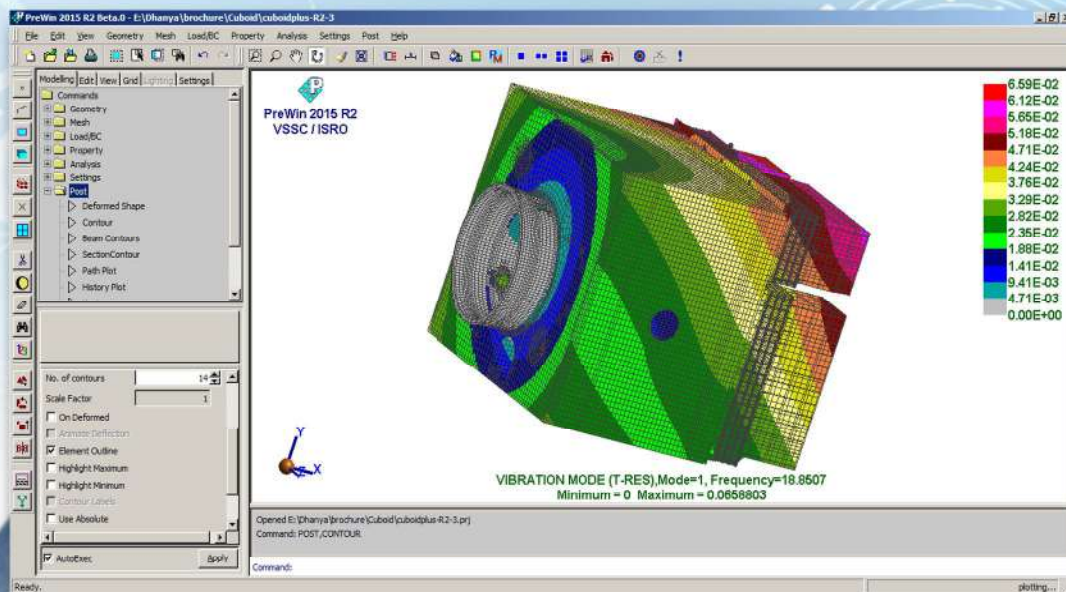# FEAST^{SMT}

Finite Element Analysis of STructures - Substructured and Multi-Threaded

Finite Element Structural Analysis Software Package

FEAST^{SMT} is ISRO's structural analysis software based on Finite Element Method (FEM) realized by Structural Engineering Entity of Vikram Sarabhai Space Centre (VSSC). The software is developed using C++ programming language based on Object Oriented software development principles.

PreWin is the Graphical User Interface (GUI) based pre and post processor of FEAST^{SMT}. It provides state of the art features for geometric modelling, mesh generation, model editing and results visualization.

FEAST^{SMT} solver is seamlessly integrated with PreWin as a single application. Substructured and Multi-Threaded (SMT) implementation of the solver ensures high performance by exploiting multi-core architecture of modern computing platforms. With advanced solution algorithms the solver is able to handle large order problems of structural engineering.

Analysis capabilities

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

- Linear static
- Free-vibration
- Buckling
- Visco-elasticity
- Thermo-elasticity
- Transient response
- Frequency response
- Random response
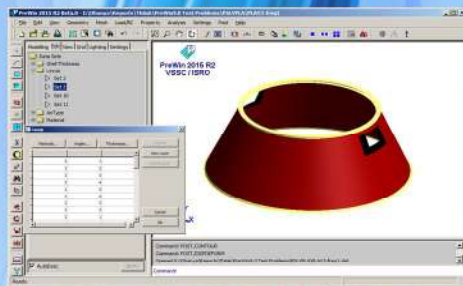- Fluid-structure interaction
- Base excitation

The software is also customized for specific user requirements, like inertia relief method for static analysis of unconstrained structures and Zernike Polynomial computation for analysing optical surface aberrations.
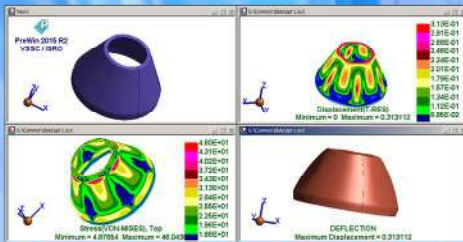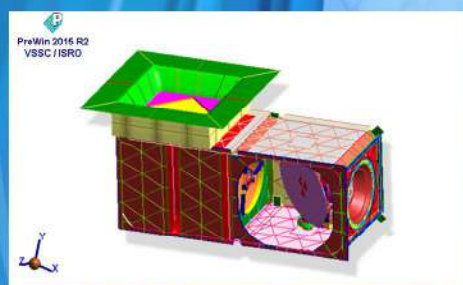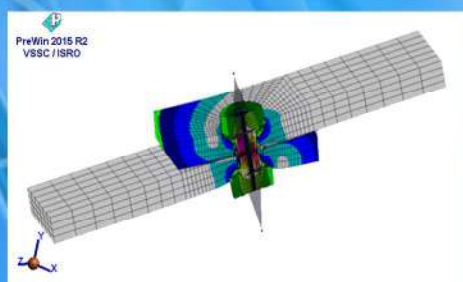
Supports multi-port view
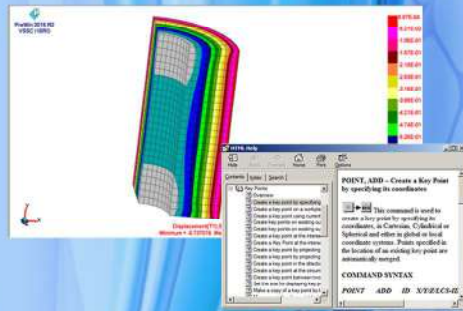
Context sensitive help for all commands

Convenient user interface for composite laminates

Colour shading based on attributes

Resultant displacement contour at a lap joint using linear gap element

Graphical User Interface

Reduced learning time and improved efficiency with an intuitive GUI.

