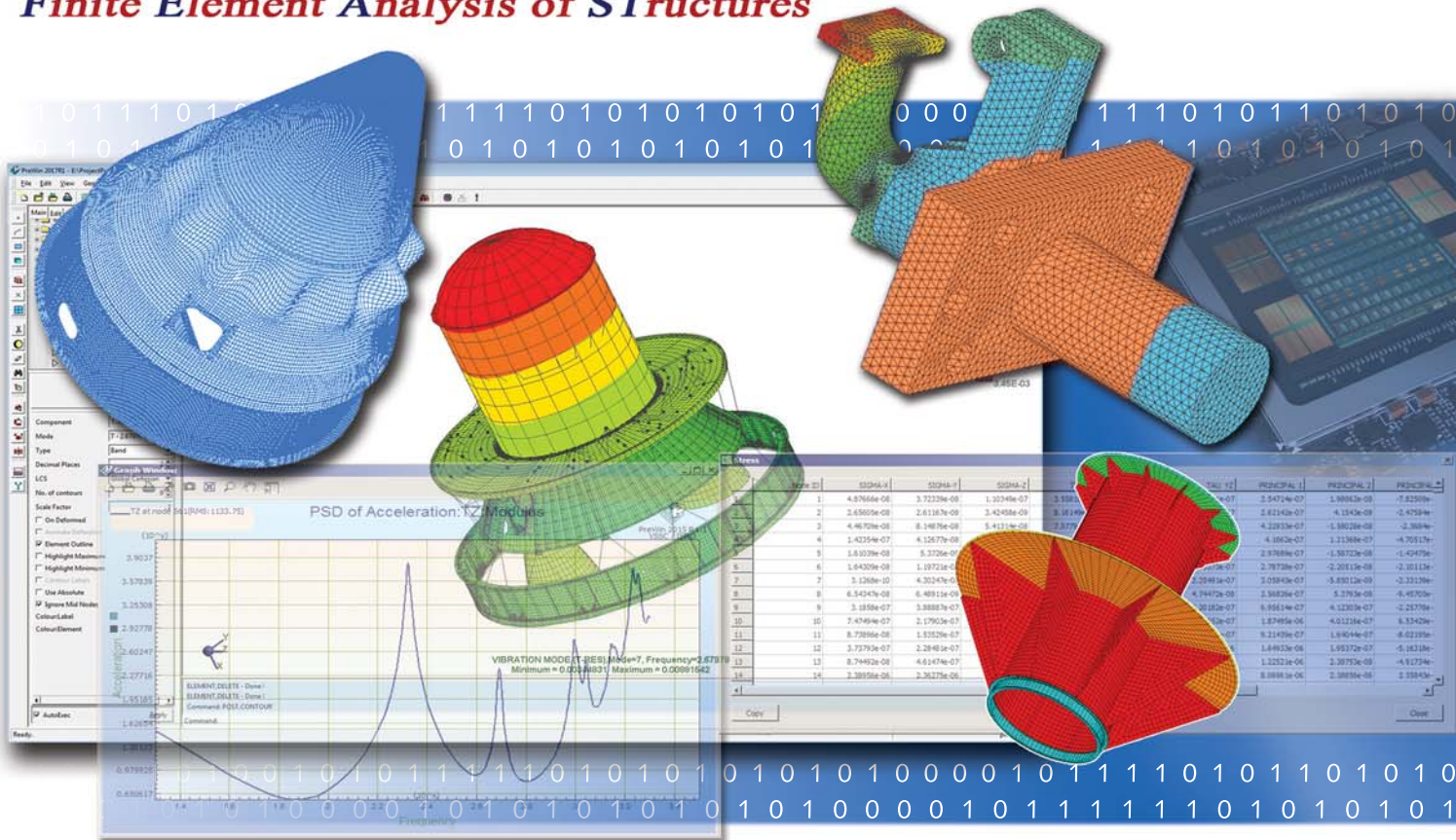


FEAST

Finite Element Analysis of Structures



FEAST (Finite Element Analysis of Structures) is the structural and heat transfer analysis software based on finite element method realized by Vikram Sarabhai Space Centre / Indian Space Research Organisation. It is supported by state-of-the-art pre/post processor - PreWin. Sub-structured and multi-threaded implementation of the solver ensures high performance by exploiting multi-core architecture of modern computing platforms.

