

LUSAS

Getting Started Guide

Getting Started Guide

LUSAS Version 17.0 Issue 1

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

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

Distributors Worldwide

Copyright ©1982-2018 LUSAS
All Rights Reserved.

Table of Contents

Getting Started Guide	1
Introduction.....	1
About LUSAS.....	1
LUSAS Application Products.....	1
What's New	2
Contact Information	2
Your LUSAS Installation.....	3
Teaching and Training Version.....	3
Hardware Requirements	3
Using LUSAS - An overview.....	3
Model Types.....	3
Creating models	4
Modelling.....	4
Running the Analysis	6
Viewing the Results	7
Worked examples	7
The LUSAS Modeller User Interface	8
View window	8
Main panels.....	8
Other main panels	9
Context menus	9
Selection Panels.....	10
Text Output Window.....	10
Status bar.....	10
General notes	10
Documentation Available	11
LUSAS Product Options.....	14
Keeping up-to-date	16
Technical Support.....	16
Avoiding and Identifying Problems.....	16
Contacting the Help Desk	16
Web Resources	18
Training Services	18
Engineering Consultancy Services	19
Where Do I Go Next?	19

Getting Started Guide

Introduction

LUSAS is one of the world's leading structural analysis systems. By choosing to use LUSAS you are joining a large worldwide community of engineers who use LUSAS everyday to solve a wide range of engineering analysis problems.

Before starting to use LUSAS it is strongly recommended that you read this Getting Started Guide in its entirety. It provides an essential overview of the LUSAS Modeller user interface and of the processes involved in creating models, running analyses and viewing results. The guide also lists the LUSAS documentation available for you to use and provides information on additional resources including how to obtain technical support.

About LUSAS

The LUSAS system uses finite element analysis techniques to provide accurate solutions for all types of linear and nonlinear stress, dynamic, and thermal/field problems. The two main components of the system are:

- ☐ **LUSAS Modeller** - a fully interactive graphical user interface for model building and viewing of results from an analysis.
- ☐ **LUSAS Solver** - a powerful finite element analysis engine that carries out the analysis of the problem defined in LUSAS Modeller.

LUSAS Application Products

LUSAS is made available as different application products with the user interface tailored specially to the needs of a particular application. For example bridge engineers will see facilities provided specifically for the analysis of bridge structures. Such facilities are normally conveniently grouped together inside a product application menu, such as Bridge, Civil, Composite etc, that is loaded between the Utilities and Window menu items in the main menu.

Academic users have the choice of which application product they would like to use. When starting LUSAS Modeller an additional dialog offers this choice.

The application products include:

- ☐ **LUSAS Bridge** – for bridge engineering analysis, design and assessment.
- ☐ **LUSAS Civil & Structural** – for civil, structural, nuclear, seismic, geotechnical and offshore engineering.
- ☐ **LUSAS Analyst** – for automotive, aerospace, defence, manufacturing, and general engineering analysis.
- ☐ **LUSAS Composite** – for engineers designing composite products or components.
- ☐ **LUSAS Academic** – available to academic establishments for teaching and research use and allows use of all of the above products together with most of the options available for those products.

What's New

When you run LUSAS Modeller for the first time you will be presented with information that links to a 'What's new' page. This will tell you what has changed since the last major version and cover any interim releases. It will be of particular interest to existing users upgrading from an earlier version of LUSAS.

Contact Information

LUSAS is the trading name of Finite Element Analysis Ltd whose headquarters is located in the UK. LUSAS is supported around the world by a number of LUSAS regional offices as well as by a network of LUSAS Distributors and Resellers. See the LUSAS website for details.

LUSAS Headquarters

Forge House,
66 High Street,
Kingston upon Thames,
Surrey,
UK

Tel: +44 (0)20 8541 1999
Fax: +44 (0)20 8549 9399

Web: www.lusas.com

LUSAS Contact Email Addresses:

General Enquires : info@lusas.com

Sales Enquires : sales@lusas.com

Technical Support : support@lusas.com

Training Services : training@lusas.com

Consultancy Services : consultancy@lusas.com

Your LUSAS Installation

Your LUSAS software is provided to work with either a standalone licence or a network licence. A security key code is also required to use the software. Full details on how to install, modify or configure LUSAS are provided in a separate *Installation Guide* which is available in print form or in PDF format on the installation CD.

