LUSAS

New features and improvements in Version 16.0

New features / improvements

LUSAS Version 16.0

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 Features and Improvements in Version 16.0

Overview

Version 16.0 of LUSAS is a major software release that sees the introduction of steel frame design checking capabilities into LUSAS Modeller and a host of other new features and enhancements to make collaboration between staff easier, and to improve general usability.

In addition to steel frame design. other design code-related improvements and enhancements include those for reinforced concrete slab design, traffic load optimisation, design load combinations, and response spectra for various design codes. New worked examples are provided to show how the new design checking and design load combinations facilities are used.

Analysis-related improvements include the ability to define a cable tuning loadcase in a nonlinear analysis, the introduction of p-delta analysis capabilities, time-dependent prestress, and eigenvalue buckling of stressed structures. A time management facility allows for easy adjustment of the duration of pre-defined construction stages in a staged construction analysis.

Of many new results processing features, inspection locations, which obtain results for user-defined positions of interest anywhere on a model are of particular note, along with enhanced averaging of results, and improvements to graphing, section slicing and combinations. For printed results, the Print Results Wizard dialog is now used to add chapters of selected loadcase results to a model report.

For general modelling, model analysis categories have been introduced. Selection of an analysis category simplifies the user interface to generally only show those menu items and options that are appropriate to the type of model being defined - all helping to enhance the modelling experience and adjust the user interface based on the analysis category. Models can now be merged together allowing two or more people to create separate models of specific

parts of a structure simultaneously. BIM import / export is now provided. Section property calculation now supports compound sections, infilled and encased sections, and thin walled arbitrary sections defined solely by lines. Perspective viewing has been introduced, and layout grids and new point, line and surface drawing options speed-up the creation of frame models. Temperature dependent material properties can be defined in LUSAS Modeller for all materials.

A range of new 2D and 3D beam elements that provide more functionality have been introduced, with some becoming the default elements of choice for beam and frame analysis.

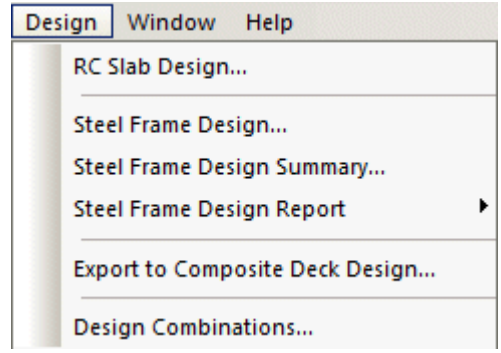
An updated user interface provides a more modern look and feel, provides tabs for window selection panels, easier ways to interact with the view windows, and permits multiple selection of treeview data for assignment or deletion purposes.

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.

Design-related improvements

New Design Menu

A new Design menu has been introduced for LUSAS Bridge and LUSAS Civil & Structural software products. Currently it lists menu items for RC Slab Design, Steel Frame Design, Export to Steel/Composite Frame Design, and Design Combinations.



Steel Frame Design menu items will be omitted if a model **analysis category** of 3D is not in use, and menu items will be 'greyed-out' if your software licence key does not currently support them.

Please contact your account manager if you wish to have access to any licensed options, and are unable to do so.

For more information see **Application Manual (Bridge, Civil and Structural) - Introduction**

Steel Frame Design

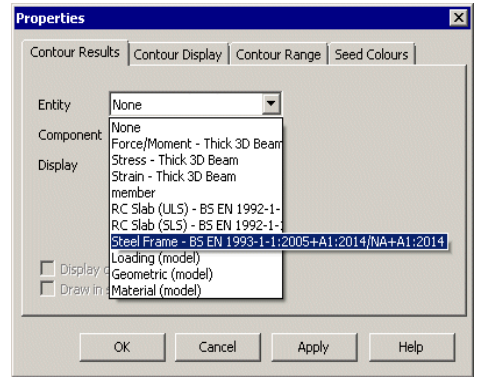
Design code checking of steel members can now be easily carried out in LUSAS Modeller. This is performed as a results processing operation following the solving of a model, and the assignment of **Steel frame design attributes** to lines that represent steel members, in order to provide design related information for a specified design code.

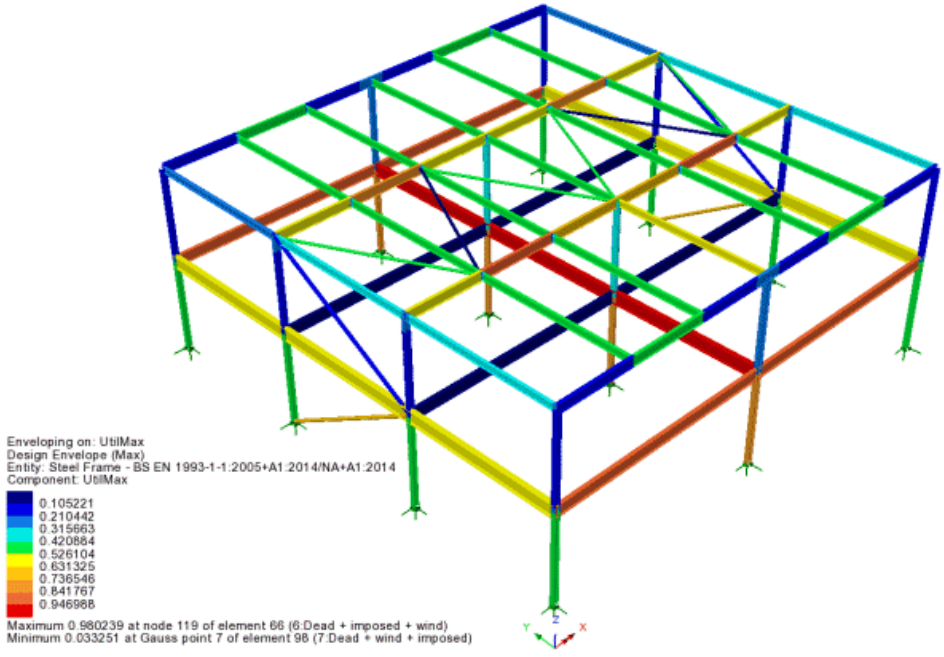
The following design codes are currently supported:

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

Design check results are visualised as Utilisation ratios on a results viewing layer for a selected design code, and active loadcase, load combination or envelope.

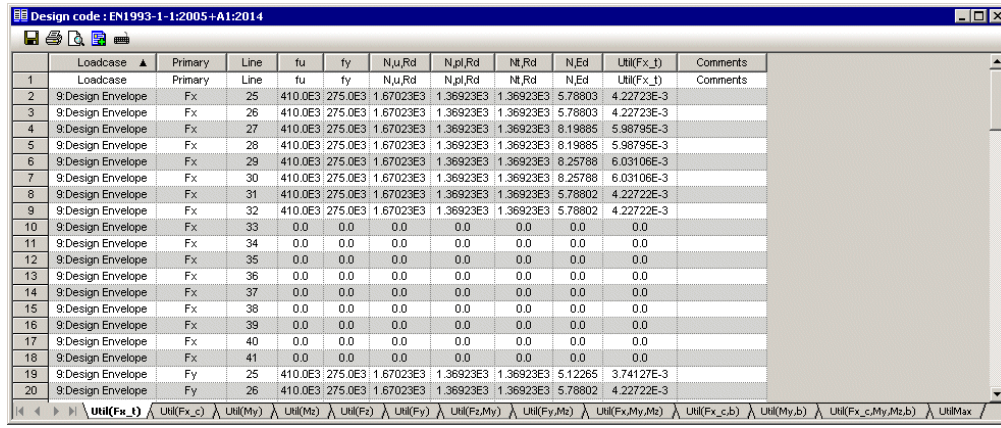
Results components for individual design checks can be chosen for viewing, and maximum utilisation factors can also be obtained.





Design summary reports

A tabular summary of design check results can be produced for selected members and loadcases. Results may be saved for use with Microsoft Excel or saved to a text format. Results can also be added to a model report, and each time the main report is generated the design summary data will be updated to match the current state of the model.