The software has Static, Free-vibration, Buckling, Transient, Frequency, Random response and Heat transfer capabilities. These capabilities are supported by a rich element library comprising beam, shell, solid, axisymmetric, spring, rigid-links, gap etc. and can handle isotropic, orthotropic, laminate composites and viscoelastic material models. The above features enable real-life applications of FEAST in structural engineering, solid mechanics and heat transfer problems of Aerospace, Automobile, Civil, Mechanical and Marine engineering. The cost of owning the software is competitive in comparison with similar non-indigenous software packages. The software can be deployed in Linux and Windows operating systems with minimum hardware requirements. It is available in three versions. The classification is based on number of finite element nodes viz; Academic (for students and educational institutions), Premium (for small and medium scale industries) and Professional (for general large scale applications).

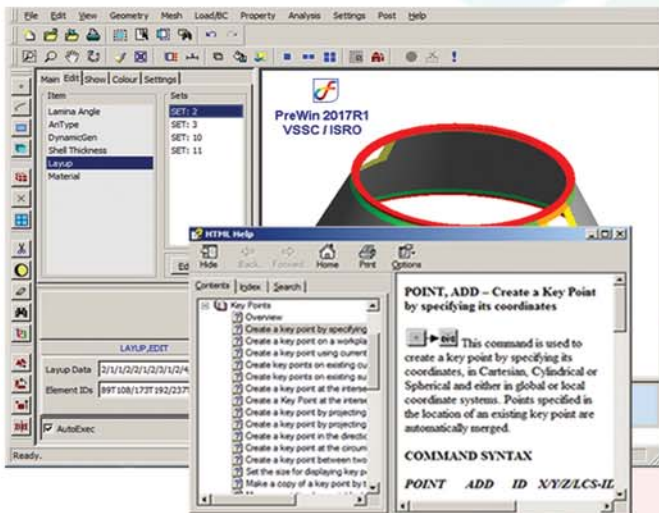




Analysis capabilities

The latest version of FEAST caters to linear and nonlinear analysis capabilities. This is packaged in modular form for academic and commercial usage. It is capable of handling following analyses with metallic and composite material models.

- Linear static
- Free-vibration
- Buckling
- Visco-elasticity
- Thermo-elasticity
- Heat transfer
- Contact analysis
- Transient response
- Frequency response
- Random response
- Shock response / spectra
- Fluid-structure interaction
- Base excitation

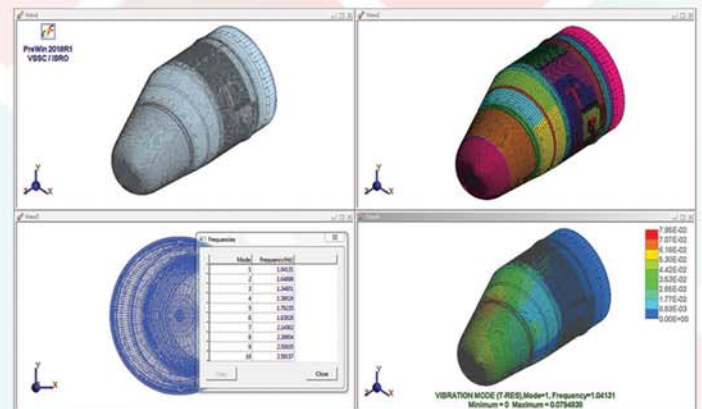


The software can be customised for specific user requirements. Inertia relief method for static analysis of unconstrained structures and Zernike Polynomial computation for assessing optical surface aberrations, are specific cases of customisation.