Teaching and Training Version

Your LUSAS software can be used for teaching and training purposes, and when done so the software runs with limitations on problem size. Please refer to the *Installation Guide* for further information or contact your LUSAS supplier if you wish to use LUSAS for teaching or training.

Hardware Requirements

For the latest information on the recommended and minimum system requirements for LUSAS please refer to <http://www.lusas.com/pcspec.html>

Using LUSAS - An overview

Carrying out an analysis with LUSAS will involve three main stages:

- ☐ **Modelling**
- ☐ **Running the analysis**
- ☐ **Viewing the results**

Modelling is sometimes referred to as 'pre-processing' and viewing and manipulating the the results is sometimes referred to as 'post-processing'. Further details of these stages are provided below.

Model Types

Two types of model can be created in LUSAS:

- ☐ **Feature-based geometry models** these are based on features (such as points, lines, surfaces and volumes) that are defined inside LUSAS, or are created from imported geometry and require the definition of mesh objects such as elements with their associated nodes.
- ☐ **Mesh-only models** - these comprise only mesh objects (elements and their associated nodes, edges and faces) and are created by importing only those types of LUSAS model data, or third party data that are supported.

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

Creating models

When creating a new model number of settings will have to be initially made. These include a model filename, model units and an analysis type and category.

Choosing an Analysis Type

Choosing an **Analysis type** tailors the interface for the type of analysis to be carried out and also defines the class of model. When creating a new model it can be defined as being for a Structural, Thermal, Hygro-Thermal, Coupled Thermal/ Structural, or Coupled Hygro-thermal/Structural analysis dependent upon the licence key in use.

Choosing an Analysis Category

Choosing an **Analysis category** modifies the user interface to generally only show those options that are appropriate to the type of model being defined. Analysis category choices are: 2D Grillage/Plate, 2D Inplane, 2D Axisymmetric, and 3D.

Selecting an analysis category as part of creating a new model defines which startup templates are available for selection. It also determines the initial vertical axis for a new model, and the orientation of element types and library items as displayed on the Geometric Line dialog prior to them being added to a model.

Modelling

Modelling involves creating a geometric representation of a structure and defining its characteristic behaviour in terms of its physical properties such as material, loading, and support.

Geometry

Within LUSAS Modeller the geometry of the model is defined using points, lines, surfaces and volumes. A volume needs surfaces to enclose it and to define its boundary. Similarly a surface needs lines to form its perimeter and lines need points to define their ends. The shape of a surface between its boundary lines, and the shape of a line between its end points can be simple (straight lines, flat surfaces) or complex, depending on the manner in which it was created.

Many different tools exist within LUSAS Modeller to aid the creation of geometry. In particular the geometry can be entered numerically, in terms of coordinates, by cursor (using the mouse) or by copying, splitting, intersecting and transforming previously existing parts of the model. Geometry can also be imported from third-party interface files.

Meshing attributes

Whilst points, lines, surfaces and volumes allow the geometry of the problem to be represented or imported, to solve the problem, the geometric features must be broken down into discrete nodes and elements. This process is known

as meshing and the collective term for all the elements and nodes, once created, is the mesh. Special attributes, called mesh attributes, are created and assigned to relevant geometry features to define the type and number of nodes and elements that will be used to represent each part of the geometry.

A significant advantage of this approach is that the density of the mesh can easily be changed without rebuilding the geometry or reassigning any attributes. Simply modifying a mesh attribute automatically changes the mesh density in any part of the model where it is assigned.

Similarly, the element type can be changed in exactly the same way. Different element types are suitable for different kinds of analysis, and the system provides an easy way to change from one to another.