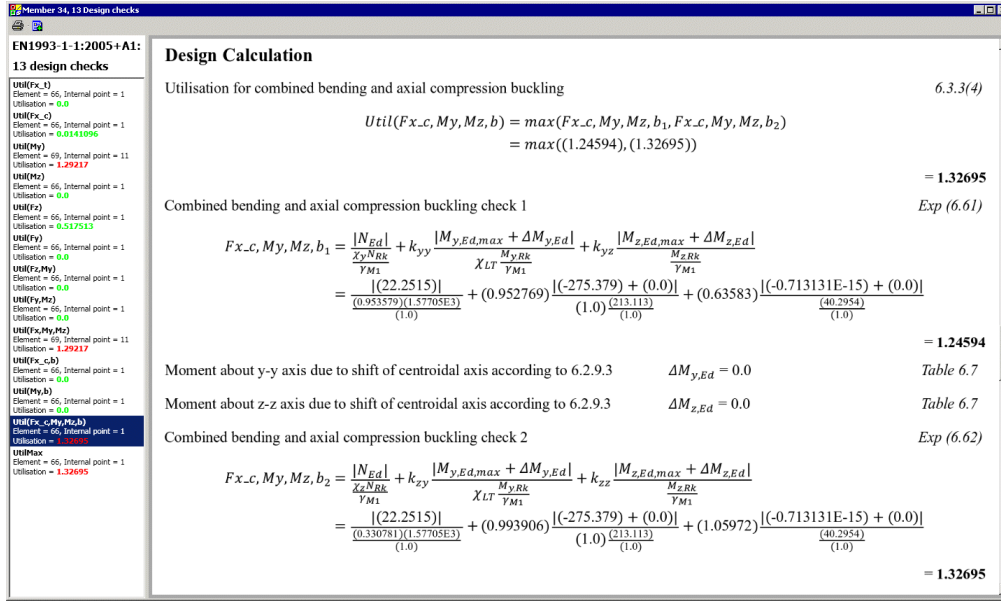
Design code : EN1993-1-1:2005+A1:2014

	Loadcase	Primary	Line	fu	fy	N,u,Rd	N,pl,Rd	Nt,Rd	N,Ed	Util(Fx_t)	Comments
1	Loadcase	Primary	Line	fu	fy	N,u,Rd	N,pl,Rd	Nt,Rd	N,Ed	Util(Fx_t)	Comments
2	9:Design Envelope	Fx	25	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78803	4.22723E-3	
3	9:Design Envelope	Fx	26	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78803	4.22723E-3	
4	9:Design Envelope	Fx	27	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.19885	5.98795E-3	
5	9:Design Envelope	Fx	28	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.19885	5.98795E-3	
6	9:Design Envelope	Fx	29	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.25788	6.03106E-3	
7	9:Design Envelope	Fx	30	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	8.25788	6.03106E-3	
8	9:Design Envelope	Fx	31	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78802	4.22722E-3	
9	9:Design Envelope	Fx	32	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78802	4.22722E-3	
10	9:Design Envelope	Fx	33	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
11	9:Design Envelope	Fx	34	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
12	9:Design Envelope	Fx	35	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
13	9:Design Envelope	Fx	36	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
14	9:Design Envelope	Fx	37	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
15	9:Design Envelope	Fx	38	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
16	9:Design Envelope	Fx	39	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
17	9:Design Envelope	Fx	40	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
18	9:Design Envelope	Fx	41	0.0	0.0	0.0	0.0	0.0	0.0	0.0	
19	9:Design Envelope	Fy	25	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.12265	3.74127E-3	
20	9:Design Envelope	Fy	26	410.0E3	275.0E3	1.67023E3	1.36923E3	1.36923E3	5.78802	4.22722E-3	

Util(Fx_t) | Util(Fx_c) | Util(My) | Util(Mz) | Util(Fz) | Util(Fy) | Util(Fz,My) | Util(Fy,Mz) | Util(Fx,My,Mz) | Util(Fx_c,b) | Util(My,b) | Util(Fx_c,My,Mz,b) | UtilMax

Formatted member reports

To investigate the design calculations carried out for particular members, **formatted design reports** can be generated, showing the calculations made and referencing clauses and equations from the code. A formatted design report for the selected member can be added to a model report.



Member 34, 13 Design checks

EN1993-1-1:2005+A1:
13 design checks

Util(Fx_t)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(Fx_c)
Element = 66, Internal point = 1
Utilisation = 0.0141096

Util(My)
Element = 69, Internal point = 11
Utilisation = 1.29217

Util(Mz)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(Fz)
Element = 66, Internal point = 1
Utilisation = 0.517513

Util(Fy)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(Fz,My)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(Fy,Mz)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(Fx,My,Mz)
Element = 69, Internal point = 11
Utilisation = 1.29217

Util(Fx_c,b)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(My,b)
Element = 66, Internal point = 1
Utilisation = 0.0

Util(Fx_c,My,Mz,b)
Element = 66, Internal point = 1
Utilisation = 1.32695

UtilMax
Element = 66, Internal point = 1
Utilisation = 1.32695

Design Calculation

Utilisation for combined bending and axial compression buckling *6.3.3(4)*

$$Util(Fx\_c, My, Mz, b) = max(Fx\_c, My, Mz, b_1, Fx\_c, My, Mz, b_2)$$
$$= max((1.24594), (1.32695))$$

= **1.32695**

Combined bending and axial compression buckling check 1 *Exp (6.61)*

$$Fx\_c, My, Mz, b_1 = \frac{|N_{Ed}|}{\frac{\chi_y N_{Rk}}{\gamma_{M1}}} + k_{yy}\frac{|M_{y,Ed,max} + \Delta M_{y,Ed}|}{\chi_{LT}\frac{M_{y,Rk}}{\gamma_{M1}}} + k_{yz}\frac{|M_{z,Ed,max} + \Delta M_{z,Ed}|}{\frac{M_{z,Rk}}{\gamma_{M1}}}$$
$$= \frac{|(22.2515)|}{\frac{(0.953579)(1.57705E3)}{(1.0)}} + (0.952769)\frac{|(-275.379) + (0.0)|}{(1.0)\frac{(213.113)}{(1.0)}} + (0.63583)\frac{|(-0.713131E\text{-}15) + (0.0)|}{\frac{(40.2954)}{(1.0)}}$$

= **1.24594**

Moment about y-y axis due to shift of centroidal axis according to 6.2.9.3 $\Delta M_{y,Ed} = 0.0$ *Table 6.7*

Moment about z-z axis due to shift of centroidal axis according to 6.2.9.3 $\Delta M_{z,Ed} = 0.0$ *Table 6.7*

Combined bending and axial compression buckling check 2 *Exp (6.62)*

$$Fx\_c, My, Mz, b_2 = \frac{|N_{Ed}|}{\frac{\chi_z N_{Rk}}{\gamma_{M1}}} + k_{zy}\frac{|M_{y,Ed,max} + \Delta M_{y,Ed}|}{\chi_{LT}\frac{M_{y,Rk}}{\gamma_{M1}}} + k_{zz}\frac{|M_{z,Ed,max} + \Delta M_{z,Ed}|}{\frac{M_{z,Rk}}{\gamma_{M1}}}$$
$$= \frac{|(22.2515)|}{\frac{(0.330781)(1.57705E3)}{(1.0)}} + (0.993906)\frac{|(-275.379) + (0.0)|}{(1.0)\frac{(213.113)}{(1.0)}} + (1.05972)\frac{|(-0.713131E\text{-}15) + (0.0)|}{\frac{(40.2954)}{(1.0)}}$$

= **1.32695**

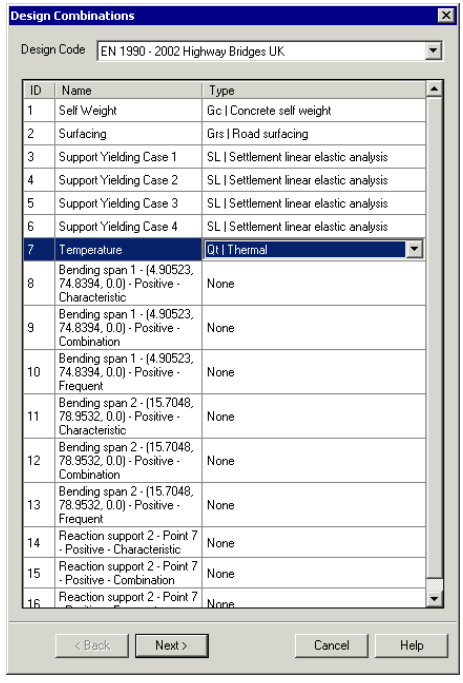
For more information see **Steel Frame Design - Code Checking**

New worked example

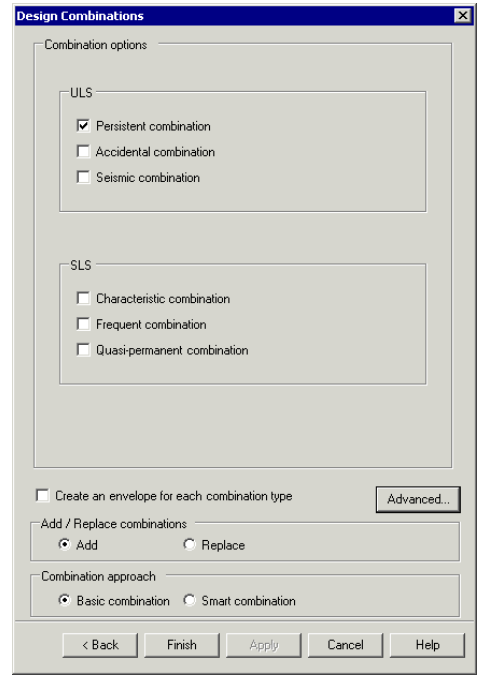
- **Steel Frame Design to EN 1993-1-1** shows the steps required to prepare a model for a steel frame design check in accordance with EN 1993-1-1, and viewing results of member utilisation both as contour plots and as fully annotated design reports.