- Multiple ways of command access to create and manipulate the models
  - Command line
  - Menu driven
  - Tree based
- 3D visualization of models
- Seamlessly integrated pre and post processing, data translation and solution.
- Context-sensitive help
- Log file generation
- Inbuilt scripting language
- Multilevel undo / redo

Element library

Rich set of element library to model wider class of engineering problems

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

Material models

Multiple types to represent different material behaviour suitable for linear analyses

- Isotropic
- Orthotropic
- Incompressible
- Layered orthotropic

Solution schemes

Efficient solution methods for static and dynamic structural problems

- Linear Solver
  - Cholesky solver
  - Multifrontal solver
  - Pre-conditioned conjugate gradient (PCG) solver
- Eigen Solver
  - Lanczos
  - Subspace iteration

Loads and boundary conditions

- Single point constraints
- Multi point constraint equations
- Prescribed displacements
- Nodal forces and moments
- Pressure on plate and solid elements
- Nodal temperatures
- Load application in user specified rectangular, spherical or cylindrical co-ordinate systems
- Define material and loading properties with functions

Model viewing and displaying

Wide range of navigational options for 3D models

- Rotate, zoom, pan models
- Independently display
  - Elements, Nodes
  - Geometry
  - Loads, Boundary conditions
  - Coordinate systems
- Choose standard or user definable views
- Mask selected entities
- Wire frame, hidden, filled, shaded views
- Independent management of multiple views
- Moving cross-sectional plots
- Colour shading based on material, property, type, assigned colours (attribute based)
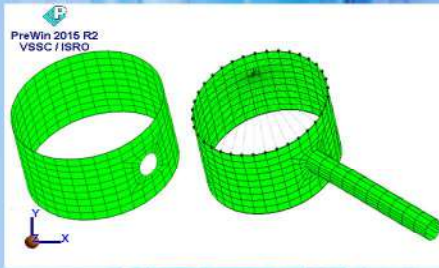
Selection options

Various options available for selecting entities from a complex FE model

- Rectangular / Circular / Polygonal region
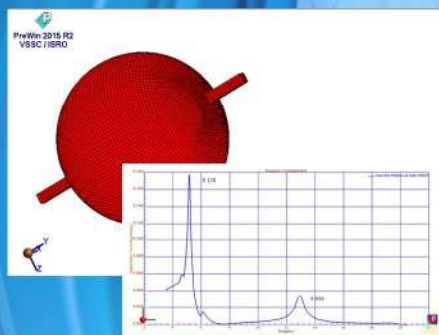- Pick topmost
- Geometric associativity
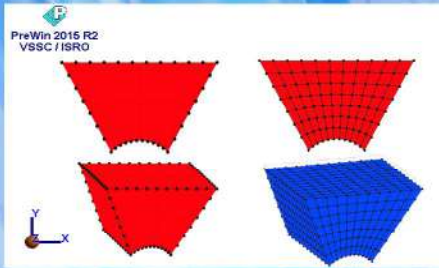
User friendly mesh generation capabilities

Direct creation of cut-outs on FE mesh and extruding boundary edges

State-of-the-art post processing and visualization capability

Result visualization in response analysis

List of natural frequencies from free vibration analysis

- Complementary set
- Part selection
- Element attributes and groups

Building geometry model

- Large set of modelling tools mimicing CAD software.
- Geometrical model is created using points, curves, surfaces and volumes and expanded through the following manipulations
  - o Translation
  - o Rotation
  - o Reflection
  - o Scaling
  - o Sweeping
  - o Extrusion
  - o Projection

Working with FE mesh

Initial FE mesh is generated on the geometry and further manipulated through the following transformations and commands

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

Model editing

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

Model checking

Several options for model verification

- Highlight free edges

- Identify duplicate elements
- Visualize plate warpage
- Check aspect ratio
- Determine bounds of interior angles of elements

Mesh generation

Supports mapped and automatic meshing techniques
- Bar2, Bar3
- Quad4, Quad8, Quad9
- Tria3, Tria6
- Hexa8, Hexa20, Hexa27
- Tetra
- All QUAD automesh

Data translators

Data translators for importing / exporting finite element models of the popular commercial FE packages
- NASTRAN, MARC
- ANSYS
- NISA 2

Post processing

Multiple ways of visualizing the analysis results
- Deformed geometry
- Contour
- Animation
- X-Y plot
- Vector plot
- Tabulated display

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
- Static Substructuring
  - o Based on static condensation technique for stress analysis
- Dynamic Substructuring
  - o Based on Craig-Bampton method

Support

- The software is updated regularly ensuring two releases every year
- Training programmes are conducted periodically
- Theoretical and user manuals are provided
- Available on Windows and Linux OS
- System configuration: PC / Workstation with a desirable memory of 4GB

Contact

Deputy Division Head,
Technology Transfer & Industrial Coordination (TT & IC)
Technology Transfer & Documentation Group
Management Systems Area
Vikram Sarabhai Space Centre
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 ∎ Visualize