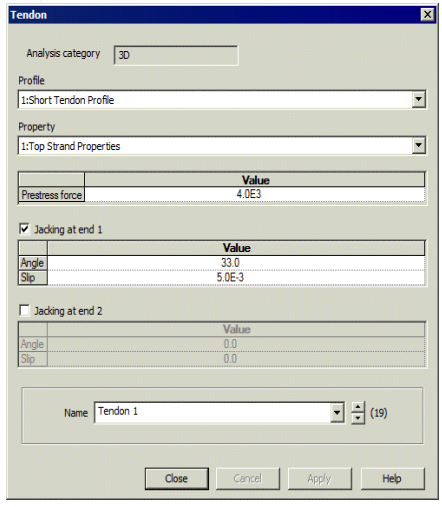
Prestress improvements

Prestress modelling restructured and easier to use

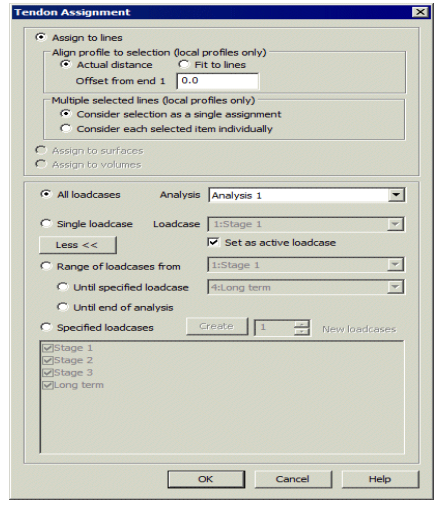
The single tendon and multiple tendon prestress wizards that in previous versions were used to generate prestress loading have been retired.

Instead, a **tendon loading** attribute, which references **tendon profile** and **tendon property** data, is defined having a specified prestress force and associated jacking and loss details. Once defined, this tendon loading attribute is added to the Attributes Treeview, where it can be directly assigned to selected line, surface or volume features in a model consistently with all other attribute assignments made within LUSAS.

Tendon loading assignment options state how the tendon loading is to be applied and, for assignment to lines, mapping and offset options can be used, as well as specifying in which loadcase(s) the tendon loading should apply.



Tendon loading attribute dialog



Tendon loading attribute assignment dialog

Other improvements

Prestress loading is now calculated as a part of solving a model. When solved, LUSAS still calculates equivalent nodal loading from the assigned prestress tendon loads and assigns these forces automatically to selected features (and hence nodes and elements) of the model for a specified loadcase. The discrete loads and the applied forces / moments calculated are not created as attributes in the treeview but can be reproduced / viewed for checking purposes. See **viewing prestress loading data.**

The properties, profile, loading, losses and setting-out data for all tendons defined and assigned in a model can now be reported upon by adding a **prestress chapter** to a model report.

Graphs showing prestress losses in tendons are now produced using the **graph wizard**.

Editing of tendon loading, property and profile data has been improved and can be done by individual selection of one or more attributes, or by selection of the parent folder containing a set of attributes. See **Viewing and Editing of Multiple Attribute Assignments** and **Viewing and Editing of Multiple Load Attributes**.

Pre-version 17 models

When opening pre-version 17 models containing single and multiple tendon generated tendon data note that:

- Single tendon prestress wizard generated data is not converted automatically to the latest tendon modelling data structures. Tendon properties, profiles and discrete load data generated by the wizard is retained.
- Multiple tendon prestress wizard generated data is converted automatically to the latest tendon modelling data structures, with tendon properties, profiles and loadings all being updated and assigned to the correct features. Discrete load and search area data attributes that previously revealed the underlying implementation no longer appear in the Attributes Treeview. However, once the model is solved the discrete load data calculated can be viewed as described previously.

For more information see **Prestress Loading**

Prestress loading and editing is further enhanced by general viewing and editing enhancements made. See **Viewing and Editing of Multiple Attribute Assignments** and **Viewing and Editing of Multiple Load Attributes** and **Properties panel** for details.

Prestress loading now supported by plain stress, plane strain, shell and solid elements

The prestress loading capability has been extended to allow assignment to surfaces modelled with plane stress, plane strain and shell elements, and volumes modelled with solid elements. Assignment of tendon loads to a surface or volume modelled with these elements requires a tendon profile to be defined using 'Global coordinates' on the Tendon Profile dialog. The defined tendon length is used and the tendon is positioned in the model at the start coordinate specified within the tendon profile dataset.