Design Code Load Combinations

The new Design Combination wizard is used to assign a loadtype to a loadcase, envelope or basic combination for a supported design code. Based on the assignments and settings made, load combinations for those load types are automatically generated by LUSAS Modeller, rather than having to be user-defined one-by-one.



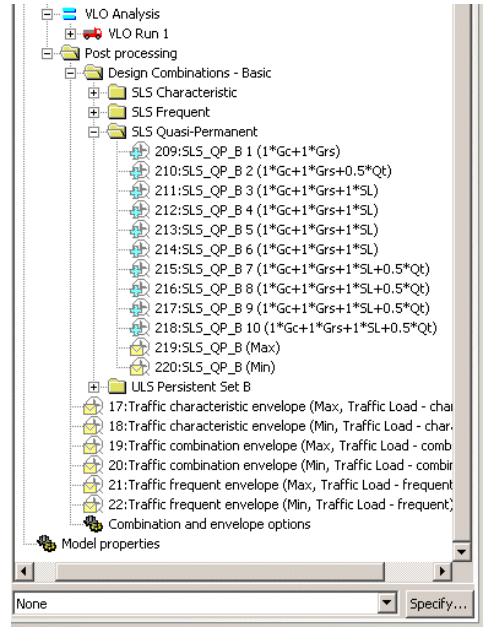
Loadcase / loadtype definition



Combination options

Design combinations for the following codes of practice are supported.

- **AASHTO 7th Edition**
- **AS/NZS 1170**
- **BD21/01**
- **BD37/01**
- **CSA-S6-14**
- **EN1990 (Buildings) Recommended Values**
- **EN1990 (Buildings) to Irish National Annex**
- **EN1990 (Buildings) to UK National Annex**
- **EN1990 (Bridges) Recommended Values**
- **EN1990 (Bridges) to Irish National Annex**
- **EN1990 (Bridges) to UK National Annex**
- **GB 50009 - 2012**
- **JTG D60-2004**



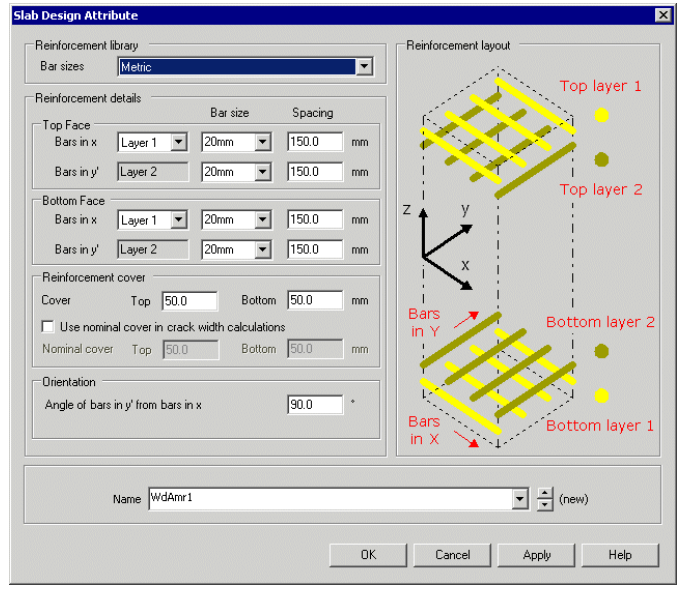
Design combinations created

For more information see **Design Combinations**

New worked example

- **Bridge Design Load Combinations to EN1990** shows the use of the new design load combinations wizard to generate factored design combinations from codes of practice.

RC Slab design with bending and in-plane effects



The reinforced concrete slab design facility has been improved to consider in-plane effects for structures modelled using shell elements. It supports plotting of contours and values that indicate flexural reinforcement requirements at Ultimate Limit State (ULS) or design crack width at Serviceability Limit State (SLS) for design codes that support this. Calculations now carried out are for reinforced concrete slabs (without prestressing) that are modelled using plate or shell elements. For ULS, both bending only (using Wood-Armer), and bending and in-plane effects (using Clark-Nielsen) can be considered. For SLS, both bending-only (using principal moments), and bending and in-plane effects (using principal stresses) can be considered, if supported by a chosen design code.

Reinforcement arrangements are now defined using **RC slab design attributes**. Multiple slab design attributes may be defined and assigned to relevant features on a model.

Values and results contours for a chosen results component can be displayed for a stated slab face and for a previously chosen Code of Practice using standard values and contour plotting facilities.

For more information see **RC Slab Design: Overview**

RC Slab design to AASHTO LRFD 7th

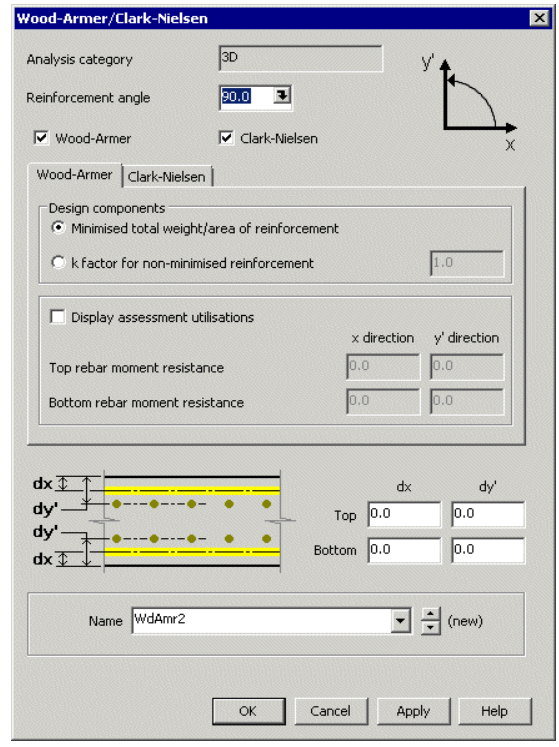
The RC slab design facility now supports **AASHTO LRFD Bridge Design Specifications, 7th Edition**, American Association of State Highway and Transportation Officials, 2014.

For more information see **RC Slab Design: Design Code Settings for AASHTO LRFD**

Wood-Armer attribute

The introduction of Wood-Armer attributes simplifies the specification of slab reinforcement arrangements. When assigned to features in a model, they permit calculation of top and bottom reinforcement moments for shell, plate and grillage elements, and top and bottom reinforcement forces for shell elements only according to Wood-Armer / Clark-Nielsen theory. Multiple Wood-Armer attributes may be defined and assigned to relevant regions and features on a model.

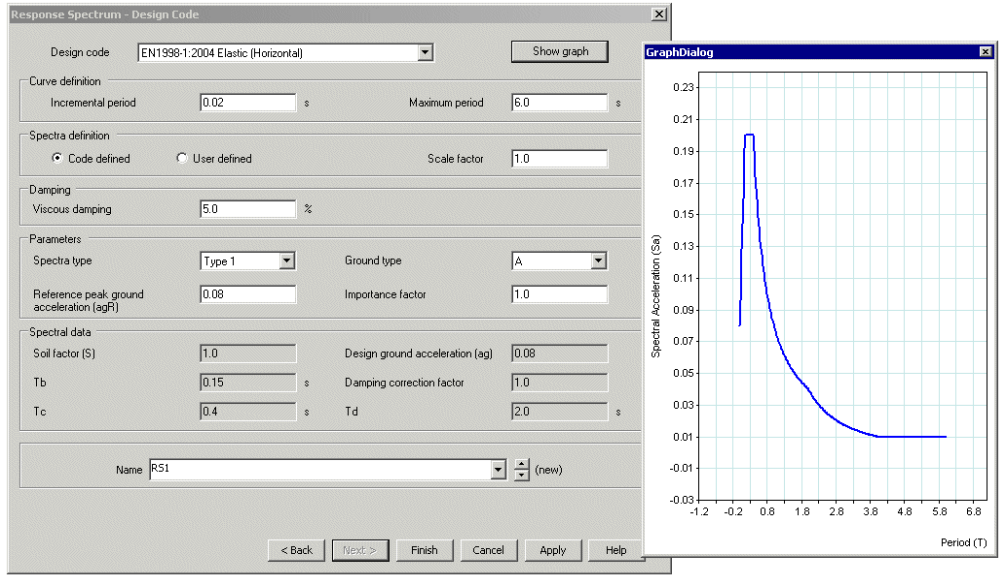
For more information see **Wood-Armer / Clark-Nielsen Attributes**



