OBJECT ORIENTED DESIGN AND IMPLEMENTATION OF FINITE ELEMENT AND FATIGUE ANALYSIS SOFTWARE

Outline

Design & Implementation

- Package structure
- Fundamental classes
  - FEA
  - GUI
  - Output

Mechanics of the Software

- Preprocessor
  - Libraries
  - Modeling tools
  - Assignment tools
  - Display tools
- Processor
  - Solvers
- Postprocessor
  - Texture output
  - Visual output

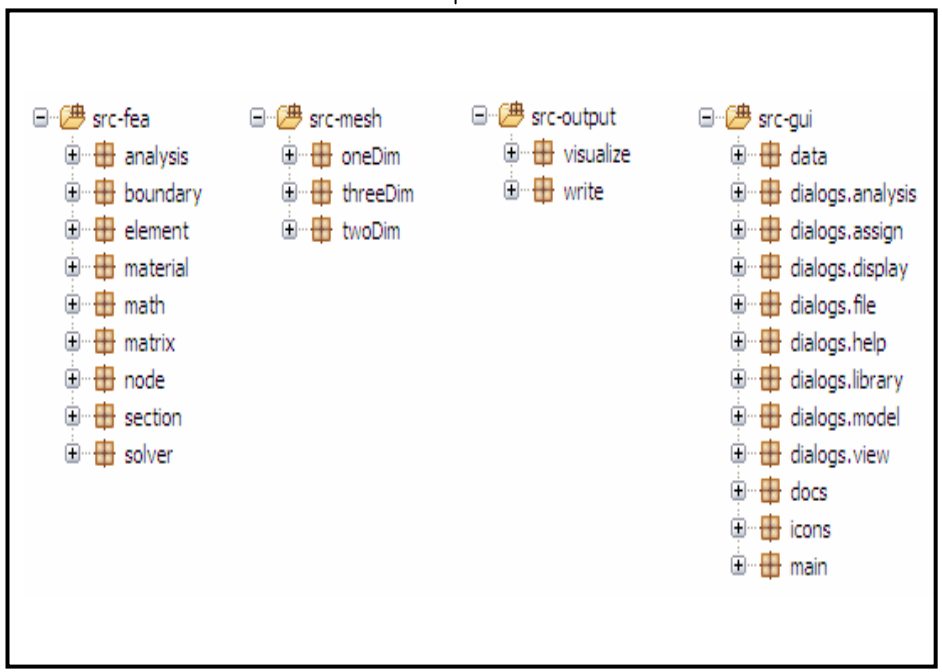
Applications

- Static analysis
- Transient analysis
- Buckling analysis
- Modal analysis
- Fatigue analysis

DESIGN AND IMPLEMENTATION

Package structure

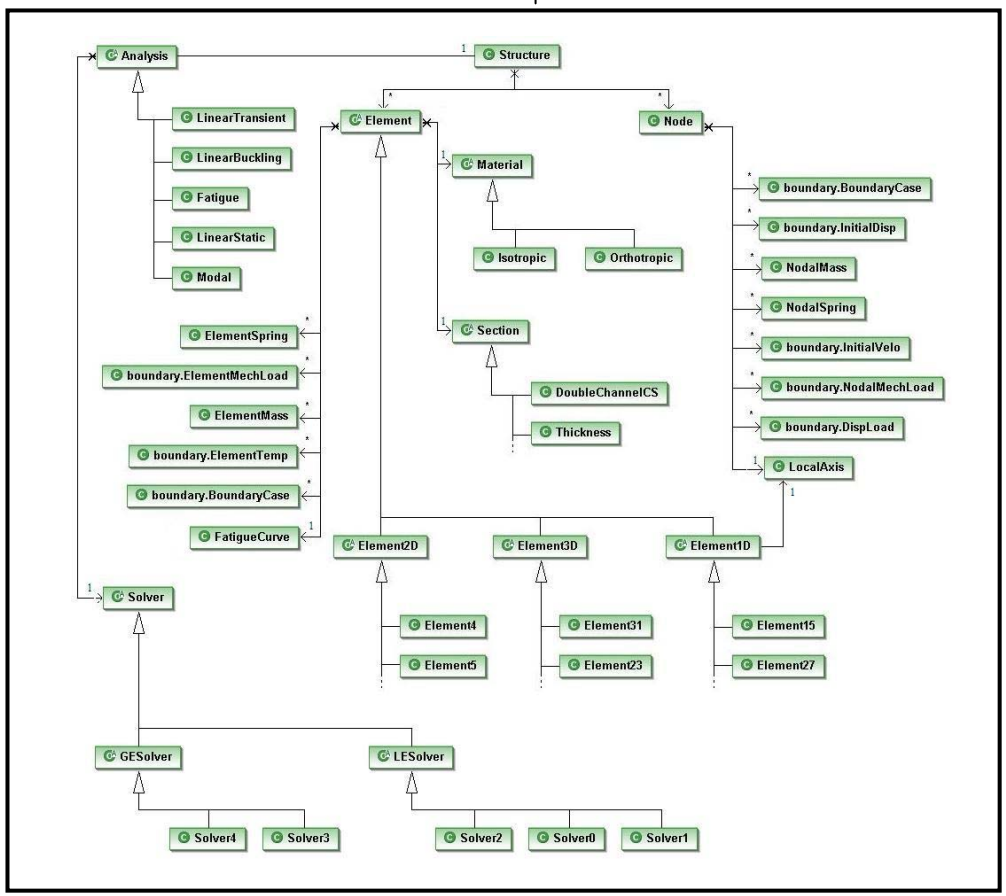
- Languages
  - Java (Analysis, GUI, Visualization)
  - FORTRAN (Solvers)
  - C++ (Fatigue module)
- Initialization of the project
  - Definition of the problems
  - Capabilities
- Simultaneous development
  - FEA
  - Mesh generators
  - Output tools
  - GUI