For more information see **Defining a Tendon Profile** and **Tendon Loading Assignment (Multi-tendon)**

Material models

AASHTO LRFD 7th edition creep and shrinkage material model added

The AASHTO LRFD 7th edition creep and shrinkage model can be used with beam elements, 2D/3D continuum elements, semi-loof and thick shells, and composite solid elements. AASHTO creep and shrinkage is only strictly applicable to beams, however, in LUSAS the creep equations have been extended to 2D and 3D stress states. The assumptions made in the derivation of this extension can be found in the LUSAS Theory Manual.

- Creep loss can be calculated with reference to either article 5.4.2.3.2 or the commentary clause C5.4.2.3.2-2.
- Shrinkage loss can be calculated with reference to equation 5.4.2.3.2-2 or the commentary clause C5.4.2.3.2-2

For more information see **Concrete Creep and Shrinkage to AASHTO LRFD 7th Edition**

IRC112-2011 creep and shrinkage material model added

The IRC:112-2011 creep and shrinkage model can be used with beam elements, 2/3D continuum elements, semi-loof and thick shells, and composite solid elements. IRC:112-2011 creep and shrinkage is only strictly applicable to beams, however, in LUSAS the creep equations have been extended to 2D and 3D stress states. The assumptions made in the derivation of this extension can be found in the LUSAS Theory Manual.

- Creep calculations are carried out to IRC:112-2011 clause 6.4.2.7 and Annexure A2, but elevated or reduced temperatures during curing are not considered.
- Shrinkage effects can be included in accordance with clause 6.4.2.6 and Annexure A2 clause A2.6.

For more information see **Concrete Creep and Shrinkage to IRC:112-2011**

Duncan-Chang soil material model added

The Duncan-Chang soil material model is a simple model whose parameters are calculated by curve fitting triaxial test data over a range of cell pressures. It takes an incremental elastic approach in which Young's modulus and Poisson's ratio are evaluated from current stresses. A predictor step is used to determine whether the soil

is loading or unloading, following which, corresponding values of Young's modulus and Poisson's ratio are calculated for use in the incremental elastic modulus matrix.

The model can be used with standard continuum elements as well as the two-phase elements. The modelling parameters required can be evaluated in standard laboratory tests.

For more information see **Duncan-Chang Model**

Matrix properties for joint elements

The definition of symmetrical matrix properties (stiffness, mass or damping) to specify a joint representing a pilecap sitting on a group of piles in a global (3D) model is now supported. This is achieved by selecting the **Attributes > Material > Joint** menu item and choosing the **Matrix properties** option.

	u1	v1	w1	THx1	THy1	THz1	u2	v2	w2	THx2	THy2
u1	K1				K8		-K1				-K8
v1		K2		K7				-K2		-K7	
w1			K3						-K3		
THx1		K7		K4				-K7		-K4	
THy1	K8				K5		-K8				-K5
THz1						K6					
u2	-K1				-K8		K1				K8
v2		-K2		-K7				K2		K7	
w2			-K3						K3		
THx2		-K7		-K4				K7		K4	
THy2	-K8				-K5		K8				K5
THz2						-K6					

Data entry for a symmetrical matrix

For more information see **Matrix properties for joint elements**

Initialisation of ground conditions (K_0)

K_0 data can now be defined for material models used in soil analyses, principally, elastic models, **Modified Mohr-Coulomb**, **Duncan-Chang** and **Modified Cam-clay**. Both the Duncan-Chang and Modified Cam-clay models need the definition of an initial stress state to calculate an initial stiffness.

The initial stresses in the soil can be calculated by providing K_0 data. On element activation, the effective vertical stresses in the soil are calculated from the soil weight and any additional loads such as overburden. Horizontal stresses are then calculated using the K_0 value.