Material models

Multiple types to represent different material behaviour

- Isotropic
- Orthotropic
- Incompressible
- Layered orthotropic



Graphical User Interface

Reduced learning time and improved efficiency with intuitive GUI

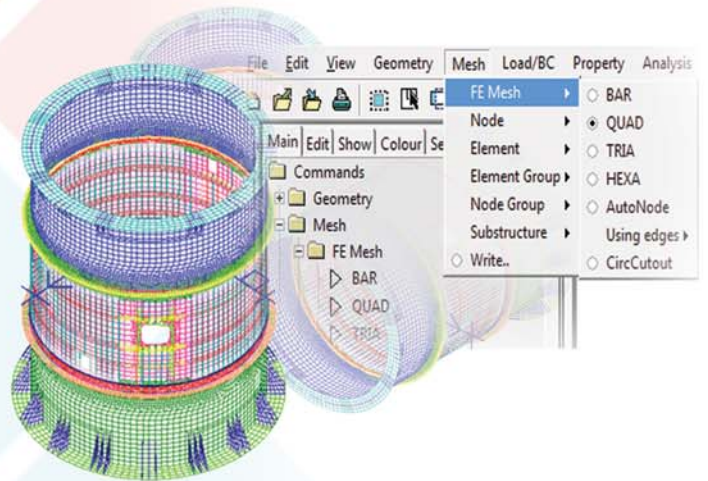
- Multiple ways of command access to create and manipulate models
 - o Command line
 - o Menu driven
 - o Tree based
- 3D visualisation of models
- Seamlessly integrated pre and post processing, data translation and solution
- Multi-port views that are independently controllable
- Context-sensitive help
- Log file generation for every analysis session
- In-built, easy to master scripting language
- Multi-level undo / redo with user settings



Element library

Rich set of element library to model structural engineering and heat transfer problems. It includes 1-D, 2-D and 3-D elements and some of them are

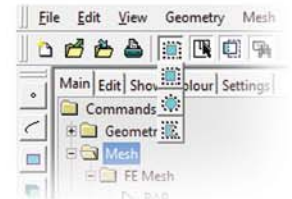
- Beam
- Shell
- Solid
- Axisymmetric
- Plane stress / strain
- Fastener
- Mass
- Gap, glue
- Spring or scalar element
- Rigid link
- Stiffener
- Truss



Selection options

Various options are available for selecting entities from a complex finite element model

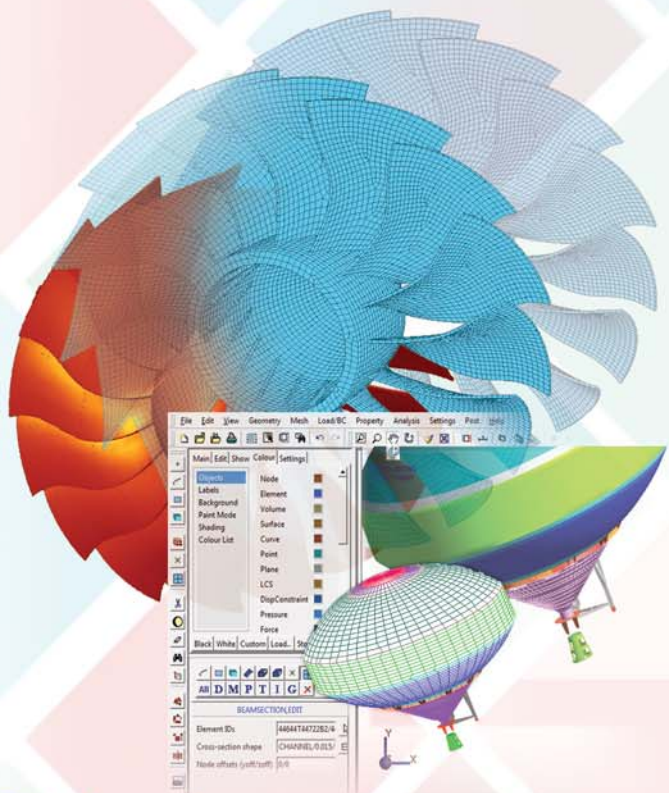
- Rectangular / Circular / Polygonal region
- Pick topmost geometric associativity
- Complementary set
- Part selection
- Element attributes and groups