DESIGN AND IMPLEMENTATION

Classes for Finite Element Analysis

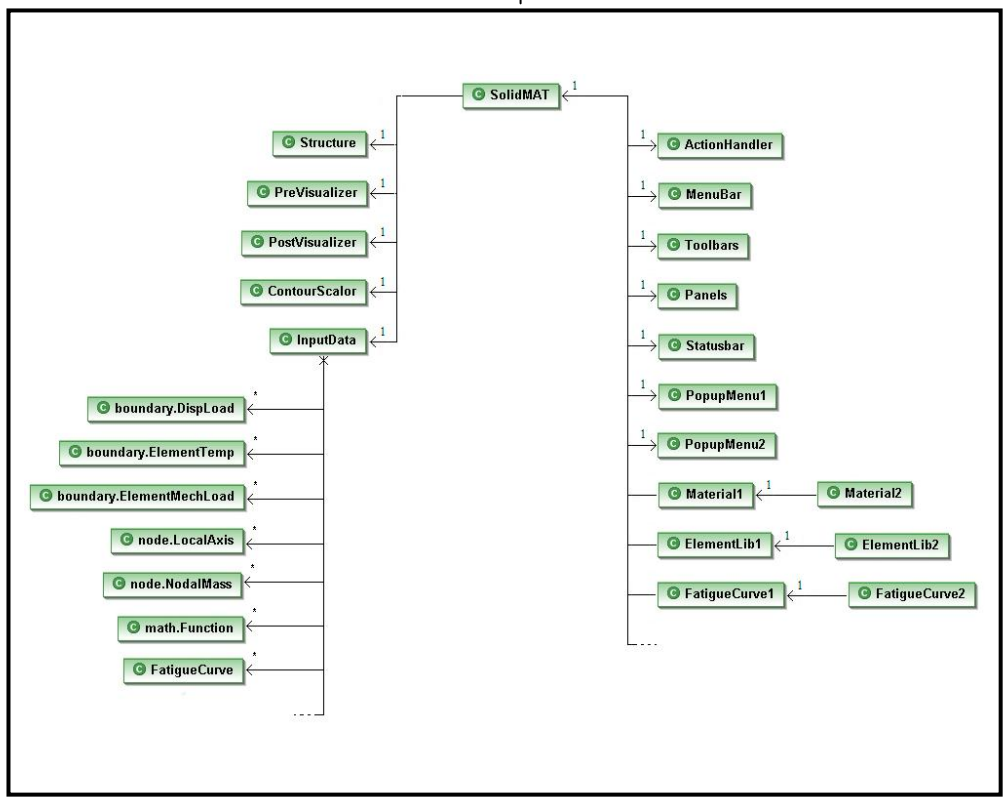
- **Structure class:** Responsible for the collection and management of nodes / elements.
- **Element class:** Super class of all 1D / 2D / 3D elements. Core element formulations take place at the bottom of the hierarchy.
- **Analysis class:** Super class for all analysis classes that include different analysis procedures.
- **Solver class:** Top level class for all linear equation and eigen-system solvers. Core solvers implemented by the use of Java Native Interface.



DESIGN AND IMPLEMENTATION

Classes for Graphical User Interfaces

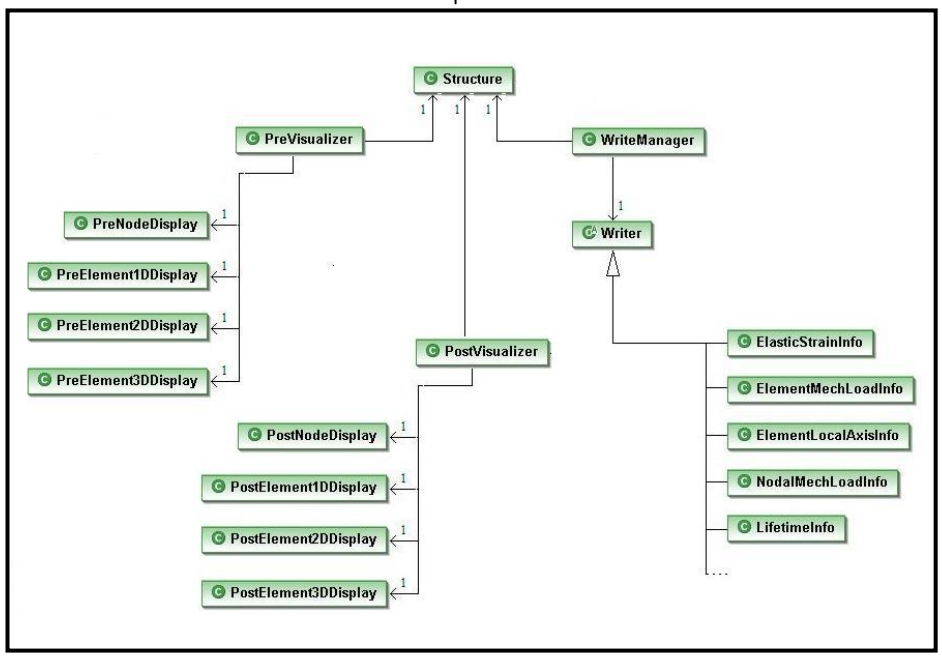
- **MainFrame class:** Main frame of the GUI, relating all other interacting components such as child dialogs, menus, toolbars etc.
- **Dialog classes:** Classes for user dialog interfaces to get user input and store information.
- **InputData classes:** Collect information entered through dialogs and builds input for FEA motor.