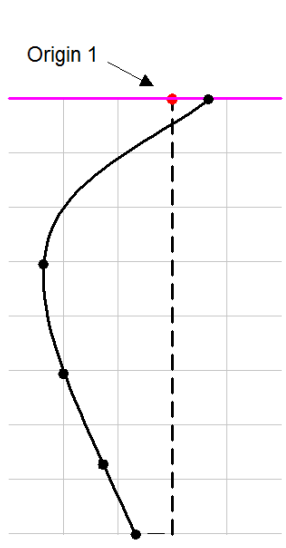
For more information see **K_0 Initialisation**

Profile variations

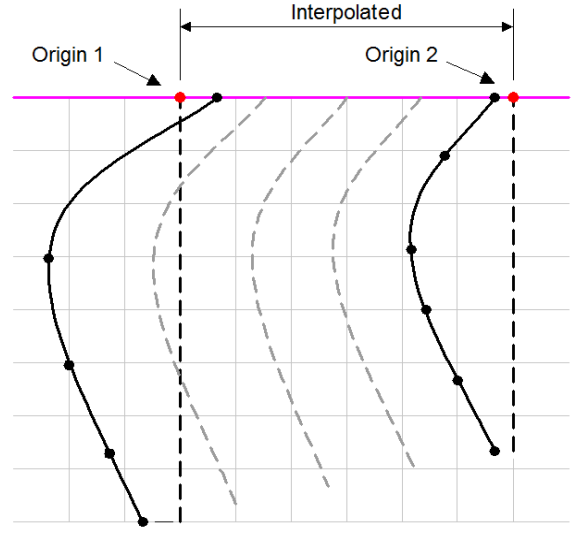
Modelling of the variation of material properties at any location in a model can now be achieved by defining a profile variation. These are defined by selecting the **Utilities > Variation** menu item. A typical use is modelling the variation of soil properties with depth, as obtained by borehole data.

A single profile variation is defined by sets of values against distance, but any number of profile lines in-plane or out-of-plane may be grouped to form a profile set variation. When used to define particular material properties in a supported material model, material properties for elements in any region of a model can be calculated by LUSAS by interpolating between the distance and values defined for each element between each profile variation location.

Examples of profile variations

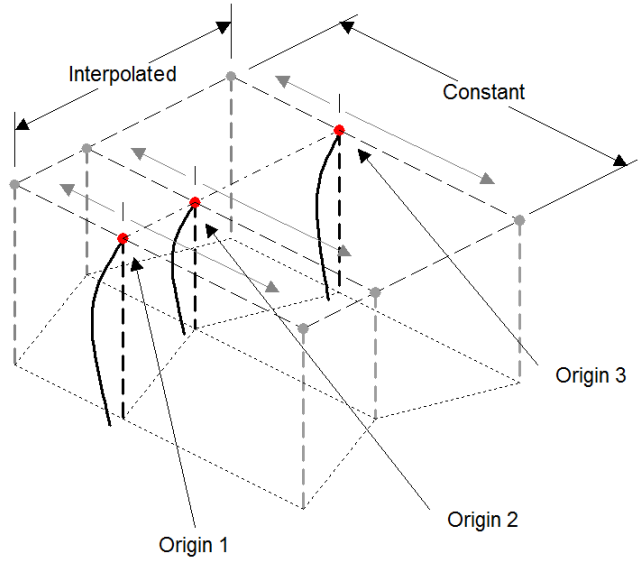


Single profile variation
(Variation with depth only)

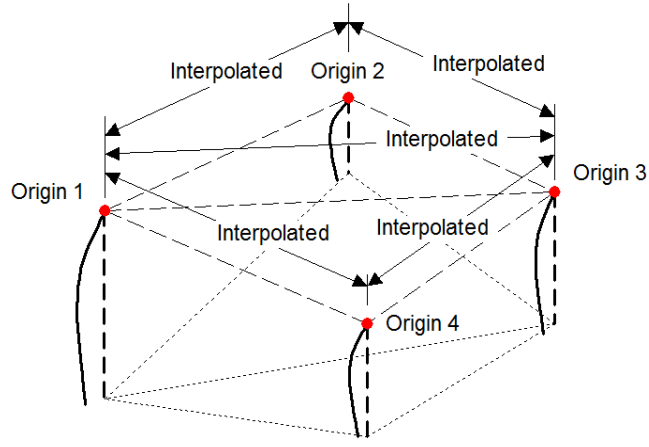


In-plane set of two (or more) profile variations
(Variation with depth interpolated in-plane only)

Muliple sets of profile variations



In-plane set of profile variations in a 3D model
(Variation with depth is only interpolated in the plane of the profile set)