Response spectra provided for numerous design codes

Design code response spectrum data for use with IMD loadcases can be defined by selecting the general **Utilities > Response Spectra** menu item. The following codes of practice are currently supported:

- **ASCE 7-10 (2010)**
- **ATC-40 (1996)**
- **China GB 50011-2010**
- **EN1998-1:2004 Design (Horizontal)**
- **EN1998-1:2004 Elastic (Horizontal)**
- **FEMA 356 (2000)**



For more information see **Response Spectrum - Code Based (For IMD loadcase use)**

Material library enhanced

Supplied library materials are now accessed by choosing a material type, for a region or country (when applicable), for design code or standard (when applicable), and for a grade of material (when applicable). Defined material can now be named explicitly, rather than have just a system generated name assigned to it, as in previous versions.

For some materials in some regions a user-defined grade can also be specified. An example of this would be for Concrete, USA, AASHTO LRFD 7th, where a compressive strength, weight, aggregate correction factor, and cement type can be defined.

For more information see **Material Library**

LUSAS Traffic load optimisation improvements

China JTG D60-2015

When the country name 'China' and Design Code 'JTG D60-2015' is selected on the Vehicle Load Optimisation dialog, traffic loading can now be generated according to China standards:

- JTG D60-2015 General Code for Design of Highways Bridges and Culverts, Chapter 4: Actions.

For more information see **China JTG D60-2015 Optional Code Settings**

Denmark - DS/EN 1991-2 DK NA:2015

When the country name 'Denmark' and Design Code 'EN1991-2 Denmark 2015' is selected on the Vehicle Load Optimisation dialog, traffic loading can now be generated according to Denmark standards:

- EN 1991-2 DK NA:2015 DS/EN 1990/A1 DK NA:2015 National Annex to Eurocode 0: Basis of structural design, Annex A2 Applications for Bridges, BaneDanmark/ Vejdirektoratet, 27 Apr 2015.
- DS/EN 1991-2 DK NA:2015 National Annex to Eurocode 1: Actions on structures – Part 2: Traffic load on bridges, BaneDanmark/ Vejdirektoratet, 1 May 2015
- Annex A (Normative) Load Models for Classification and Assessment of Load-carrying Capacity, BaneDanmark/ Vejdirektoratet,

For more information see **EN1991-2 Optional Code Settings - Denmark**

Saudi Arabia - MOMRA Bridges Design Specifications

When the country name 'Saudi Arabia' and the Design code 'MOMRA', is selected on the main Vehicle Load Optimisation dialog, road traffic loading data and parameters can now be specified with reference to MOMRA Bridges Design Specifications.

For more information see **MOMRA Optional Code Settings**

Sweden TDOK 2013_0267 (NATO Vehicles)

When the country name 'Sweden' and Design code 'TDOK 2013:0267 Military Vehicles' is selected on the main Vehicle Load Optimisation dialog, vehicle loading can now be generated to aid with military load classification of bridges with reference to clauses 2.2.1.5 and 1.3 of TDOK 2013:0267 Version 3.0 - Bärighetsberäkning av broar. UHabb January 2016.

For more information see **TDOK 2013:0267 Military Vehicles Optional Code Settings**

United Kingdom - BS5400-2:1978 loading added

When the country name 'UK' and Design code “BS5400-2:1978” is selected on the Vehicle Load Optimisation dialog, main road traffic loading can now be generated to BS5400-2:1978 Steel, concrete and composite bridges Part 2: Specification for Loads, incorporating Amendment No 4209. British Standards Institution, March 1983.

For more information see **UK BS5400-2:1978 Implementation Notes**

United Kingdom - BD37/01 loading added

When country “UK” and Design code 'BD37/01' is selected on the main Vehicle Load Optimisation dialog, road traffic loading can now be generated with reference to BD37/01.

For more information see **UK BD37/01 Implementation Notes**

United States of America - AASHTO LRFD 7th Edition: Additional State Implementations

When country “USA" and Design code “AASHTO LRFD 7th Ed. - <State>" is selected on the main Vehicle Load Optimisation dialog, road traffic loading can now be generated with reference to the following new implementations:

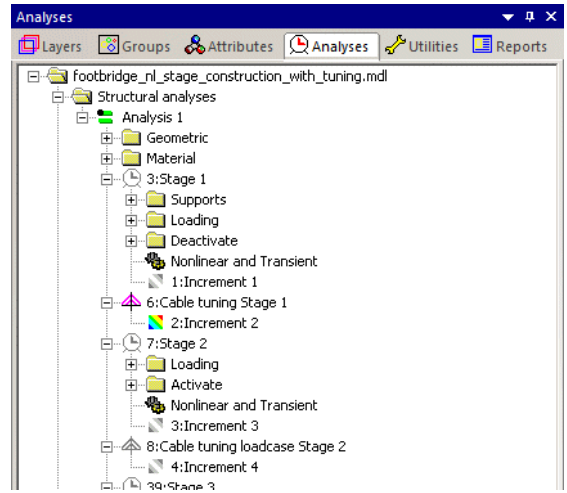
- Indiana - Indiana Design Manual (2013), Indiana Department of Transportation. The ability to choose state-specific Permit/Owner-specified vehicles is included on the Optional Code Settings dialog.
- Louisiana - LRFD Bridge Design Manual No 2 (June 2016), Department of Transportation and Development, State of Louisiana. The option to apply or not apply magnification factor (LADV-11 loading) is allowed for. When applied, an extra column in the influence dialog is provided to input the magnification factor (MF) for each influence.
- Maine - MaineDOT Bridge Design Guide (March 2014). The ability to choose between “Maine modified live load” or “Normal HL-93 live load”, for the Strength I design case is allowed for on the Optional Code Settings dialog.
- Minnesota - LRFD Bridge Design (May 2016) Minnesota Department of Transportation. The ability to choose between 125% or 110% of HL-93 is provided in the Optional Code Settings dialog. Also the option of 90% of HL-93 (general AASHTO) is given.
- Nevada - NDOT Structures Manual (May 2014). The ability to choose state-specific Permit/Owner-specified vehicles is included on the Optional Code Settings dialog.
- New York - NYSDOT LRFD Bridge Design Specifications, September 2011.

For more information see **State Implementations of AASHTO LRFD (7th Edition)**

Analysis-related improvements

Nonlinear cable tuning loadcase

The linear cable tuning analysis facility (which calculates load factors for selected lines in a model that represent cables in order to achieve defined target values for various results components) has been supplemented by a nonlinear cable tuning loadcase option. This can be configured by selecting the **Analyses> Nonlinear Cable Tuning Loadcase** menu item.



Nonlinear cable tuning is for use in nonlinear analyses where geometric, material or boundary condition nonlinearity may exist. It is also used for obtaining an equilibrium state of stressed geometry in existing structures when nonlinear effects are significant.

Unlike linear cable tuning, loadcases do not have to be explicitly selected to be included in a nonlinear cable tuning analysis. Instead, the position of a nonlinear cable tuning loadcase within the Analyses Treeview will dictate which preceding loadcase is used to provide loading data for the cable tuning analysis.

For more information see **Nonlinear Cable Tuning Loadcase**

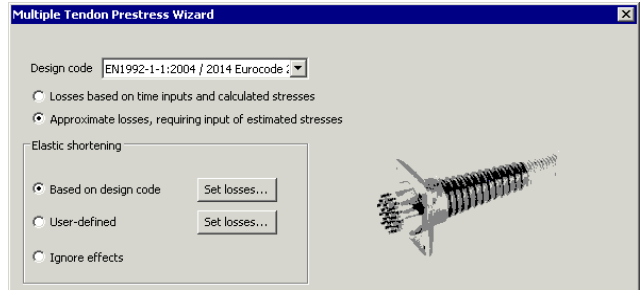
P-Delta analysis

P-Delta analysis has now been implemented for bar, beam, thick and thin shell, and 2D and 3D continuum elements with GNL capability. P-Delta analysis is an approximate geometrically nonlinear (GNL) analysis typically used to take account of the interaction between vertical and horizontal (sway) loading on tall, slender buildings. Vertical constant loads (usually dead loads) are used to form the geometric stiffness (stress stiffened) matrix for the structure; additional live load cases can then be applied and load combinations used to capture the effects of the interaction between lateral and vertical loading.