DESIGN AND IMPLEMENTATION

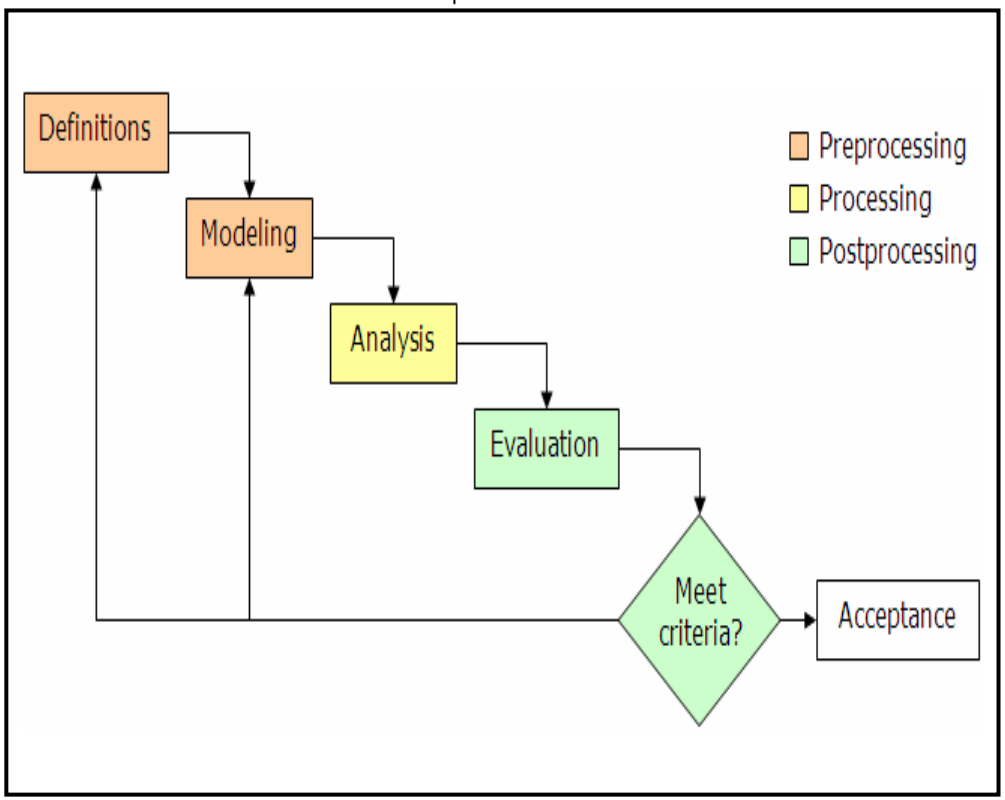
Output classes

- **Previsualizer classes:** Displaying preprocessing information on the graphical area, such as un-deformed shape, assignments and operates for all interactive events triggered by the user.
- **Postvisualizer classes:** Displaying postprocessing information on the graphical area, such as deformed shape, contour plots various results etc.
- **Writer classes:** Write demanded pre- and/or post- processing information on text files.



MECHANICS OF THE SOFTWARE

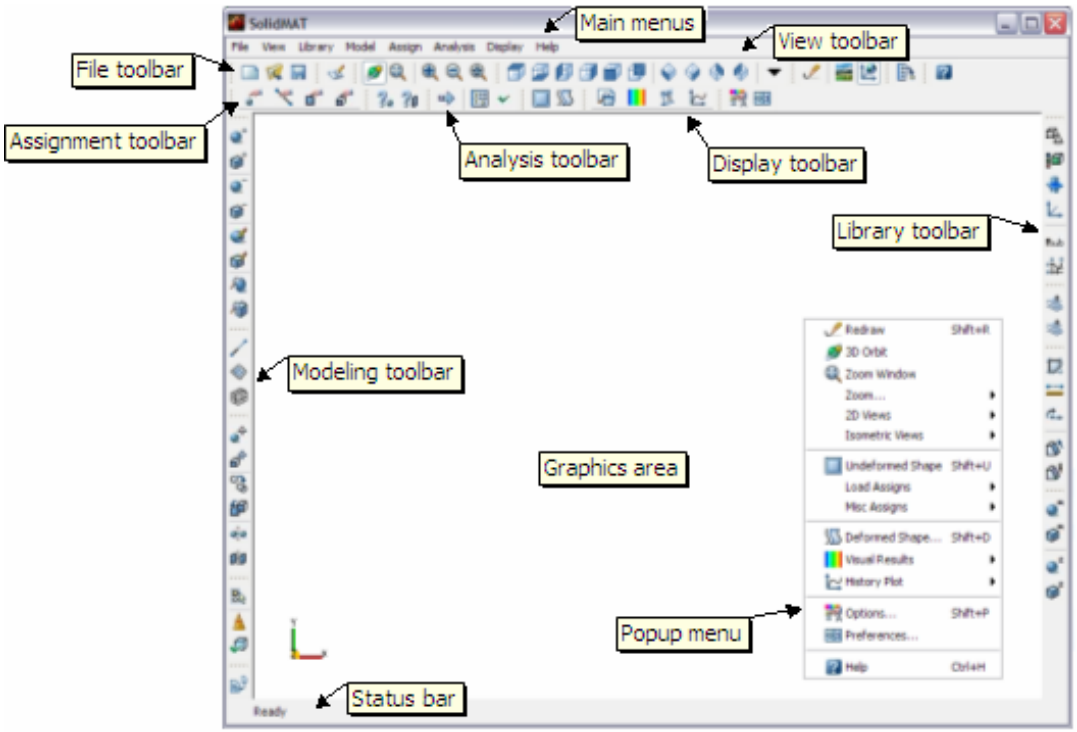
Finite Element Analysis cycle



- The five distinct steps of the Finite Element Analysis cycle provide the foundation for every facility in the software.
- These steps fall under three major units, namely;
  - Preprocessor
  - Processor
  - Postprocessor