Generally LUSAS Modeller automatically and incrementally updates the mesh whenever a change is made to any part of the geometry or any mesh attributes. However these automatic updates are not allowed when results are being displayed, and they can also be switched off if at other times if required, for example to save time. This is known as locking the mesh. When the mesh is locked it will only be updated when specifically requested.

Other attributes

Other attributes are used to define aspects of the behaviour of parts of the model, or the external factors which are imposed on it. Within LUSAS Modeller, there are several types of attribute such as materials, loading and supports. Within each attribute type there may be further sub-divisions, for example there are isotropic materials, anisotropic materials, and orthotropic materials, among others.

In each case an attribute is first created, and then subsequently attached to all or selected parts of the model. This attachment process is known as assigning. Thus a material can be assigned to a line. Once assigned, the line takes on the properties of the material until further notice. Assignment often involves specification of additional information that does not define the attribute or the geometry, but rather defines how one is applied to the other. This includes directions, scale factors and similar.

A special attribute, called a geometric attribute, is used when some or all of a model is comprised of lines or surfaces. In this situation, LUSAS Modeller needs to know the cross-sectional appearance and behaviour of lines, and also how thick the surfaces are. Geometric attribute assignments provide this information, and therefore form part of the model's appearance. Models comprised entirely of volumes do not need geometric assignments.

Many different tools exist within LUSAS Modeller to give interactive feedback on the nature of assigned attributes. This process is generally termed

visualisation. Visualisation usually takes the form of coloured infills, arrows, textual labels or similar.

Analyses

Within a single model file any number of analyses may be defined. Each analysis will inherit most of the assignments, options and settings of what is called the base analysis for the model. Use of the multiple analyses facility does away with the need to create separate models, or maintain “clone” copies of a model, in order to switch between and create results from linear static analysis and other analysis types.

Loadcases

In a linear analysis, it is convenient to gather certain types of applied loading together, as for example gravity and wind loading which are applied at the same time. A loadcase is the term for such a collection of loading. Each loadcase is solved separately and the results are available separately. Results can be combined together during the results viewing stage if required.

In a nonlinear analysis, LUSAS loadcases are equivalent to analysis stages. Within each loadcase, the loading, supports, or type of analysis can be changed. Control parameters can be set for each loadcase.

Utilities

Some kinds of analysis require or create extra data that is similar to attributes, but cannot be assigned to geometry. LUSAS Modeller refers to such data as utilities. Tendon profiles, reference paths and the settings made or entered to plot a graph or produce a report, for instance, are all examples of utilities.

Controls

Finally, for analyses other than simple linear static, it is sometimes necessary to control the analysis process itself. LUSAS Modeller uses controls for this purpose. Controls are either nonlinear, transient, eigenvalue or fourier in nature.

Further Modelling Information

Browse the LUSAS Modeller interactive online help system, or see the *Modeller Reference Manual* for detailed information about any of the above.

Running the Analysis

Once the modelling process is complete, the model is passed to LUSAS Solver for analysis. LUSAS Solver will then create results that are automatically loaded back into modeller for viewing and optional post-processing.

Viewing the Results

Results are available to be viewed in many different ways, and the results can, if desired, be combined with attribute visualisation to make a plot, for example, that shows the applied loading and the resulting deformations.

This is just a selection of some of the results facilities that are available:

- **Deformed mesh**
- **Values of stress, displacement, reactions, etc.**
- **Inspection locations**
- **Averaged contours**
- **Unaveraged contours**
- **Wood-Armer reinforcement calculations**
- **Slicing a plane through a solid**
- **Slicing a line across a surface**
- **Yield markers**
- **Graphs**
- **Animations of the above, either through time or in response to excitation.**
- **Combinations and envelopes of results.**
- **Model reports**