For more information see **Nonlinear Analysis Control Options** and **About Nonlinear Analysis.**

Time dependent prestress

The multi tendon prestress wizard has been updated for the AASHTO and Eurocodes to provide an option to allow for producing details and results of losses at any time, and for any stage of construction, for supported design codes.



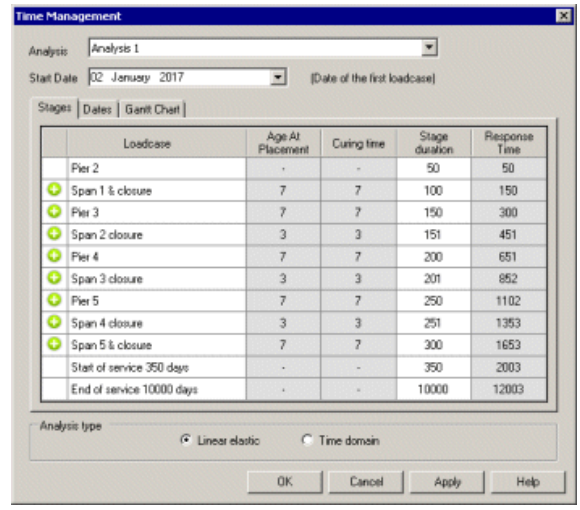
Selection of the new 'Losses based upon time inputs and calculated stresses' option simplifies and avoids the previous need to duplicate particular values for Age, material and geometric attributes that were assigned to the loaded beam. Codified equations are now used to compute creep and shrinkage coefficients that were previously difficult for users to define. Values of "stress at transfer" and "change in stress after transfer" are now computed from assigned model data by solving the model at least twice. In doing this, for an initial solve (the pre-solution), the stress at transfer is assumed to be zero, all time-dependent losses are assumed to be zero, and the prestress loads calculated accordingly. In the second (or subsequent) solve, the stress at transfer is calculated using the eccentricity of each sampling point, and stresses in concrete read from the results of the corresponding loadcase in the first (or previous) solution.

The method of defining prestress prior to this release can still be accessed using the 'Approximate losses, requiring input of estimated stresses' option which allows for producing details and results of short term and long term losses only.

For more information see **Multi Tendon Prestress Wizard** and **Selection of Design Code (Multi-tendon)** and **Tendon Loading Assignment (Multi-tendon)**

Time management facility

The time management facility provides the means to manage a simple construction schedule and easily adjust the duration of pre-defined construction stages in a staged construction analysis. It can be accessed using the **Bridge> Time Management** menu item. It can also be accessed from the **Tendon Loading Assignment** page of the **MultipleTendon Prestress Wizard**.



It can be used to easily and automatically update the total response time values in nonlinear and transient controls that have been previously specified for each loadcase of a staged construction analysis.

For more information see **Time Management**

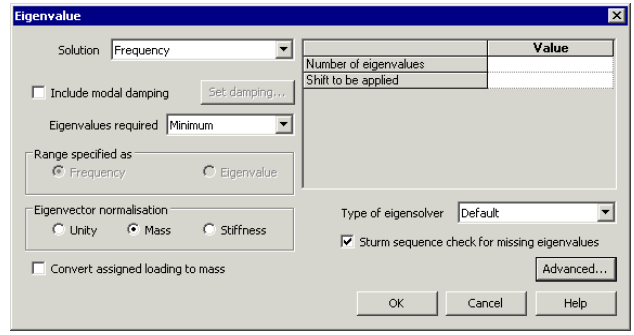
Eigenvalue buckling of stressed structures

The current linear buckling analysis facility has been extended so that it is now possible to define loads that remain constant (dead loads) and those that can vary (live loads) for the computation of a load factor to cause buckling. Linear buckling analysis can now also be carried out after a static nonlinear analysis.

For more information se **Eigenvalue Buckling Analysis**

Conversion of loading to mass for eigenvalue analysis

It is now no longer required to manually convert loading to mass if any semi-permanent loading is to be taken into account as part of an eigenvalue analysis. Instead an option on the eigenvalue control dialog can be selected to automatically make allowance for this loading.



Only loading acting in the direction of the vertical axis is considered.

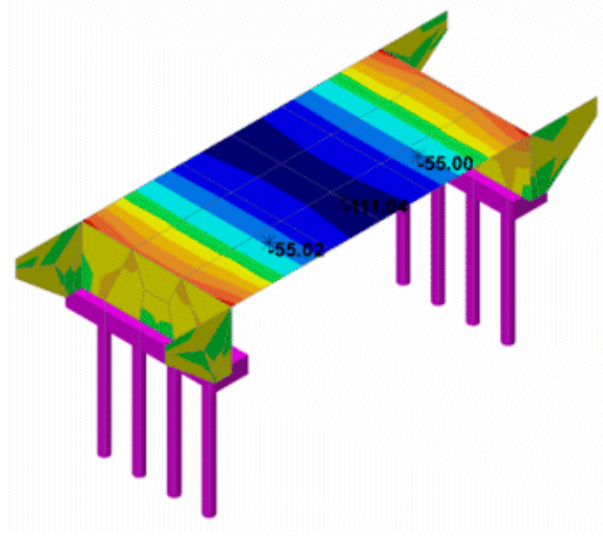
For more information see **How to define Eigenvalue analysis control**

Results related improvements

Inspection Locations

Inspection locations provide the means to obtain results for user-defined positions of interest on a model.

Model features can have multiple inspection locations assigned to them. Inspection locations can report results on elements that belong to the assigned feature, and also those elements that belong to topologically connected features.

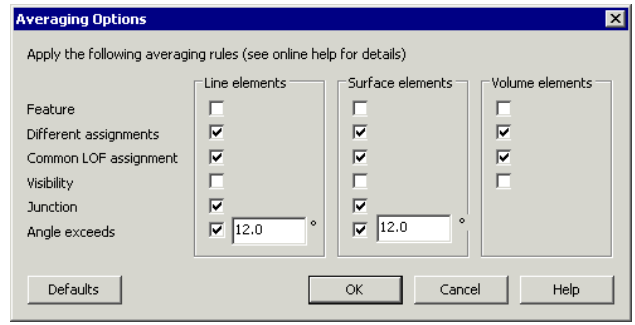


Results can be viewed for all defined inspection locations by selecting the Inspection Locations option from a relevant results dialog, or for individual chosen locations.

For more information see **Inspection locations**

Improved averaging of results

Results averaging rules can now be accessed via the Model Properties dialog or the Model Properties control object in the Analyses Treeview. Averaging rules are common for all analyses.



By default in LUSAS Modeller, averaging of results is done for all situations where it is appropriate. Where averaging is inappropriate, multiple values instead of an averaged value are now displayed. The new Averaging Options dialog allows users to state when averaging rules should and should not take place between adjacent pairs of elements in line, surface and volume features.

For more information see **Model Properties - Options (Averaging Options)**

Enhanced averaging for Direct Method Influence analysis

The former requirement in Version 15 to specify unaveraged or averaged influence results for each participating element when defining an Direct Method Influence attribute has been removed. Instead, for Version 16, this decision will be made automatically. See **Model Properties - Options (Averaging Options)** dialog.

Influence attributes defined in and assigned to a model in Version 15 are unaffected by this new behaviour because the type of results averaging method that was specified at the time the influence attribute was generated is defined within the attribute.

For more information see **Direct Method Influence Attributes**

Selective Results Output

Following results calculation for the whole model, the results written to a results file can be stated to either be for all elements, or now, for a specified subset of elements (those part of a group) within a model. Choosing the latter produces smaller results files more quickly than would be created for a whole model.