MECHANICS OF THE SOFTWARE

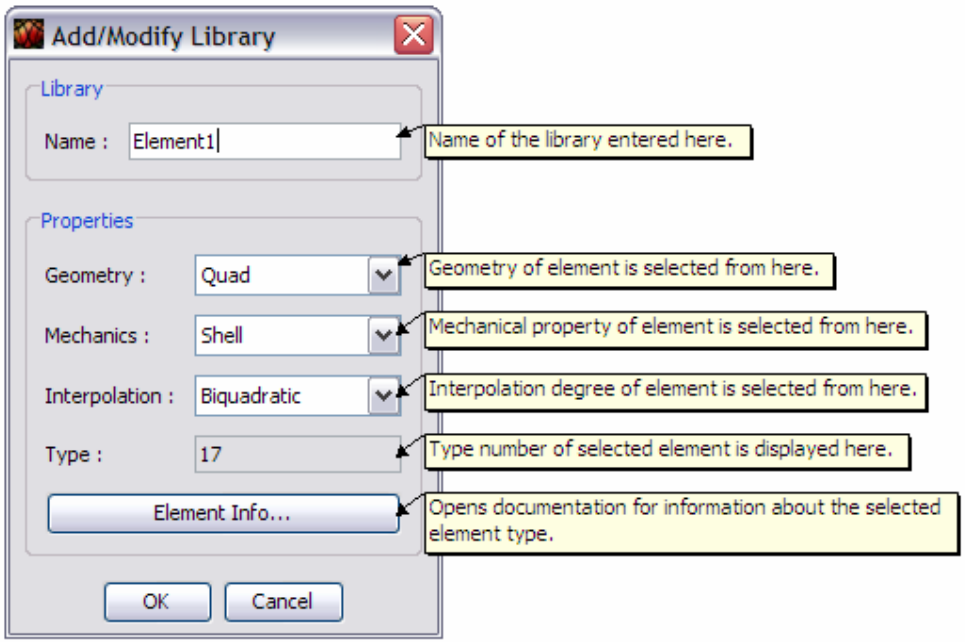
Main frame



- **Main menus** – Contains all commands that are collected under eight menus; File, View, Library, Model, Assign, Analysis, Display and Help.
- **Graphics area** – Used to display the current state of the model.
- **Toolbars** – Provide quick access to most commonly used commands.
- **Popup menu** – Contains most commonly used View and Display commands.
- **Status bar** – Displays the name of currently opened model file.

MECHANICS OF THE SOFTWARE

Preprocessor – Libraries

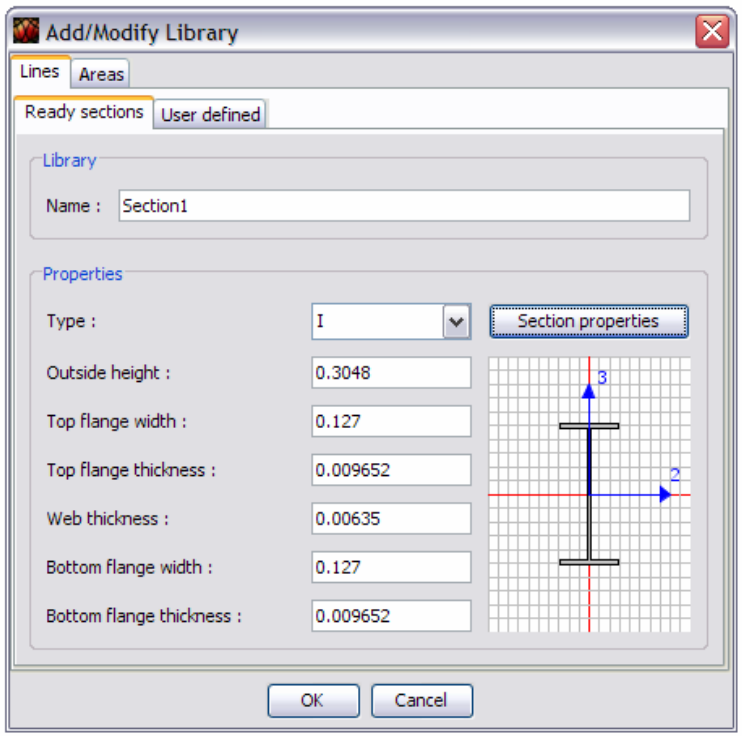


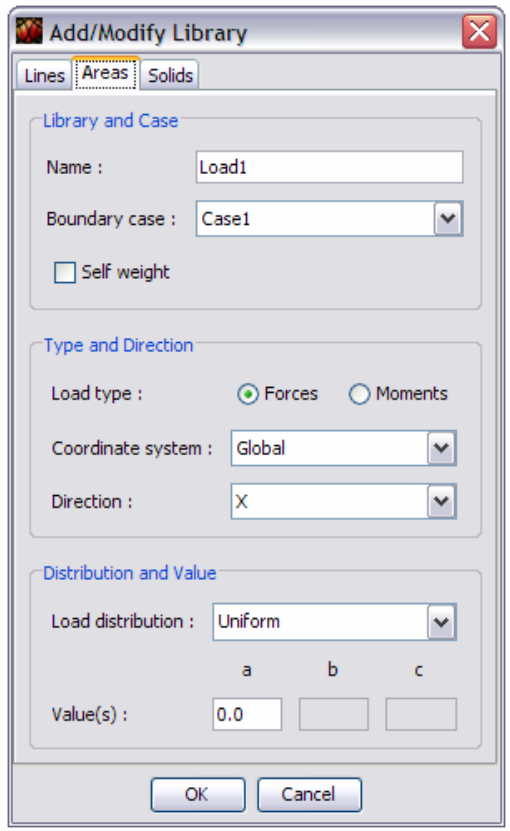
• **Element library:** Contains 32 different types of elements. Provide coverage of truss, beam, plane stress/strain, plate, shell and solid structures.