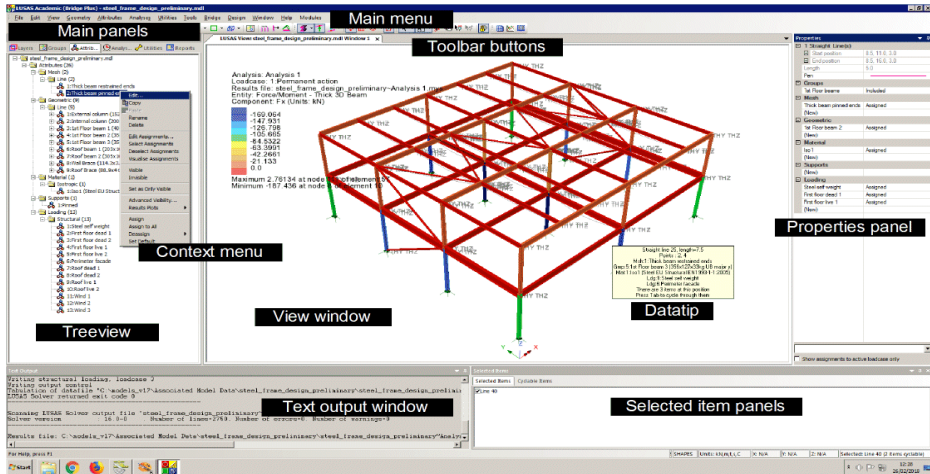
Worked examples

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

After running LUSAS, see **Help > Help Topics > Worked Examples** for more information.

The LUSAS Modeller User Interface

The image below shows the principal user interface components of LUSAS Modeller.









View window

- ❑ A **View Window** shows the model according to user-defined settings made elsewhere on the user interface. Each view window is completely independent and may show the model from a different angle, using different visualization options, with or without results. The image in a view window is built up from several drawing layers as listed in the Layers Treeview (see Main Panels below). Each draws over the top of the one before it.
- ❑ A **Datatip** provides model-related information about whatever is under the cursor.
- ❑ A **Tool tip** reports on uses of toolbar buttons or expected input for grid cells in dialogs etc.

Main panels

Each of the main panels has its own **Treeview** showing a list of objects of a particular type. There are panels of Layers, Groups, Attributes, Analyses, Utilities and Reports. Each object in a Treeview has its own functionality. This can be invoked using a context menu, the main menu or the toolbar buttons. In summary:

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

- ☐ The **Groups**  Treeview is used to store selected user-defined collections of objects (geometry, nodes or elements) under a collective name.
- ☐ The **Attributes**  Treeview is used to store named collections of data that may be attached (assigned) to the model geometry, such as the element type and discretisation on the geometry; section properties and thicknesses; the materials used; how the model is supported or restrained; and how the model is loaded.
- ☐ The **Analyses**  Treeview shows all analyses that have been defined including loadcases, eigenvalues, results, and control chapters. Also in this treeview are post-processing concepts such as combinations and envelopes.
- ☐ The **Utilities**  Treeview contains utility items used in the definition of model geometry or attributes, or to control an analysis, or to provide a particular functionality, such as to define a load combination or produce a report for example.
- ☐ The **Reports**  Treeview contains named reports, each of which is made up of multiple chapters. Chapters may be defined to view modelling information, results, screenshots and similar.

Other main panels

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

- ☐ **Properties** - displays the current properties of selected geometry features (points, lines, surface, volumes) and selected mesh objects (nodes, edges, elements, faces).
- ☐ **Hierarchy** - shows the merge status, and lists the higher and lower order features/objects of a single selected geometry feature or mesh object. The type and feature number of the selected feature is shown.

Context menus

- ☐ A **Context menu** (displayed by pressing the right-hand mouse button with an object selected) provides access to additional functionality such as editing of data, control of visibility, visualisation of assignments, and selective control of results plotting on selected attributes. Context menus are provided for objects selected in the View Window, and most items in the various Treeview panels.

Selection Panels

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

Text Output Window

- ❑ The **Text Output Window** provides continuous feedback during a modelling or analysis process, and includes errors, warnings and progress messages. When an error message or warning relating to a particular object is written to the text output window, extra information is sometimes available by double-clicking on that line of text.