Model editing

Simplified finite element related editing operations

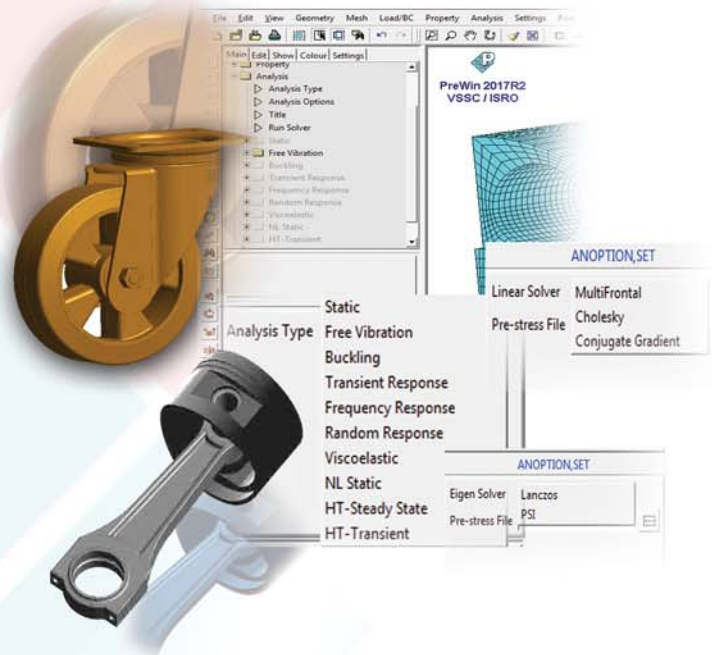
- Convert element types
- Split elements
- Direct editing of all data through EDIT page
- Update or edit loads and local coordinate systems



Solution schemes

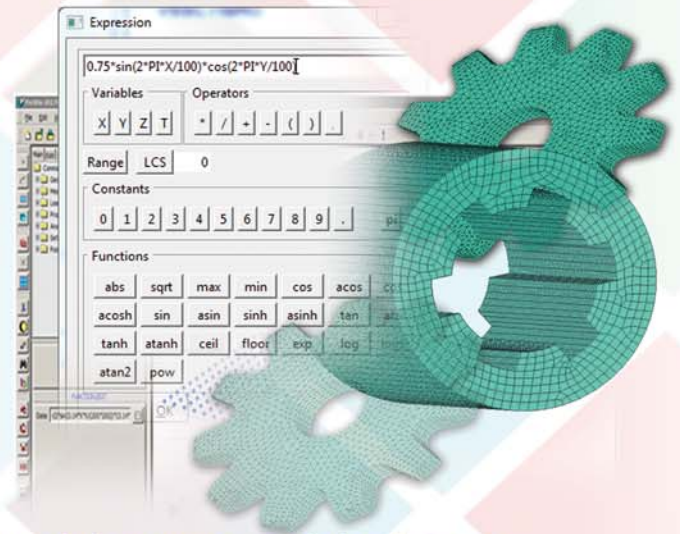
Efficient solution methods for structural engineering / heat transfer problems

- Linear solver
 - o Cholesky solver
 - o Multi-frontal solver
 - o Pre-conditioned conjugate gradient (PCG) solver
- Eigen solver
 - o Lanczos
 - o Sub-space iteration