General Classification			
Type	Geometry	Mechanics	Interpolation
0	Line	Truss	Linear
1	Line	Truss	Quadratic
2	Line	Truss	Cubic
3	Quadrilateral	Plane Stress	Linear
4	Quadrilateral	Plane Stress	Quadratic
5	Triangular	Plane Stress	Linear
6	Triangular	Plane Stress	Quadratic
7	Quadrilateral	Plane Stress	Cubic
8	Quadrilateral	Plane Strain	Linear
9	Quadrilateral	Plane Strain	Quadratic
10	Quadrilateral	Plane Strain	Cubic
11	Triangular	Plane Strain	Linear
12	Triangular	Plane Strain	Quadratic
13	Line	Beam	Linear
14	Line	Beam	Quadratic
15	Line	Beam	Cubic
16	Quadrilateral	Shell	Linear
17	Quadrilateral	Shell	Quadratic
18	Triangular	Shell	Quadratic
19	Quadrilateral	Doubly Curved Shell	Linear
20	Quadrilateral	Doubly Curved Shell	Quadratic
21	Triangular	Doubly Curved Shell	Quadratic
22	Hexahedral	Solid	Linear
23	Hexahedral	Solid	Quadratic
24	Quadrilateral	Plate	Linear
25	Quadrilateral	Plate	Quadratic
26	Triangular	Plate	Quadratic
27	Line	Curved Beam	Linear
28	Line	Curved Beam	Quadratic
29	Line	Curved Beam	Cubic
30	Tetrahedral	Solid	Linear
31	Tetrahedral	Solid	Quadratic

MECHANICS OF THE SOFTWARE

Preprocessor – Libraries



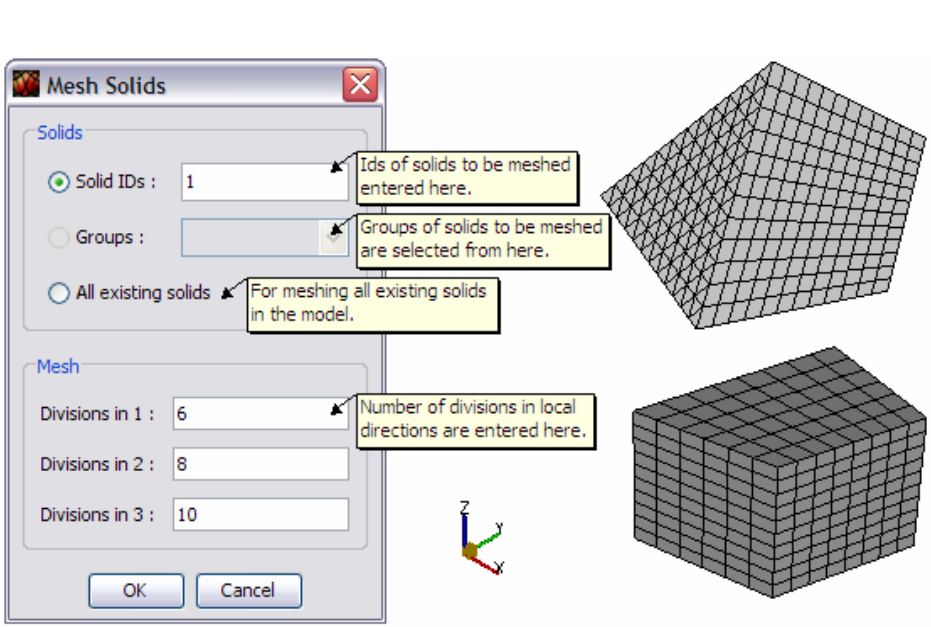


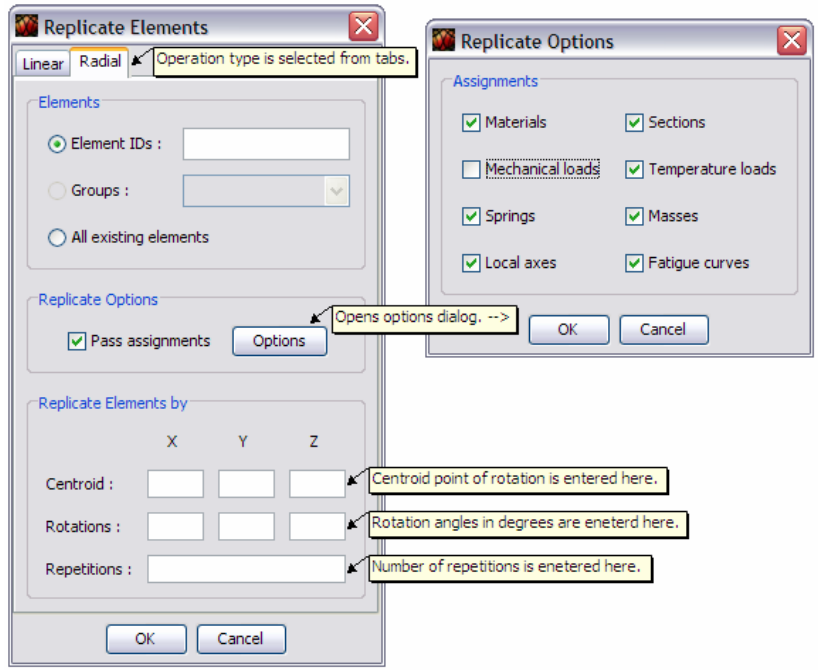
• **Section library:** Line cross-sections can be defined by selecting among the ready sections or by entering the section properties from user defined sections.

• **Mech. Load library:** Distributed line, area and volume loads can be defined as uniform or linearly changing functions.