Status bar

- ❑ The **Status Bar** displays progress messages and help text during a modelling session, the model units, the current cursor position in model units (if the model is displayed in an orthogonal plane) and the item or number of items in the current selection. A view of the model can be obtained by pressing one of the primary axes shown in the status bar.

General notes

Within each View window, and within each Treeview, there is a current selection and all commands are assumed to apply to it. In both the View window and for particular entries within the Treeview several objects can be selected at the same time. Main menu and toolbar commands apply to whichever window was most recently clicked, either a View window or a Treeview panel.

Some items within a Treeview support drag and drop functionality. Typically this will move them, copy them or connect them together. For example, dragging an attribute

from the Treeview onto a View window assigns the selected attribute to the selected object in that window.

All the menus, toolbars, tree frame, text window, status bar and other similar windows can be resized, moved, made visible or invisible, and docked or separated from the main program window. LUSAS Modeller will remember their new location for the next time the program is run. Additional customization is possible - see the *Modeller Reference Manual* for details.

Documentation Available

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

The documentation includes:

- ☐ **Installation Guide**
- ☐ **Getting Started Guide**
- ☐ **Modeller Reference Manual**
- ☐ **Worked Examples**
- ☐ **Application Manual (Bridge, Civil & Structural)**
- ☐ **Autoloader Reference Manual**
- ☐ **IMDPlus User Manual**
- ☐ **Rail Track Analysis User Manual**
- ☐ **Element Reference Manual**
- ☐ **Solver Reference Manual**
- ☐ **Theory Manual (Volume 1 and 2)**
- ☐ **Verification Manual**
- ☐ **CAD Toolkit User Manual**

Installation Guide

Details of installing LUSAS software for various licensing options.

- Available in PDF and printed form.

Getting Started Guide

Contains a brief overview of LUSAS.

- Available in PDF and printed form.

Modeller Reference Manual

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

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

Worked Examples

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

- Available in PDF form.

Application Manual (Bridge, Civil & Structural)

This describes the bridge, civil and structural application specific features of LUSAS and their uses.

- Available in PDF and printed form.

Autoloader Reference Manual

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

- Available in PDF form.

IMDPlus User Manual

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

- ☐ Seismic response analysis of structures subjected to acceleration time histories of support motion.
 - ☐ Moving load analysis of structures, such as bridges, subjected to moving vehicle or train loads, where the magnitude and configuration of the loading remains constant throughout the analysis.
 - ☐ Moving mass analysis of structures, such as bridges, subjected to moving vehicle or train loads, where spring-mass systems are used at the axle/bogie positions of a vehicle.
- IMDPlus help is provided in on-line help format and also available in PDF and printed form.

Rail Track Analysis User Manual

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

- Available in PDF form.

Element Reference Manual

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

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

Solver Reference Manual

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

- Available in PDF and printed form.

Theory Manuals (Volume 1 and 2)

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

Theory Manual 1 topics include:

- ☐ Analysis procedures including: linear, nonlinear, dynamics, eigenvalue extraction, modal analysis, all forms of field analysis, fourier analysis and superelement analysis.
- ☐ Geometric nonlinearity.
- ☐ Constitutive material model formulations.
- ☐ Loads and boundary conditions with particular reference to general load types, constraint equations, slidelines and thermal surfaces.
- ☐ More complex post processing calculations, including nodal extrapolation and calculation of Wood Armer reinforcement moments
- Available in PDF form only.

Theory Manual 2 topics include

- Element formulation theory.

- Available in PDF and printed form.

Verification Manual

A manual of LUSAS testcase examples benchmarked against known solutions.

- Available in PDF form only.

CAD Toolkit User Manual

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

- Provided in PDF form only.

LUSAS Product Options

In addition to LUSAS software being provided as particular individual application products, product options are also available which extend the analysis capabilities of these products. Product options can be made available for use by the supply of an appropriate security key that 'unlocks' the product option, allowing immediate access to the extended facilities in your currently installed software. If your current license doesn't support these product options then please contact your LUSAS supplier to find out how you can gain access to them.