Loads and boundary conditions

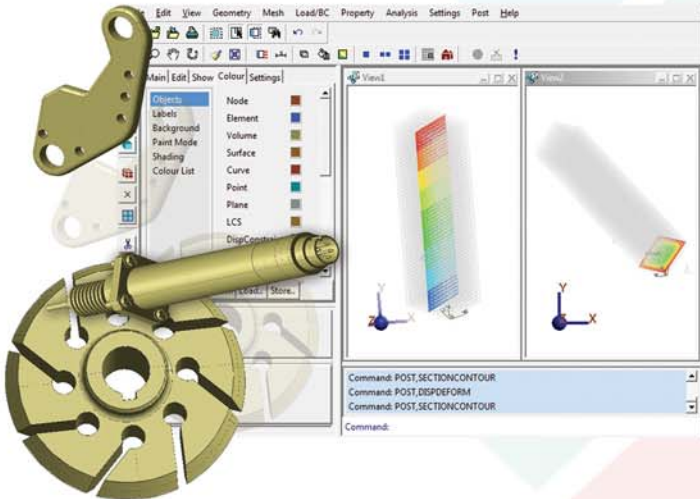
- Single-point constraints
- Multi-point constraint equations
- Prescribed displacements
- Forces and moments at nodes
- Pressure on plate and faces of solid elements
- Temperatures at nodes and source points
- Load application in user specified rectangular, spherical or cylindrical co-ordinate systems
- Define spatial and temporal variation of material and applied loads using graphical user interface based expression editor or tabular data



Model viewing and displaying

Wide range of navigational options for 3D models

- Rotate, zoom, pan
- Independently display
 - o Elements, nodes
 - o Geometry
 - o Loads, boundary conditions
 - o Coordinate systems
- Choose standard or user definable views
- Mask selected entities
- Wire frame, hidden, filled, shaded views
- Moving cross-sectional plots on any arbitrarily oriented plane
- Colour shading based on material, property, type; with option for user specified colour settings



Building geometry

- Large set of geometry modelling tools mimicking CAD software
- Create geometry using points, curves, surfaces and volumes and expand it through following manipulations
 - o Translation
 - o Rotation
 - o Reflection
 - o Scaling
 - o Sweeping
 - o Extrusion
 - o Projection
- Import of CAD models in IGES / STEP format





Working with finite element mesh

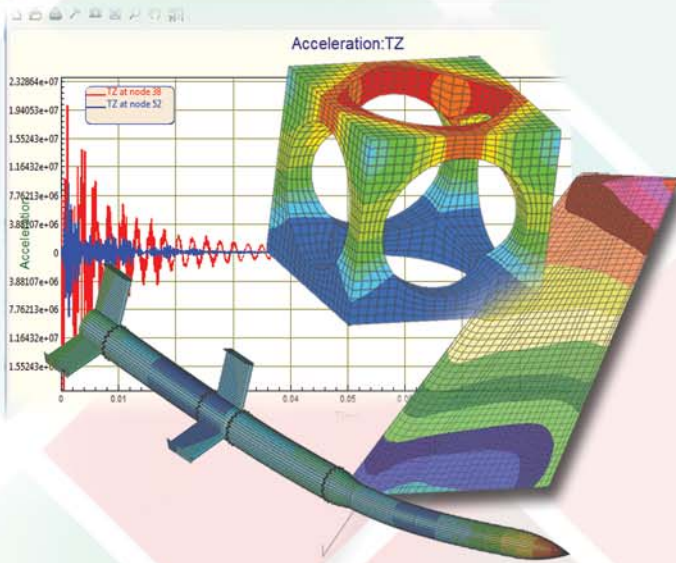
Initial finite element mesh generated using geometry can be manipulated with following transformations and commands for desired mesh

- Translation
- Rotation
- Reflection
- Scale
- Sweep
- Extrude
- Extrude to surface
- Project
- Convert
- Break
- Offset
- Cut-out