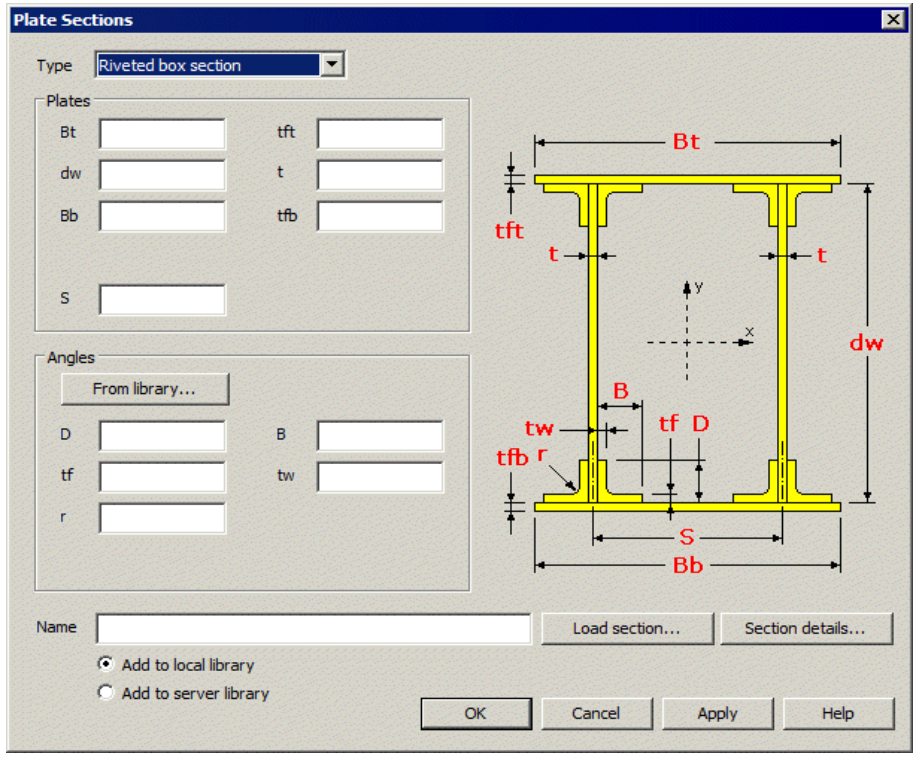
Out-of-plane set of profile variations in a 3D model
(Variation with depth is interpolated in 3 dimensions)

For more information see **Profile Variation**

General enhancements

Section property calculation of plated sections

The Plate Section Property Calculator is provided in Bridge and Civil & Structural software products only and is accessed from the **Tools> Section Property Calculator > Plate Sections...** menu item. It allows definition of section geometry for a range of riveted or welded section types, either with reference to sections held in the main section library or from user-defined values, for saving into a section library.



Section types available

The following sections are supported via the Type drop-down:

- **Riveted I section**
- **Riveted box section**
- **Riveted trough**
- **Riveted box from I sections**
- **I section with channel**
- **Riveted box from channels (back to back)**
- **Riveted box from channels (face to face)**

- **I section with doubler plates**
- **Welded plate box**
- **Riveted T section**
- **Riveted cruciform**

For more information see **Plate Section Property Calculator**

Indian steel section library added

Section properties for a range of Indian steel sections have been added. The following are supported:

- **Junior beams**
- **Light weight Beams**
- **Wide flange beams**
- **Heavy weight beams**
- **Junior channels**
- **Light weight channels**
- **Square hollow sections**
- **Rectangular hollow sections**

For more information see **Section Library**

New Zealand steel section library added

Section properties for a range of New Zealand steel sections have been added. The following are supported:

- **Universal Beams**
- **Universal Columns**
- **Universal Bearing Piles**
- **Taper Flange Beams**
- **Parallel Flange Beams**
- **Equal Angles**
- **Unequal Angles**
- **Welded Beams**
- **Welded Columns**
- **Circular Hollow Sections**
- **Square Hollow Sections**
- **Rectangular Hollow Sections**

For more information see **Section Library**

Australian taper flange beam properties updated

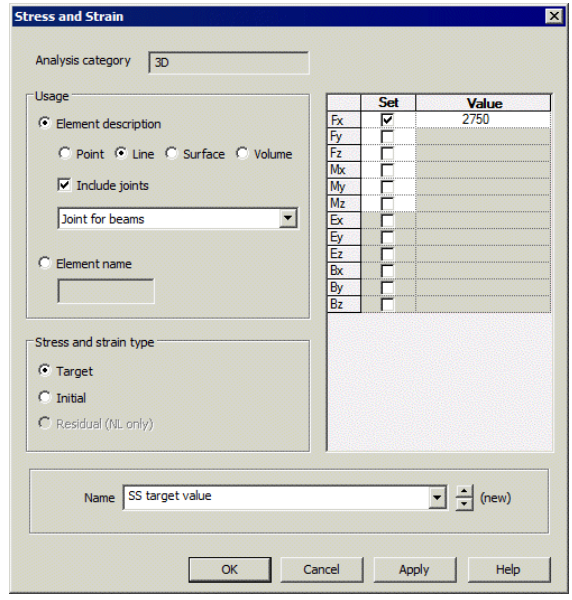
The section properties for 125TFB and 100TFB have been updated.