MECHANICS OF THE SOFTWARE

Preprocessor – Modeling tools



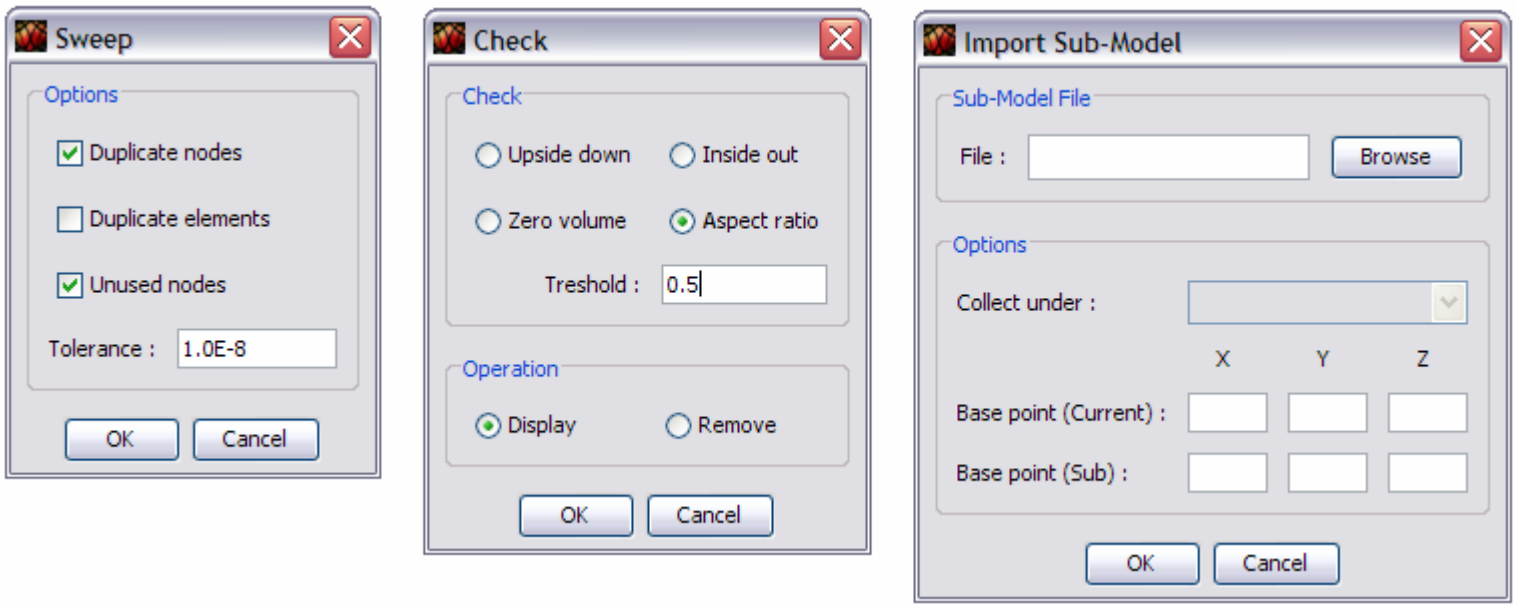


• **Meshing tools:** Element sub-dividers for line, quad, triangular and hexahedral element geometries.

• **Move, Replicate and Mirror tools:** Enable user to create complicated geometries. All operations can be linear or radial.

MECHANICS OF THE SOFTWARE

Preprocessor – Modeling tools



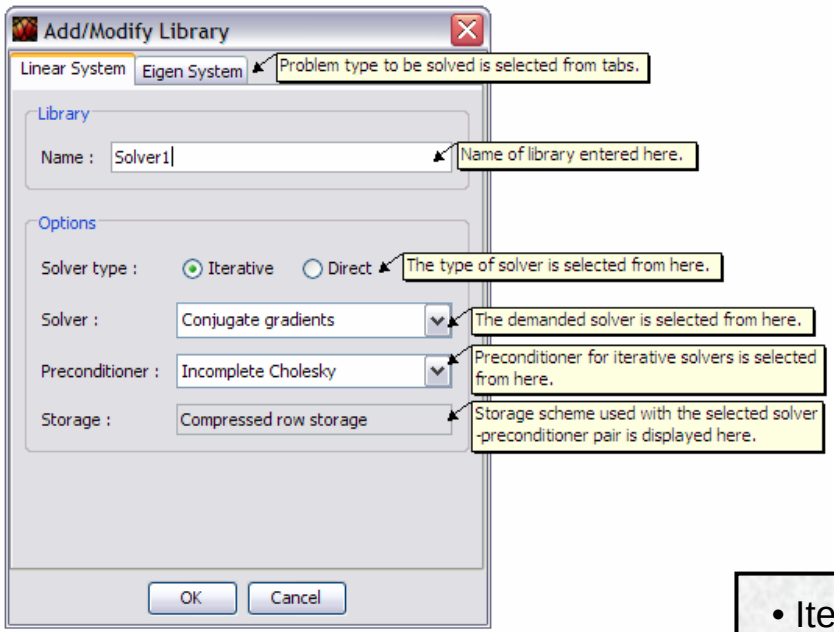
- **Sweep tool:** Used for clearing the model from duplicate nodes / elements and un-connected nodes with the given tolerance.

- **Check tool:** Used for displaying or removing distorted and/or zero jacobian elements.

- **Import tool:** Used for importing other model files or substructures. Enables simultaneous model creation.