The following software options are available for all products:

- ☐ **Fast Solvers** - This option includes additional solvers. The Fast Multifrontal Direct Solver can provide solutions several times faster than the standard Frontal Direct Solver for certain analysis problems. The Fast Multifrontal Block Lanczos Eigensolver can, similarly, return results several times faster than the standard Frontal Eigensolvers for certain problems. The complex eigensolver provides efficient solutions for large-scale damped natural frequency problems.
- ☐ **Nonlinear** - Nonlinear stress analysis is becoming increasingly important with designers employing a wider variety of materials in a multitude of different applications. The Nonlinear option, rightly regarded as the leader in nonlinear analysis, provides the very latest powerful techniques for solving problems having either material, geometric or boundary nonlinearity.
- ☐ **Dynamics** - Straightforward modal dynamics problems can be solved using Interactive Modal dynamics (IMD) techniques provided in all LUSAS products. The Dynamics Option contains the facilities required to solve a wider range of dynamic problems in both the time and frequency domains. By combining the Dynamics and Nonlinear options both high and low velocity

nonlinear impact problems can be solved using either implicit or explicit solution techniques.

- ❑ **Thermal/Field** - The Thermal / Field option contains extensive facilities for both simple and advanced steady state, and transient thermal / field analyses. By combining the Thermal / Field option with other appropriate options, heat transfer due to conduction, convection and radiation can be analysed. In addition, the effects due to phase change of material may also be included.
- ❑ **Heat of Hydration** - Allows modelling the heat of hydration of concrete. The heat of concrete hydration can be computed during a thermo-mechanical coupled analysis and the temperatures and degree of hydration can be read into a structural analysis.
- ❑ **IMDPlus** - The IMDPlus option extends the Interactive Modal Dynamics (IMD) techniques provided in all LUSAS products. Whilst IMD models a single loading event in a single direction, IMDPlus allows multiple loading events with more advanced loading conditions to be solved. IMDPlus is used for three primary uses: Seismic response analysis of structures subjected to acceleration time histories of support motion; moving load analysis of structures, such as bridges, subjected to moving vehicle or train loads, where the magnitude and configuration of the loading remains constant throughout the analysis; and moving mass analysis of structures, such as bridges, subjected to moving vehicle or train loads, where spring-mass systems are used at the axle/bogie positions of a vehicle.

The following options are available for LUSAS Bridge only:

- ❑ **Vehicle Load Optimisation** - The Vehicle Load Optimisation facility complements and extends the vehicle loading capabilities of LUSAS Bridge and helps to simplify the evaluation of worst load position for various load configurations. Vehicle load optimisation is used to identify critical vehicle loading patterns on bridges and apply these loading patterns to analysis models. It reduces the amount of time spent generating models and leads to more efficient and economic design, assessment or load rating of bridge structures.
- ❑ **Steel and Composite Deck Designer** - This carries out comprehensive calculations for multiple sections on steel or steel/composite bridge decks to the Eurocodes.
- ❑ **Rail Track Analysis** - The Rail Track Analysis option permits track/bridge interaction analysis to the International Union of Railways Code UIC 774-3. It allows you to automatically build models from data defined in MS Excel spreadsheets, run an analysis, and quickly produce results in spreadsheet or results file formats.

The following option is available for LUSAS Bridge and LUSAS Civil & Structural only:

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

Keeping up-to-date

Our occasional emailed newsletter, LUSAS eNews, will keep you informed of latest developments, new software versions and updates etc. [Click here](#) to subscribe, or email enews@lusas.com.

You can check on the latest news, software releases and events by visiting the LUSAS website at <http://www.lusas.com/news.html>

Technical Support

Avoiding and Identifying Problems

A team of engineers in LUSAS Technical Support is available to all clients who have a current support and maintenance contract, to provide modelling advice and assistance with any problems that may be encountered. However, to identify the cause of a problem may take time, especially if the analysis is large. Therefore preventative measures should always be taken in an effort to avoid problems in the first place.