Data translators

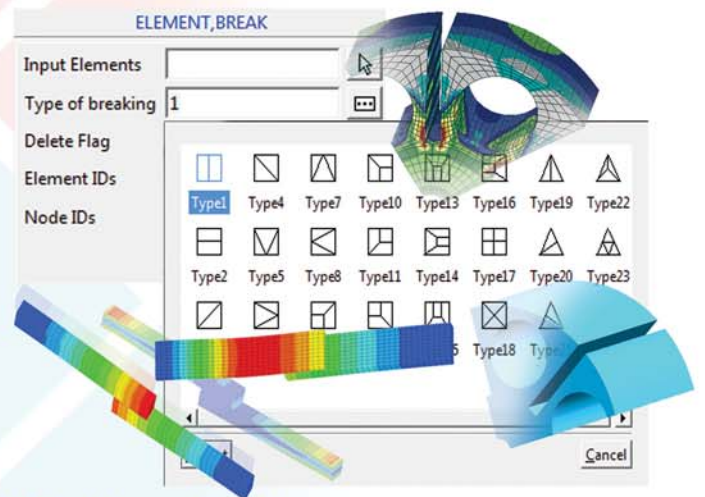
Data translators for importing / exporting model data from popular commercial finite element software packages

- MSC NASTRAN, MSC MARC
- ANSYS, NISA 2



High performance computing

- Analysis jobs run in parallel on multi-core processors for faster simulation of results
- Exploits substructuring technique to create multiple independent jobs
 - o Static condensation technique for stress analysis (Static analysis)
 - o Based on Craig-Bampton method (Dynamic analysis)



Model verification

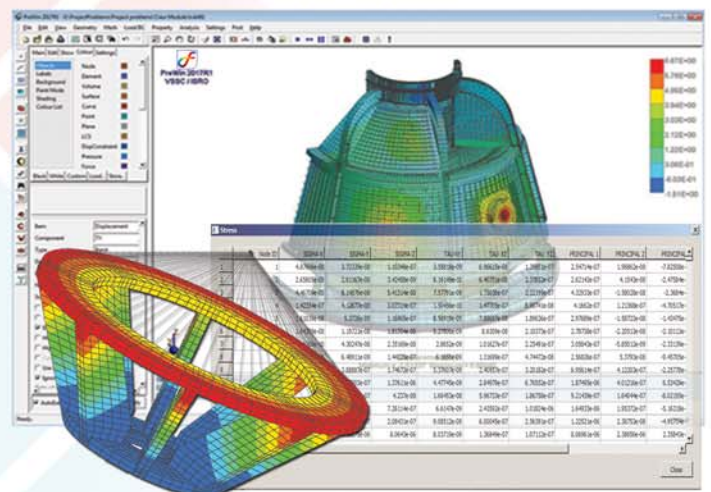
Several options for checking finite element model

- Highlight free edges
- Identify duplicate elements
- Visualise plate warpage
- Check aspect ratio
- Determine bounds of interior angles of elements

Post processing

Multiple ways of visualizing analysis results

- Deformed geometry and contour
- Animation
- X-Y plot (linear, semi-log, log-log) in many documentation standard picture formats
- Vector plot
- Tabulated display: easily transferable to spread sheet applications of standard office suites



Support

- FEAST software is updated continuously with two releases every year
- Training and outreach programmes are conducted periodically and on user request
- Available on Windows and Linux OS
- System configuration: PC / Workstation with a desirable memory of 4GB and above



Contact

Division Head,
Technology Transfer & Industrial Coordination (TT & IC)
Technology Transfer & Documentation Group
Management Systems Area, VSSC
Thiruvananthapuram - 695 022, Kerala, India

Phone: +91 471-256 4081/4207/5006
Fax : +91 471-256 4134
E-mail: feast@vssc.gov.in
Web : <http://feast.vssc.gov.in>



Model • Execute • Visualise



<http://feast.vssc.gov.in>