For more information see **Advanced Output Options for General Structural/Thermal Analysis** and **Direct Method Influence Analysis**

Improvement to Graph through 2D facility

A slice section can now be created with reference to a selected line, arc or combined line, optionally specifying that the projected length of the line (in a chosen direction) be used to define the length of slice section line on the model. In addition line sections can be created

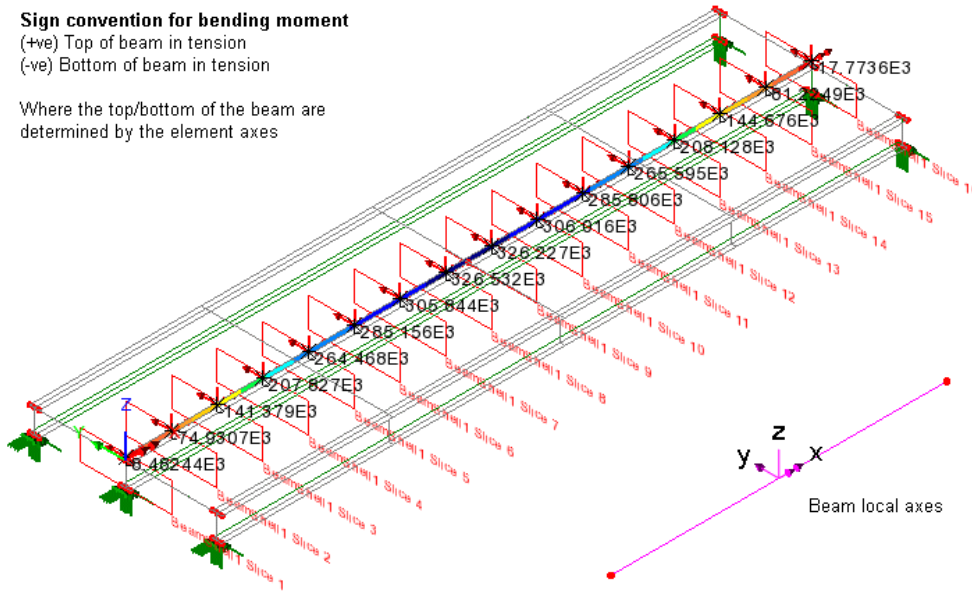
from intersecting surfaces (advanced use) or at the location of an existing graphed location. When defining the line section by cursor or for lines generated by projection, the section line created will, by default, be drawn in the Utilities layer. This may be used later for repeating the cut if a graph along the same line is required.

A 'Width for corridor averaging' value can optionally be specified to increase the number of results used when creating a graph of slice results in order to avoid localised results concentrations.

For more information see **Results on a slice through the model**

Improvements to Beam and shell / slicing resultants

It is now no longer necessary to have to view a model along one of the primary view axes prior to defining a slice location on a model. The slice plane and the orientation of the slice local axes no longer follow the view's rotation axes. Instead, the local yz slice plane of the slice path is used to create the slice planes and this defines the slice local axis.



Multiple beam/shell slice definitions can now be defined in a single model and are saved as utilities within the model. The definitions have been enhanced to provide more spacing options and control over the extent of the model to be sliced.

Results are no longer only displayed immediately after visiting the Slice Resultants Beams and Shells dialog. Now, when slice locations are defined and results are available, results for the slice locations can now be viewed by selecting the Beam/Shell Slice Resultants entity

from the results properties dialogs (Contours, Diagrams etc). The time taken to produce beam shell slice results has been reduced.

The **Print Results Wizard** is now used to select Beam/Shell Slice Resultants of interest for viewing in tabular form or adding to a model report

For more information see **Beam Stress Resultants From Beams and Shells**

Combination and Envelope facilities now support User-Defined Results

Beam stresses and results components defined by the User Defined Results facility can now be used as primary components when calculating and viewing results for combinations and envelopes.

For more information see **Combinations and Envelopes**

Name scoping for User Defined Results

Name scoping (an association of a name to an entity, such as a component or variable) has now been introduced into the User Defined Results facility to help clarify the actual meaning of an included component or variable. Name scoping is optional as long as there is no ambiguity. The range of functions supported by the User Defined Results facility has also been extended.

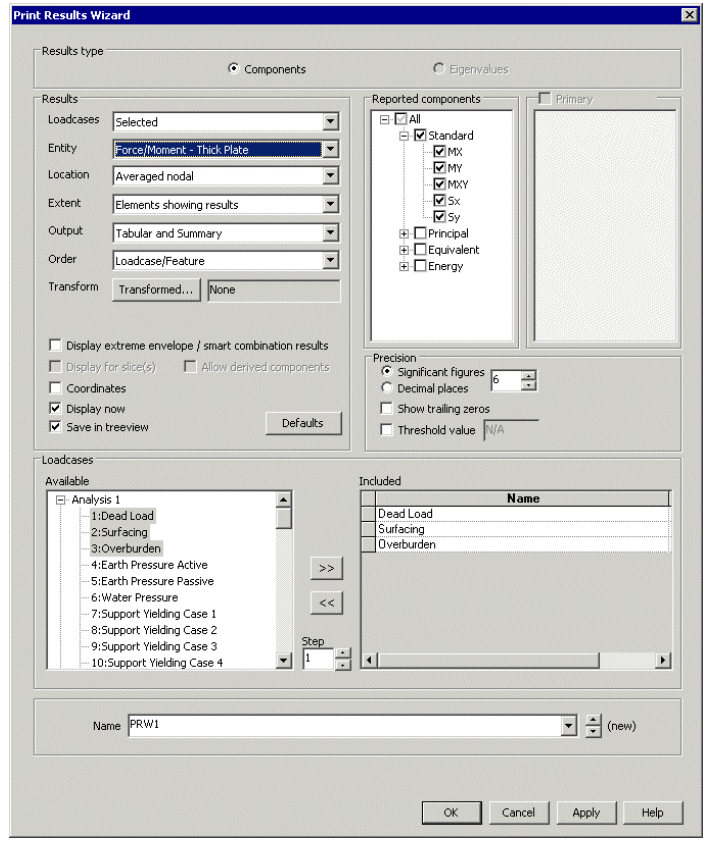
For more information see **User Defined Results** and see **Input and Output of Real Numbers in LUSAS**

Print Results Wizard updated

The Print Results Wizard has been updated to consist of a single dialog containing all possible results selections.

The use of the Print Results Wizard is now consistent with the Graphs and Saved Views facilities with regard to the way in which generated data can be added to reports by either using:

- An **Add to Report** button on the printed results window that is displayed.
- An **Add to Report** context menu for the Print Results Wizard entry in the Utilities Treeview.



The summary facility now presents its output in a consistent, copy and paste friendly, format.

For more information see **Print Results Wizard**

Loadcase results now generated in a model report using the Print Results Wizard facility

The Print Results Wizard dialog is now used from within the 'Add or Edit a Results Chapter' dialog of the Report facility to add chapters of selected loadcase results to a model report.

For more information see **Add or Edit a Loadcase Results Chapter**

Progress bar introduced for Modeller results calculation

A progress bar is now displayed at the bottom of the user interface to show the approximate progress being made by LUSAS Modeller in calculating results for a particular task, such as creating a design envelope, or combination. Each progress step represents one results component being calculated for one constituent loadcase.

For more information see **Progress bar**

Improved storing of Modeller-calculated results

Results for Modeller-calculated results components (such as those calculated for combinations and envelopes, and also for other derived components such as design code-based utilisations, Wood-Amer, user-defined results etc.) can be stored in order to speed-up the initial and subsequent display of those results. This can be done either by setting options to store calculated results in LUSAS Modeller interactively, or by specifying that all or selected results components should always be stored each time a model is solved.

For more information see **Combination and Envelope Options**

Modelling-related improvements

New Modeller User Interface

The Modeller User Interface has been updated to provide a more modern look and feel, easier ways to interact with the view windows, and now supports multiple selection of treeview data for assignment or deletion purposes.

- View windows now appear with individual tabs for easy selection. These can be dragged and dropped within the interface to create side-by-side views.
- Selection Memory, Selected Items, Cyclable Items, and Visible Items selection panels are also tabbed and can be activated and deactivated individually by using the View > Selection Panels menu item. These panels can now be dragged and dropped to any location on screen, docked with other panels and set to Auto Hide, becoming visible again when the cursor is positioned over their tab identifier. Double-clicking a docked window will isolate it again.
- Context menus for each selection panels can be accessed by right-clicking on the tab heading for each panel.
- Pre-version 16 window behaviour can be reinstated by selecting the Window> Layout> Multiple Documents menu item.

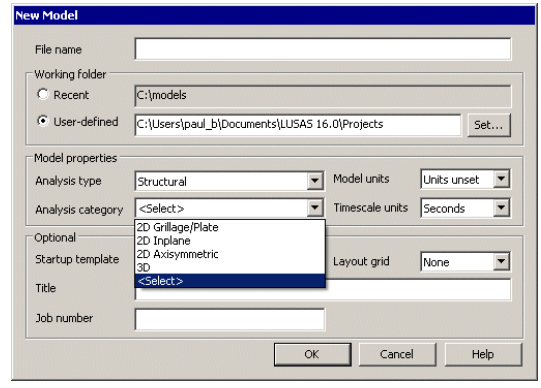
Multiple selection of treeview data

- All Treeviews (with the exception of the Layers Treeview) now support multiple selection of objects in order to carry-out an operation on the items, such as to delete a range of loadcases, or to copy attribute data within the treeview.
- In the Analyses Treeview multiple loading assignments can now be selected to have their load factors changed to one common value at the same time by using the 'Change Load Factor' context menu item.
- Other selection-specific context menus provide options for each treeview entry. Only appropriate context menu options are provided for multiple selected items. For instance a 'Deassign' option may be provided for a particular entry because it would be valid for all selected objects, but an 'Assign' option would be disabled because confusion could occur as the additional assignment data (such as loadcase or analysis) is often required and may conflict.
- Dialogs that show treeview-style check boxes (such as used in the Print Results Wizard dialog) now also support multiple selection.