1. Your analysis model should be thoroughly checked against intended data input to verify that all geometry, attributes and assignments have actually been input correctly.
2. Read and follow the advice in the *Pre-Analysis Checks* and *Post-Analysis Checks* sections of the Running an Analysis Chapter of the *Modeller Reference Manual*. It is always good practice to systematically carry out these checks as a matter of course whether or not problems are encountered.
3. Contact your local LUSAS office or LUSAS distributor, preferably by email including the relevant information suggested above.

Details for local offices and distributors can be found at <http://www.lusas.com/distributors>

Contacting the Help Desk

With the aim of producing a more efficient service, some general guidance is given as to what information should be prepared before calling the Help Desk.

Please note that if LUSAS was supplied to you by a LUSAS distributor then they should be your first point of contact for support. Your local distributor will have access to the main LUSAS Technical Support Centre for assistance if required.

This information will help the LUSAS support engineers understand your problem more quickly:

- The exact text of any warning or error message(s). Preferably copied and pasted or in the form of a screen dump.
- Machine specification - operating system, physical memory, virtual memory and available disk space.
- A copy of the model or data file causing problems.
- A list of the last commands used in LUSAS Modeller or a copy of the session file.
- The contents of the last LUSAS Modeller error log, LUSASM_x.ERR
- Full details of the LUSAS Modeller/LUSAS Solver version numbers in use, the LUSAS Solver version number is written to the header section of the output file and the LUSAS Modeller version number is obtained from the **Help> About LUSAS Modeller** menu item.
- For complex or difficult to describe problems, email or fax a simple diagrammatic representation before calling to aid any discussion.
- If telephoning the Help Desk try to be logged onto the computer when calling or be close enough to the computer to try any suggestions provided by the support engineer.
- A support tool is available to help ensure that if a support query is raised all relevant model files can be submitted to LUSAS Technical Support to aid with the resolution of the query. It is accessed using the **Help > Support Tool** menu item.

LUSAS Technical Support

Contact details for the main LUSAS Technical Support Centre are:

LUSAS Technical Support

Forge House,
66 High Street,
Kingston upon Thames,
Surrey,
UK

Tel: +44 20 8541 1999

Fax: +44 20 8549 9399

Email: support@lusas.com

Web Resources

A number of resources are also available via the main LUSAS website, regional LUSAS Websites and distributors' own websites.

Main LUSAS website

<http://www.lusas.com>

General support services

<http://www.lusas.com/support>

LUSAS User Area

<http://www.lusas.com/usrcheck.html>

Note that the User Area requires a username and password for access. This will be provided initially to the named LUSAS contact within your organization who should be consulted if necessary. It can also be requested by and sent to registered users of LUSAS. The User Area contains:

- Frequently asked questions (FAQs)
- Downloads (Software updates, latest examples etc.)
- Knowledge base
- And much more ...

Training Services

Comprehensive training services are available to help get you up-to-speed quickly and ensure that you have sufficient knowledge of the relevant facilities in LUSAS to tackle the types of analysis that you want to do. The structured courses cover all aspects of LUSAS and run regularly throughout the year at designated venues. They can also be run at your site at a time to suit you. In addition we can also provide bespoke training to satisfy any particular requirements.

For further information see <http://www.lusas.com/support/training.html> or contact us at training@lusas.com

Engineering Consultancy Services

If you need greater assistance than that available through Technical Support then our team of engineering consultants is here to help. Whether you have a difficult analysis to carry out or need help with a peak workload, our rapid and cost-effective service will ensure that you get the best out of your designs.

For further information see <http://www.lusas.com/consultancy> or contact us at consultancy@lusas.com

Where Do I Go Next?

Worked examples covering different modelling and analysis procedures using LUSAS are provided in PDF format as part of a LUSAS installation from a software download file or release CD. These and any additional or updated examples are also available from the protected User Area of the LUSAS website.

Read the 'Introduction to Worked Examples' first, and work through the first example in your specific field of interest, followed by as many other examples that may be relevant to you.



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 | info@lusas.com | www.lusas.com