MECHANICS OF THE SOFTWARE

Processor – Solvers



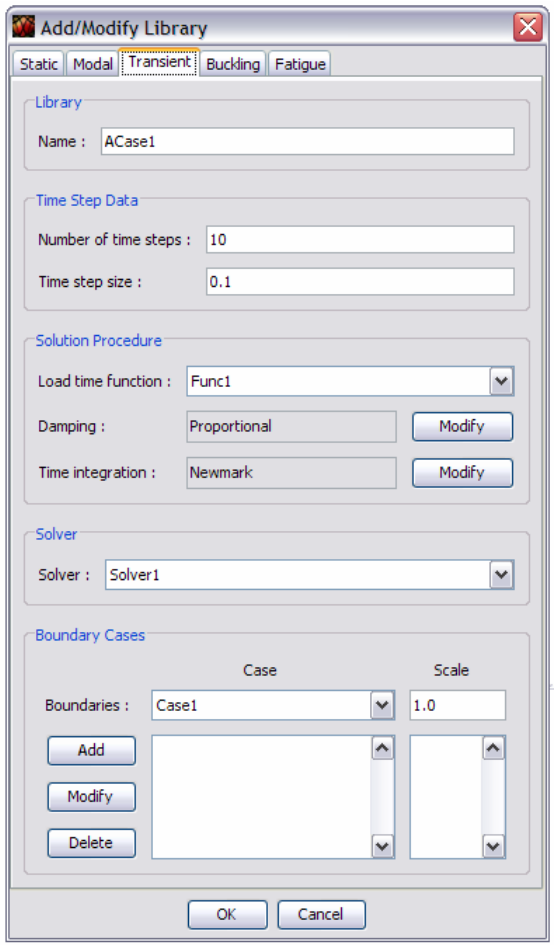
Solver Classifications

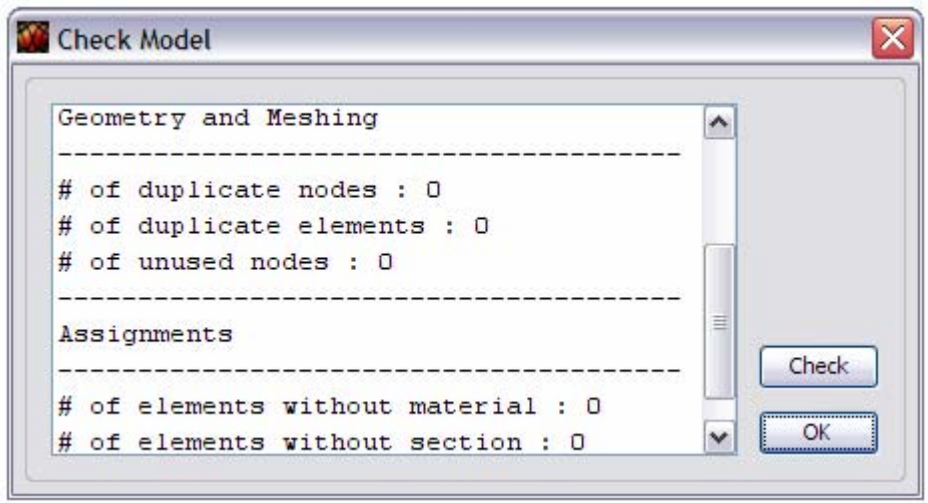
Solver	Purpose	Type	Storage
Conjugate gradients	Linear equation solver	Iterative	Compressed row or diagonal storage
Conjugate gradients squared	Linear equation solver	Iterative	Compressed row or diagonal storage
BiConjugate gradients	Linear equation solver	Iterative	Compressed row or diagonal storage
BiConjugate gradients stabilized	Linear equation solver	Iterative	Compressed row or diagonal storage
Quasi-minimal residual	Linear equation solver	Iterative	Compressed row or diagonal storage
Generalized minimal residual	Linear equation solver	Iterative	Compressed row or diagonal storage
Iterative refinement	Linear equation solver	Iterative	Compressed row or diagonal storage
Active column solver	Linear equation solver	Direct	Upper symmetrical banded 1D storage
Gauss elimination, symmetric	Linear equation solver	Direct	Upper symmetrical banded 2D storage
Subspace iteration	Generalized eigenvalue solver	Iterative	Upper symmetrical banded 1D storage
Direct eigen solver	Generalized eigenvalue solver	Direct	Upper symmetrical packed storage

- Iterative and direct solvers can be used for the solution of linear equation systems and eigen systems.
- Every solver has its own matrix storage scheme for efficient solution time and memory allocation.

MECHANICS OF THE SOFTWARE

Processor – Analysis library and model check tool





- **Analysis library:** Parameters for various analysis types are entered and analysis cases are created through analysis case library.
- **Check model:** Performs final checks for the consistency of the model. No analysis can be initiated if any warning is given during the check.

