

# Technical Specification

## V14.7 LUSAS software products

### Modelling, analysis, results, and product specific features

#### Modelling features

##### User Interface

- Full Native Windows XP/Vista/Windows 7 implementation
- User configuration utility for licensing and specifying primary software defaults
- User-defined startup templates
- Treeview-based viewing of model data
- Separate viewable layers for mesh, geometry, attribute, annotation and utility data.
- Multiple model windows, and optional text output window and command line
- On/off display control of model layers
- Display interrupt facility
- Dialogs, file selection boxes, toggle buttons
- Datatips shown when hovering over features
- Evaluation of expressions during data input
- Open Interface to ActiveX compliant Windows software such as MS Excel, Word, Access etc.
- Hardware/software OPENGL displays
- Cut and paste of graphical and text data
- Online help

##### Geometry

- Fully associative feature modelling - based on CAD type data structure
- Full range of modelling units
- Boolean geometry operations
- Fleshing of geometric attributes
- Definition of tapered arbitrary cross-sections
- Definition of multiple varying sections based upon a distance along a line or path of lines
- Reference path facility
- Definition of extreme beam fibre locations for viewing of results
- Analytical Shape wizard
- Generation of points, lines, surfaces, and volumes in Cartesian, cylindrical or spherical coordinate systems
- Automatic generation of surface and line intersections
- Feature manipulation with copy, mirror and translate commands
- Accurate arc definition
- Multi level undo/redo facility
- Options for manifolding, filleting and splitting
- Section property calculator for standard, arbitrary and pre-cast sections
- Standard steel section libraries for UK, USA, EU, and many other countries / regions

## **Meshing**

- Feature-based meshing capability
- Isoparametric meshing in 2D and 3D
- Automatic irregular meshing in 2D
- Extruded irregular meshing in 3D
- Adaptive meshing
- Mesh interrupt facility
- Mesh-only modelling capability
- Mesh import facility.

## **Attributes**

- Creation and visualisation of varying attributes over features
- Drag and drop assignment to model
- Generation of geometric properties, material properties, boundary conditions
- Automatic support stiffness distribution
- Automatic constraint equation generation
- Generation of wide range of load types
- Definition of load case titles for ease of results interpretation
- Feature based sideline and thermal surface generation and visualisation
- Automatic generation of data input for all analysis types
- Gravity loading applied on a feature or loadcase or model basis

## **Display**

- Hidden line removal
- Display of element numbers, node numbers, material type, boundary conditions and load data
- Exploded element views
- Colour fill options
- Interactive rotation and transformation options to enable viewing of model from any angle or eye position
- Pre-defined primary views and isometric view
- Selective element viewing by element type, area, geometry type, material type or feature
- Pan, zoom and windowing options
- 3-plane drawing grid
- Multiple graphic windows for unlimited number of views of a model
- Group facility
- Full plot annotation
- Animation over multiple model load cases
- Working and Page layout views

## **File Handling**

- Command file input
- Scripting language support for application customisation
- Session file logging

## **Interfacing**

- PATRAN, IDEAS, FEMVIEW, FAM and STEP interfaces
- IGES import and DXF import and export
- LMS interface
- Import mesh data from Abaqus input files, Ansys cdb files, and Nastran bulk data files
- Extensive user interfaces
- Picture file option for hardcopy output
- Print directly to Windows printers

## **Analysis features**

### **General System Facilities**

- Online help
- Comprehensive error diagnostics
- User defined element and nodal output options
- Full range of load types
- General purpose load curve input
- Generalised constraint equations
- Resolution for multiple load cases
- Flexible restart facility
- Superelements (substructures)
- Efficient automatic frontwidth optimisation
- Efficient frontal equation solver for both large and small problems
- Pre-conditioned conjugate gradient iterative solver for fast solutions of large problems

### **Verification**

- Rigorous internal Quality Assurance procedures
- Comprehensive machine checked testing
- HECB calibration and NAFEMS calibration tests