For more information see **The Modeller User Interface**

Model analysis categories introduced

To prevent potential issues such as those arising from mixing incompatible elements, and/or mixing elements with incompatible attributes and options, an analysis category has been introduced, which must be stated for each model.



Choosing an analysis category of either 2D Grillage/Plate, 2D Inplane, 2D Axisymmetric, or 3D simplifies the user interface where to generally show only those menu items, dialog settings and selections that are appropriate to the type of model being defined. Selecting an analysis category also defines which startup templates are available for selection.

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

For more information see **Model Startup**

Model merge facility

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

For more information see **Model Merge**

BIM / BrIM File import / export

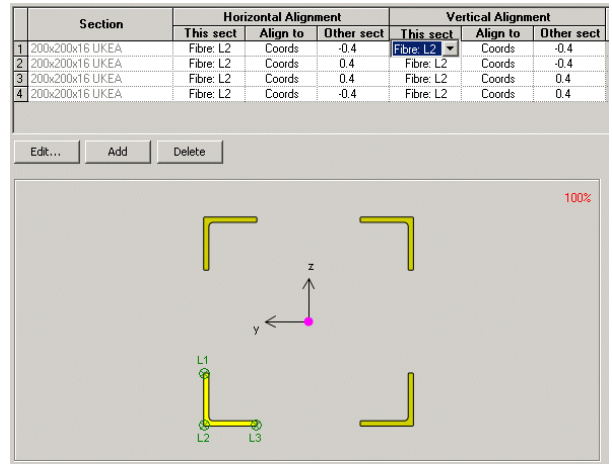
Basic geometry data from third-party BIM/BrIM files (*.ifc) can be imported to create a feature-based geometry model in LUSAS. Both BIM/BrIM Structural domain files (*.ifc) and BIM/BrIM Architectural domain files (*.ifc) are supported for export.

For more information see **BIM / BrIM Files**

Compound section property calculator

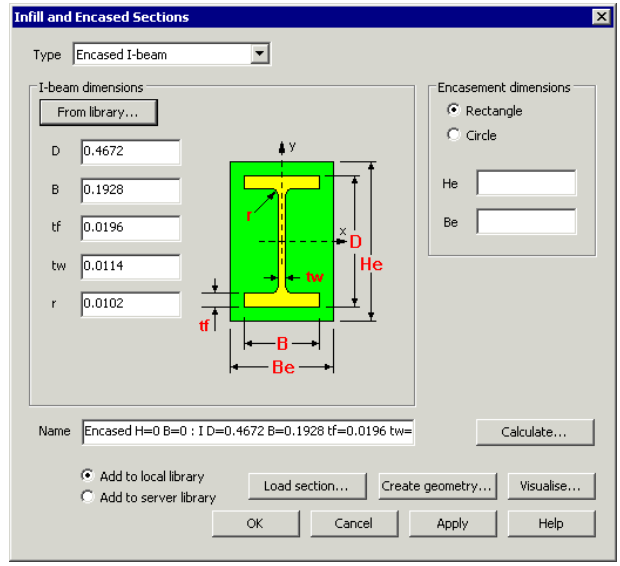
Compound sections can be defined from existing library sections. These can be positioned relative to each other and can have differing material properties assigned. The compound section that is generated can then be assigned to lines in the model and fleshed in the same way as any other geometric attributes.

For more details see **Compound Sections**



Concrete infilled and encased steel sections

The Infill/Encased Section Property Calculator is provided in Bridge and Civil & Structural software products only. The following sections are supported:



- **Filled box**
- **Filled stiffened box**
- **Filled pipe**
- **Filled stiffened pipe**
- **Encased hollow box** - with rectangular or circular encasement
- **Encased filled box** - with rectangular or circular encasement
- **Encased hollow pipe** - with rectangular or circular encasement
- **Encased filled pipe** - with rectangular or circular encasement
- **Encased I-beam** - with rectangular or circular encasement
- **Encased Cross I-beam** - with rectangular or circular encasement
- **Encased Combined-T-beam** - with rectangular or circular encasement

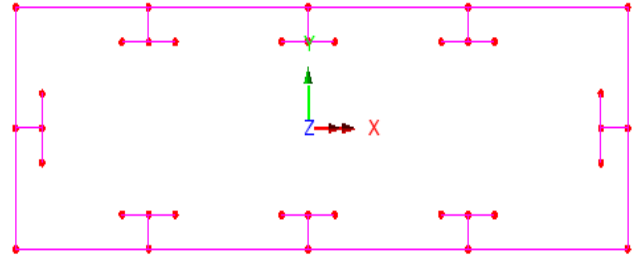
For more information see **Infill/Encased Section Property Calculator**

Chinese steel section library enhanced

Additional cold-drawn, cold-formed, hot-rolled and welded sections are now available for selection from the China section library.

For more information see **Section Library**

Improved arbitrary section property calculation



For thin box sections, only the points and lines that define the centrelines of the plated members, and the geometric thickness of each line need to be initially defined for section property calculation to take place. Surfaces of a width equal to half of the assigned thickness are automatically extruded either side of each line's centreline during the section property calculation process to determine the correct section properties. These surfaces can be optionally retained after values are calculated.

For more information see **Section Property Calculators**

Section property modification / stiffness reduction factors

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

For more information see **Geometric Properties**

Stiffness reduction factor

A stiffness reduction factor control option has been provided on the deactivation dialog of the birth and death facility. This provides control over the amount that the stiffness matrix is reduced in magnitude when elements are deactivated. It enables different reductions in stiffness to be applied to different deactivated elements such that their relative stiffness is changed, which is useful as a means of controlling the deactivated elements. For example, it may be desirable for the final connecting mesh between two cantilevered-out deck ends to have a lower stiffness reduction factor than the cantilevers whilst deactivated.

For more information see **Birth and Death**

Loadcase dependent constraints

Constraints can now be assigned (and be turned 'on' and 'off') on a per-loadcase basis. Previously, they could only be assigned and applied on a per-analysis basis. Amongst many

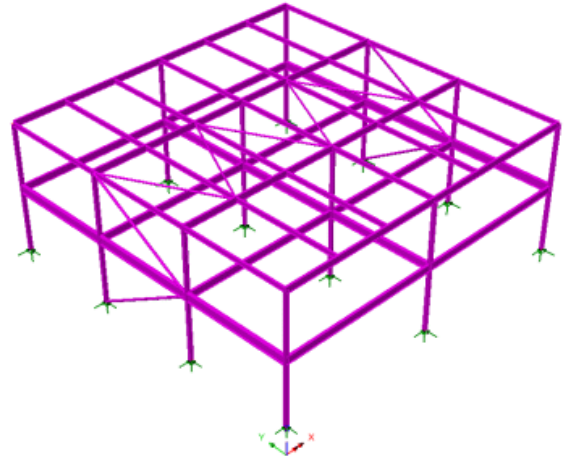
uses, this will enable the control of the positions of inactive nodes in a birth and death analysis.

For more information see **Constraint Assignment**

Perspective viewing

A perspective viewing facility has been implemented in Modeller, and can be obtained by pressing the Perspective button  on the main toolbar. It supports feature selection, resizing, construction grid and page layout facilities. Using a perspective view allows for more realistic viewing and 'fly-through's' of a model.

Parameters to control the view obtained are set on the View Properties dialog. A viewing angle, which determines the rate at which the perspective distortion occurs, and a close depth parameter, which determines how far in front of the view position objects are removed from the visualisation, can be set.

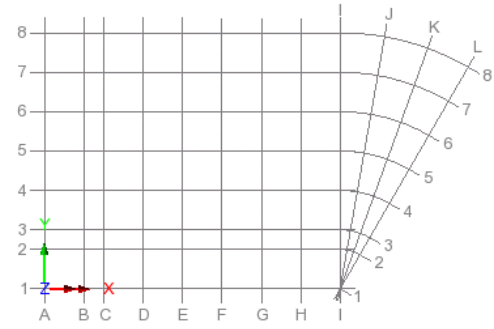


For more information **Manipulating the view of a model** and **Windows Properties.**

Layout grids

The infinite regular layout grid of points as provided in previous versions of LUSAS has been supplemented with a user-definable layout grid.