Stress and strain target values (for cables)

Stress and Strain loading has been enhanced to include a **Target** option. This is used to define a value for a selected results component for a feature that will be obtained exactly when a solve has taken place.

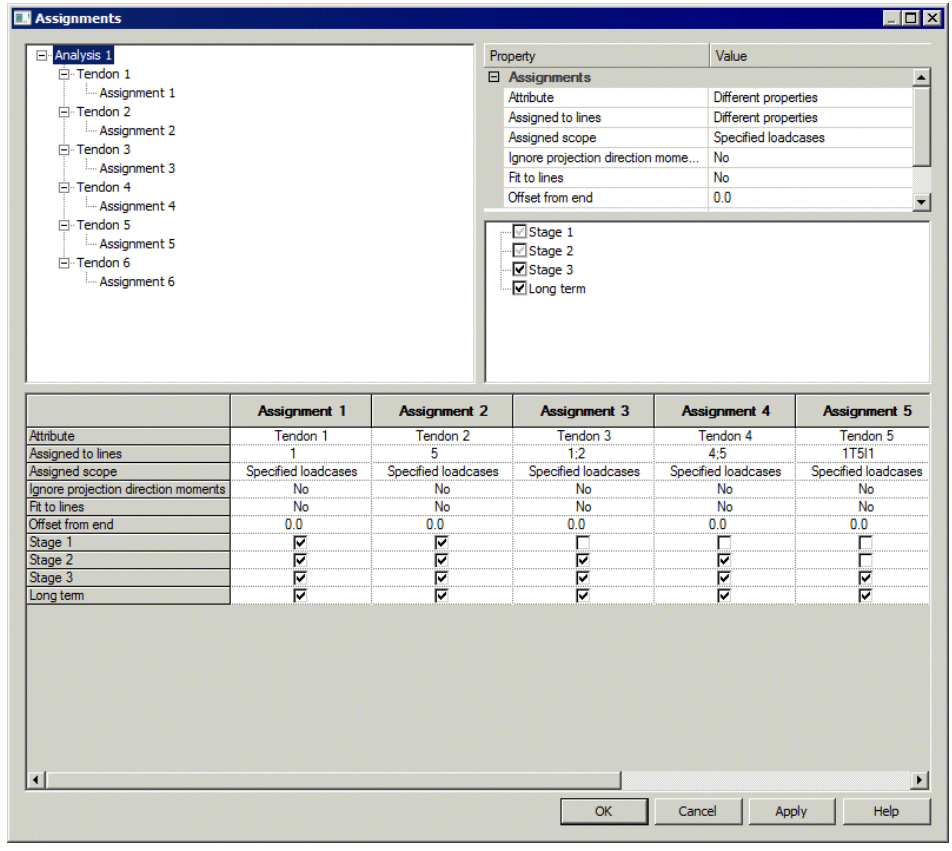
This facility is primarily for use when a specified jacking force is required to be applied to one end of a line representing a cable. Note that target values are only appropriate for linear materials.

For more information see **Structural Loads**



Improved viewing and editing of attribute assignments

The new Assignments editor is displayed when one or more attributes of the same type are selected for editing in the Attributes Treeview by using the **View Assignments** context menu item. The editor can also be accessed from any assignments that exist in the loadcases panel of the Analyses Treeview. It allows simultaneous viewing, editing and management of any number of attribute assignments. Editing of data can be by individual selection, or by selection of the parent folder containing a set of attributes. The dialog is expandable.



For more information see **Viewing and Editing of Multiple Attribute Assignments**

Improved viewing and editing of multiple load attributes

The new Attributes editor is presented for particular structural loading attributes, when two or more of the same loading type is selected for editing in the Attributes Treeview by using the **Edit** context menu item. This can be by individual selection, or by selection of the parent folder containing them (providing no other different load types are present in that folder). The dialog enables easy editing of loading data for a range of structural load types, and is of particular use when editing tendon loading values when carrying out prestress modelling.