### **Implicit Stress Element Types**

- Plane frame/truss
- Grillage
- Space frame/truss
- Curved thin and thick beams with constant/variable cross sections
- Plane stress/plane strain
- Plate flexure (thin and thick)
- Ribbed plates
- Axisymmetric solids with non axisymmetric loading
- Axisymmetric membranes
- Axisymmetric thin shell
- Flat thin shells
- Curved thin Semiloof shells
- Flat/curved thin/thick co-rotational shells
- 3D solids
- Composite shell
- Composite solids
- Generalised joint/gaps including seismic isolators, viscous dampers, lead rubber bearings and friction pendulum
- 2D plane stress/plane strain/ axisymmetric solid crack tip
- Pore water pressure modelling (Plane strain only)

### **Explicit Stress Element Types**

- Plane stress/strain with hourglass stabilisation
- Solid with hour glass stabilisation

### **Thermal (Field) Element Types**

- Bars
- Plane
- Axisymmetric solids
- Axisymmetric membranes
- 3D solids
- Links

### **Solvers**

- Iterative (PCG) solver
- Fast multifrontal direct solver
- Fast multifrontal block Lanczos eigensolver
- Fast complex eigensolver
- Frontal (direct) solver

### **Linear Materials**

- Isotropic, orthotropic, anisotropic and rigidity models
- Isotropic and orthotropic thermal materials
- Composite lay-ups for shell and solid material models
- Temperature dependency for all linear material models

### **Nonlinear Materials**

- Plasticity model with isotropic and kinematic hardening using von-Mises criteria, includes a backward Euler stress update algorithm with consistent tangents
- Anisotropic plasticity model with isotropic hardening using Hill or Hoffman criteria, includes a backward Euler stress update algorithm with consistent tangent
- Plasticity model with isotropic hardening using a modified von Mises criteria with different properties in tension and compression, includes a backward Euler stress update algorithm with consistent tangent
- Concrete models with opening and closing cracks and strain softening based on fracture energy in 2D and 3D
- Concrete creep and shrinkage model to CEB-FIP Model Code 1990
- Concrete heat of hydration modelling
- Viscous damped joints
- Multi-surface cracking concrete with crushing material model
- Geotechnical model using Mohr Coulomb criteria including non-associative flow for soils and rocks
- Volumetric deformation model for soils and crushable foams
- Ogden and Mooney-Rivlin models for rubber materials with very large strains
- Composite lay-ups for shell and solid nonlinear material models
- Temperature dependency for all nonlinear material models
- Creep model with time dependency and strain hardening
- Viscoelasticity
- Phase changes
- User-defined nonlinear material and creep interfaces
- Damage model
- Hashin material model for composite materials
- Material model interface (MMI) for user input

### **Eigen Analysis**

- Lanczos and Subspace EigenSolver
- Frequency bracketing
- Euler buckling analysis
- Guyan reduction with automatic or user defined masters

### **Nonlinear Analysis**

- Incremental solutions with iterative correction
- User defined combination of full or modified Newton Raphson iterations with line searches
- Automatic arc length solution procedures with option for non-proportional loading
- Automatic recovery upon convergence failure
- Load or displacement control
- Wide selection of convergence criteria
- Large deformation, large rotation geometric nonlinearities
- Large strains
- Follower loads
- Element birth and death facility
- Centripetal stress stiffening

- Temperature dependent material properties

### **Dynamic Analysis**

- Forced response analysis
- Modal (viscous or structural) or Rayleigh damping
- Response spectrum analysis with a choice of SRSS and CQC spectral combinations
- Modal synthesis analysis using superelements
- Implicit transient dynamic analysis using Hilber-Hughes-Taylor time integration scheme
- Explicit transient dynamic analysis using central difference time integration scheme
- Initial velocity/acceleration input
- Implicit and explicit impact
- Linear and nonlinear dynamic analysis
- Automatic time step selection
- Time dependent material properties
- Time dependent loading
- Interactive Model Dynamics option for multiple loading events and advanced loading options

### **Thermal Analysis**

- Steady state heat conduction/convection/radiation
- Transient thermal analysis with a general two point recurrence scheme
- Temperature dependent thermal properties
- Temperature dependent nonlinear heat conduction/convection/radiation
- Variable time step selection
- Conduction/convection/gap radiation
- Diffuse radiation using view factor with option to account for symmetry boundary conditions
- Full and semi thermal-structural coupling