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



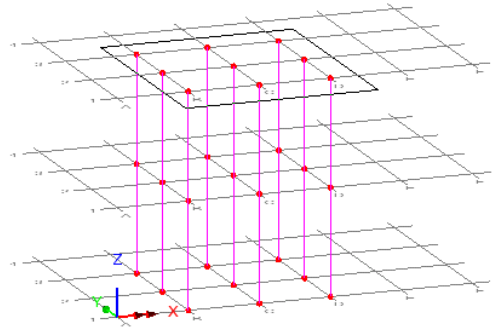
For more information see **Layout Grid**

New point, line and surface drawing options

A new **Geometry > Line> By Grid** menu item allows lines to be defined with respect to a **layout grid**. On each cursor click, a point is created at the nearest grid intersection.

New **Geometry> Line> By Grid> Settings** and **Geometry> Surface> By Grid> Settings** menu items provide access to options to control how lines or surfaces are created when defining line or surfaces features with respect to a **layout grid** using the new **Geometry > Line> By Grid** or the **Geometry > Surface > By Grid** menu items.

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



For more information see **Defining lines and surfaces with respect to a layout grid**

Section property modification / stiffness reduction factors

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

A stiffness reduction factor control option has been provided on the deactivation dialog of the birth and death facility. This provides control over the amount that the stiffness matrix is reduced in magnitude when elements are deactivated.

For more information see **Geometric Properties** and **Birth and Death**

Utilities menu simplified / Tools menu added.

The Utilities menu has been reduced to now only generally contain those menu items that create a Utility dataset in the Utilities Treeview. Other items formerly on the Utilities menu now appear on a new Tools menu, alongside any newly introduced menu items.

For more information see **Model Utilities** and **Model Tools**.

Temperature dependent material properties

Temperature dependent properties of materials of previously defined, or of new materials can be edited using the context menu item **Edit temperature dependence …** for a material in the Attributes Treeview.

For more information see **Edit Temperature Dependent Properties**

Thermal loading types rationalised

Thermal loading types have been reviewed and updated.

For more information see **Thermal Loads**

Internal position of shear springs for joint elements with non-coincident nodes

The internal position(s) of the transverse shear spring(s) for joint elements with non-coincident nodes can now be defined. Parametric distance factors (between 0.0 and 1.0) can be specified to define the position of the shear spring for the local y and z directions. This enhancement allows for import of similar elements from other finite element software.

For more information see **Geometric Properties**

3D rotated label and diagram text

Label text for lines and surfaces can now be optionally drawn to lie along local line axes, or in the local xy or xz planes.

Similarly, for diagrams, labelled values can now be drawn in the plane of the diagram, as opposed to drawing the values flat to the plane of the screen.

For more information see **Labels Layer Properties** and **Layer Properties - Diagrams**

Internal deformed shape of element

LUSAS Solver now computes deformations at each internal calculation point in each beam element, and these may be viewed in LUSAS Modeller as contours or a deformed mesh. This means that the true deformed shape of a line beam can be plotted even if one line mesh division is used to model the beam. For legacy reasons this new option is set 'off' by default.

For more information see **Model Properties - Options (Element Options)**

Changes to the Element Library

New 2D Thick Beam Elements: BMI2 and BMI3

BMI2 and BMI3 are straight and curved thick beam elements in 2D for which shearing deformations are included. The elements can accommodate varying geometric properties along their length. The elements may be used for linear and nonlinear analysis of two dimensional beam, frame and arch structures.

- BMI2 is the element selected by default when a 2D Thick Beam linear element is chosen on the line mesh dialog.
- BMI3 is the element selected by default when a 2D Thick Beam quadratic element is chosen on the line mesh dialog.

New 2D Thick Beam Element with Quadratic Cross-section: BMI2X and BMI3X

BMI2X and BMI3X are straight and curved thick beam elements in 2D for which shearing deformations are included. The elements have a quadrilateral cross section which may vary along the element length. The elements may be used for linear and nonlinear analysis of two dimensional beam, frame and arch structures.

- BMI2X is the element selected by default when a 2D Thick Cross Section Beam linear element is chosen on the line mesh dialog.
- BMI3X is the element selected by default when a 2D Thick Cross Section Beam quadratic element is chosen on the line mesh dialog.

New 2D Plane Strain Beam Elements: BMI2N and BMI3N

BMI2N and BMI3N are straight and curved thick beam elements in 2D for which shearing deformations are included. The element thickness may vary along the length. The element may be used for linear and nonlinear analysis of two dimensional long structures of box girder cross-sections such as tunnel linings and retaining walls for which the plane strain assumption is appropriate. These elements are compatible with 2D plane strain elements.

- BMI2N is the element selected when a 2D Plane Strain Beam linear element is chosen on the line mesh dialog.
- BMI3N is the element selected when a 2D Plane Strain Beam quadratic element is chosen on the line mesh dialog.

New 2D Thick Axisymmetric Shell Elements: BXSI2 and BXSI3

BMI2A and BMI3A are straight and curved thick axisymmetric shell elements in 2D for which shearing deformations are included. The element thickness may vary along the length. The elements can be used for analysing linear and nonlinear shell structures which are axisymmetric, e.g. pressure vessels or pipes.

- BXSI2 is now the element selected when a 2D Axisymmetric Thick Shell linear element is chosen on the line mesh dialog.
- BXSI3 is now the element selected when a 2D Axisymmetric Thick Shell quadratic element is chosen on the line mesh dialog.

New 3D Thick Isoparametric Beam Elements with Torsional Warping: BM21W, BMI31W

Straight and curved beam elements in 3D for which shearing deformations and torsional warping are included. The elements can accommodate varying geometric properties along the length. The elements may be used for linear and nonlinear analysis of three dimensional beam, frame and arch structures. BMI21W may also be used as a stiffener for the QTS4 shell element; while BMI31W may be used as a stiffener for the QTS8 shell element.

New 3D Isoparametic Thick Beam Elements with Quadrilateral Cross-Section and Torsional Warping: BMX21W, BMX31W

Straight and curved beam elements in 3D for which shearing deformations and torsional warping are included. The element has a quadrilateral cross section which may vary along the element length. The elements may be used for linear and nonlinear analysis of three

dimensional beam, frame and arch structures. BMX21W may also be used as a stiffener for the QTS4 shell element; while BMX31W may be used as a stiffener for the QTS8 shell element.

For more information on all these element types see *Element Reference Manual*.

Retired Elements

These elements have been retired because their use has been superseded by elements with improved capabilities. Historical models that make use of these elements are not affected and will still solve. Note that all retired elements do not appear for selection on drop lists but can still be accessed by typing the element name into the Line Mesh Attribute dialog.

- BEAM has been retired and replaced by BMI2, which additionally accommodates varying thickness, and geometric or material nonlinearity. BMI3 is also available as a higher order alternative.
- BMS3 has been retired and replaced by BMI21, which additionally accommodates varying thickness, product moment of area input (for non-symmetric sections, strain output, and geometric and material nonlinearity. BMI31 is also available as a higher order alternative.
- BTS3 has been retired and replaced by BMI21, which additionally accommodates varying thickness,and a higher order variation of forces and moments along its length. BMI31 is also available as a higher order alternative.

General changes

LUSAS installation

The LUSAS installation now follows the Windows convention

LUSAS Programmable Interface (LPI)

All new facilities in Version 16 come with corresponding extensions to the LUSAS Programmable Interface. See LPI documentation for more information.

Modeller Results File caching retired

Modeller Results Files (.mrs) that were created by LUSAS Modeller to save assembled results and speed up the results processing of combinations have been retired and have been replaced in Version 16 with an improved results caching capability. See **Combination and Envelope Options** for more information.

Bridge load combination wizards for selected design codes retired

The bridge load combination wizards for the BD37/88, BRO and the Korean Highway codes have been retired. A new **Design Combination wizard** facility has been implemented in their place.

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 software download file, and are also included on the Version 16 software DVD.

New Worked Examples

The following examples have been created to illustrate new facilities added in this release.

- **Steel Frame Design to EN 1993-1-1** shows the steps required to prepare a model for a steel frame design check in accordance with EN 1993-1-1, and viewing results of member utilisation both as contour plots and as fully annotated design reports.
- **Bridge Design Load Combinations to EN1990** shows the use of the new design load combinations wizard to generate factored design combinations from codes of practice.

Existing worked examples

All examples have been updated to ensure that the examples match changes made to the software and incorporate (where possible) new facilities.

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 may 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 16.

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