For more information see **Viewing and Editing of Multiple Load Attributes**

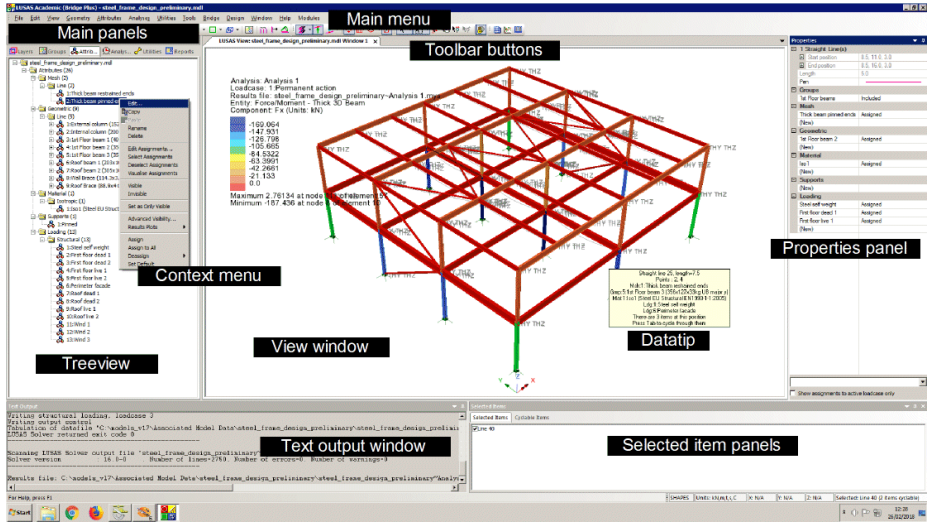
Discrete load projection over area for shell models

A new option 'Project over area (shell models)' has been added to the discrete loading assignment dialog. With this option, the offset distance from the load definition position to the load position in the search area is used to calculate a moment when the loading is assigned to shell elements.

For more information see **Discrete Structural Loads**

New Properties panel

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



For more information see **Properties panel**

New Hierarchy panel

The Hierarchy panel lists the higher and lower order features/objects of the currently selected single geometry feature or mesh object. Selection of a feature within the hierarchy panel will update the panel to show the hierarchy for that feature.

For more information see **Hierarchy panel**

Changes to the Element Library

No new elements have been introduced in this release.

The capabilities of particular elements have been enhanced to reflect relevant enhancements described elsewhere for this release. For example, certain elements now support additional creep and shrinkage codes.

User change requests

In addition to the range of new facilities and improvements listed, many user change requests have also been implemented. The originators of all requested changes to the software (some of which are included in the above list of enhancements) that have been incorporated in this release will be notified individually.

Documentation

User manuals

All online and printed documentation has been updated for this new release. Manuals are provided in PDF format as part of any Version 16 installation, and are also available for download from the LUSAS website.

Updated Worked Examples

The following examples have been updated to show the use of selected new facilities added in this release.

- **Linear analysis of a post tensioned bridge** - Updated to show the use of the restructured prestress modelling facilities.
- **Segmental construction of a post tensioned bridge** - Updated to show the use of the restructured prestress modelling facilities.
- **Drained nonlinear analysis of a retaining wall** - Updated to show use of profile variations and initialisation of soils using Ko.

All other examples have been updated to ensure that the examples match changes made to the software.

Individual worked examples in PDF format are provided as part of the LUSAS software download file or release CD, as well as being available from the LUSAS User Area.

Other

Potential issues opening PDF files referenced in CHM files

On some PCs, and for certain operating systems, the installation of security updates as released by Microsoft can affect the opening of PDF files from the table of contents panel within the CHM file-based help. Any links to PDF files from within help topic pages may similarly be affected.

If problems are found when attempting to open these files from within the online CHM file supplied please note the following:

- Selected manuals are supplied in PDF format on the installation kit and these are normally installed into the <LUSAS Installation Folder>/Programs/PDF_Manuals folder.
- Workarounds/solutions may be provided by Microsoft during the availability and support of this particular LUSAS software release.

Previous new facilities and improvements in this release

None, this is the first release of Version 17.

LUSAS

LUSAS, Forge House, 66 High Street, Kingston upon Thames, Surrey, KT1 1HN, UK
Tel: +44 (0)20 8541 1999 | Fax: +44 (0)20 8549 9399 | Email: info@lusas.com | www.lusas.com