### **Boundary Conditions**

- Choice of restrained, prescribed or spring boundary conditions
- Transformed freedom option for skew boundary conditions
- Time dependent boundary conditions and loading
- Nonlinear friction and gap models to represent deformation dependent boundary conditions and contact problems
- Slideline/slidesurface contact algorithms for use with implicit/explicit plane stress/strain, Axisymmetric, shell and solid elements
- Convection and nonlinear radiation boundary conditions
- Tied slidelines to connect incompatible meshes
- Automatic pre-contact algorithm
- Contact cushioning
- Curved surface contact

## **Results features**

### **Results Processing**

Comprehensive results processing from multiple analysis runs

Processing of averaged or element nodal and Gauss point stresses, strain results

Strain energy, plastic work, rubber stretches and user defined state variables

Internal results calculator

Automatic calculation of moments and forces from solids elements

Basic and Smart load combination facilities

Enveloping with option for different factors for positive and negative effects

Absolute maximum envelopes

Reinforcement calculation (Wood Armer) for plates and shell elements with option to minimise reinforcement areas

Interactive modal dynamics for frequency and time domain solutions

Interactive spectral response analysis

Fatigue response calculation

Report Generator

## Results Display

Window based views allow different results in different windows  
Deformed mesh plots  
Separate viewable layers for diagram, contour, vector and discrete data.  
Animation over multiple load cases/load steps/time steps with AVI output.  
Support for local or global directions for results display  
Contours of displacements, potentials, moments or forces using line or colour fill  
Contouring of joint results  
2D/3D stress and displacement vectors  
Bending moment and force diagrams for beam elements with optional peak values displayed  
Annotation facility  
3D slice facility for solids with optional contour display  
Slice section results for 3D beams and shells  
2D line sections  
Graph plotting facility with export to standard spreadsheet packages  
Display of cracked and yielded Gauss points  
Peak results value display  
Printing and viewing of numerical result values for selected areas and groups in a model  
Full results summary  
High resolution BMP, JPG or WMF output  
Results wizard  
Automatic time history plotting  
User-defined results calculation facility  
Visualisation of slideline/slidesurface forces in global or local directions  
Visualisation of slice section results locations

## Product specific features

### LUSAS Bridge

- Available in LT, Standard and Plus versions
- Grillage wizard
- Country specific bridge loading facilities
- Moving load generator
- Influence line / surface analysis
- Vehicle and Traffic load optimisation to:
  - Australia: AUSTROADS
  - China: JTG D62-2004
  - Ireland: EN1991-2
  - Italy: EN1991-2
  - Finland
  - India
  - Israel
  - Korea
  - Norway
  - Poland: EN1991-2
  - Sweden: EN1991-2, BRO
  - South Africa: TMH7
  - UK: EN1991-2, BD21/97, BD21/01, BD37/88, BD37/01 (Road+Rail), BS5400 Rail Railtrack document RT/CE/025
  - USA: AASHTO LFD, LRFD
- RC slab design wizard
- Beam stress checking facility
- Single and Multiple stage prestress wizard to:
  - AASHTO-LRFD
  - BS5400
  - Eurocode EN1992
  - JTG D62-2004.

### **LUSAS Civil & Structural**

- Available in LT, Standard and Plus versions
- RC slab design wizard
- Beam stress checking facility
- Single and Multiple stage prestress wizard to:
  - AASHTO-LRFD
  - BS5400
  - Eurocode EN1992
  - JTG D62-2004.

### **LUSAS Composite**

- Available Plus version only
- Tsai-Hill, Hoffman, Tsai-Wu (with Cowin extension), and Hashin (fibre and matrix) composite failure modelling

### **LUSAS HPM**

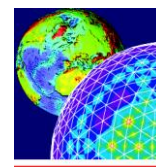
- Available Plus version only (with limited availability)
- Distortion prediction of laminated composite parts during manufacture
- Draping capability
- Materials database of resins and fibres
- Visualisation of fibre direction and skew angles
- Modelling of slip and lift off between Part and Tool during cure simulation.
- Links to third-party interfaces

### **LUSAS Analyst**

- Available in Standard and Plus versions
- Adaptive meshing in 2D for continuum elements
- Design factor plotting

### **LUSAS Academic**

- For research and teaching / training purposes
- Fully functional software with no limit of problem size
- Allows running of any LUSAS software product
- Extensive customisation capabilities through the LUSAS Programmable Interface
- Special interfaces for researchers to incorporate their own research work



**LUSAS**  
Forge House  
66 High Street  
Kingston upon Thames  
Surrey  
KT1 1HN

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