MECHANICS OF THE SOFTWARE

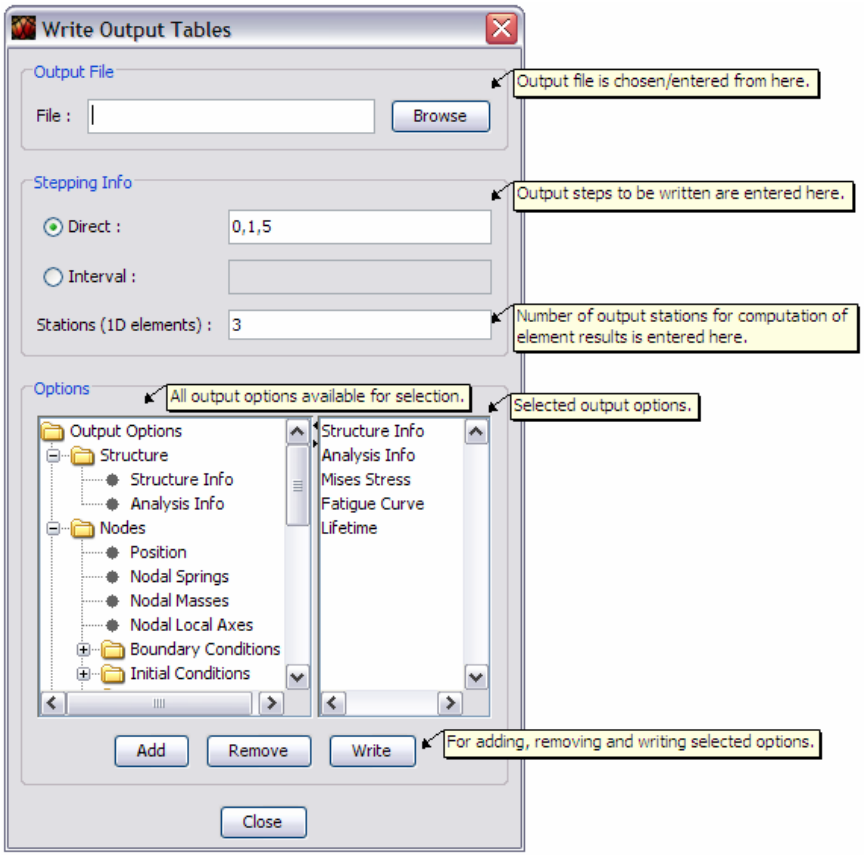
Postprocessor

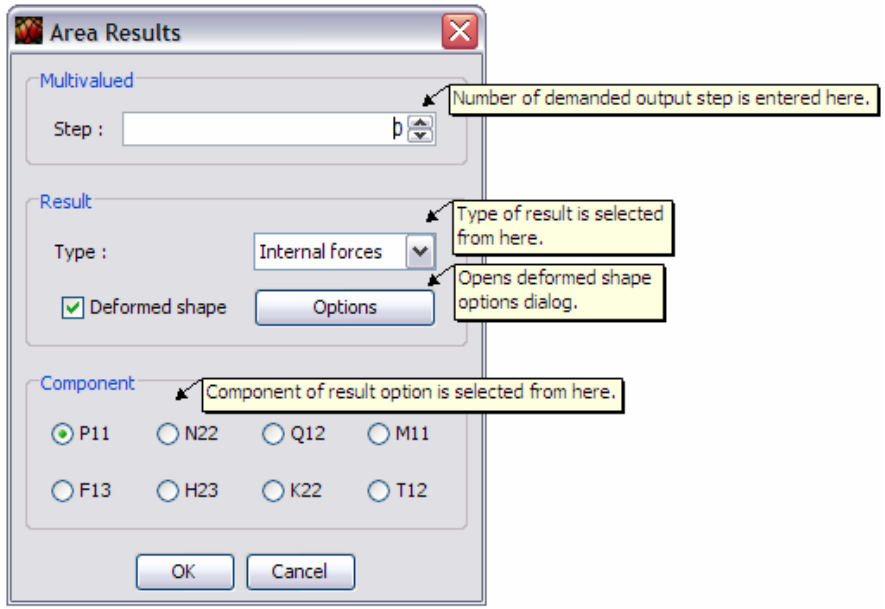
- Visual output tools
- Texture output tools

- **Structure information** composed of number of nodes and elements in the model, volume, mass and weight of the structure.
- **Analysis information** composed of name and type of analysis case, and additional parameters set depending on the analysis type (such as solver library used in the solution, boundary cases and etc.)
- **Deformed shape of the model** for the demanded step number (for multi-valued results)
- **Nodal results** composed of nodal displacements and reaction forces,
- **Element results** composed of element local displacements, elastic strains, stresses, internal forces, principal strains, principal stresses, Mises stresses and lifetime values.
- **Time-history plot** composed of any element result derived-collected and plotted for the demanded number of time steps and element. This option is available only for linear transient and fatigue analysis cases.

MECHANICS OF THE SOFTWARE

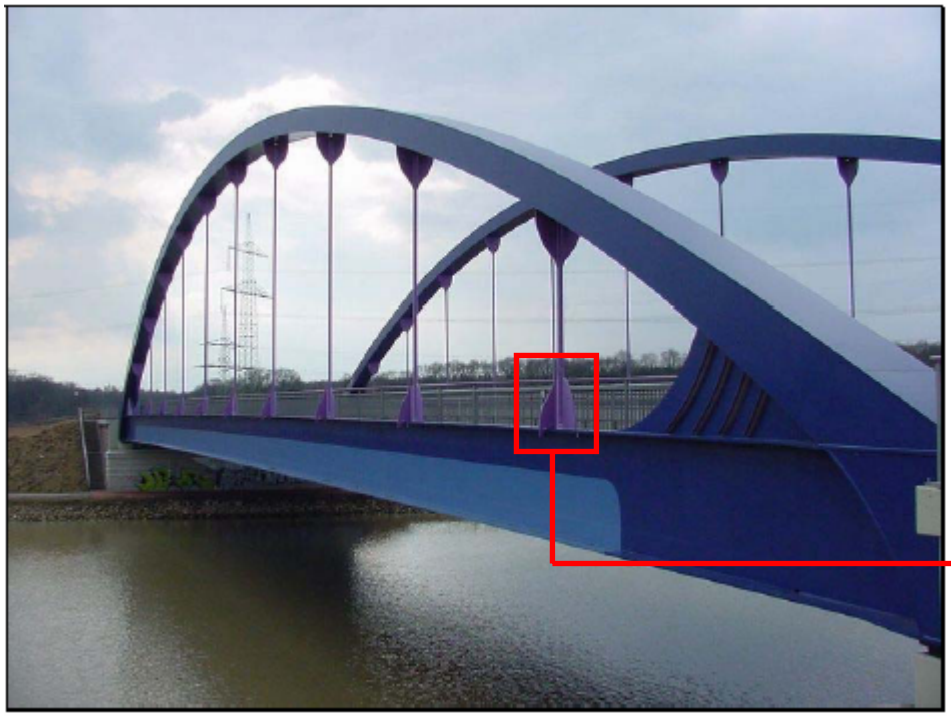
Postprocessor

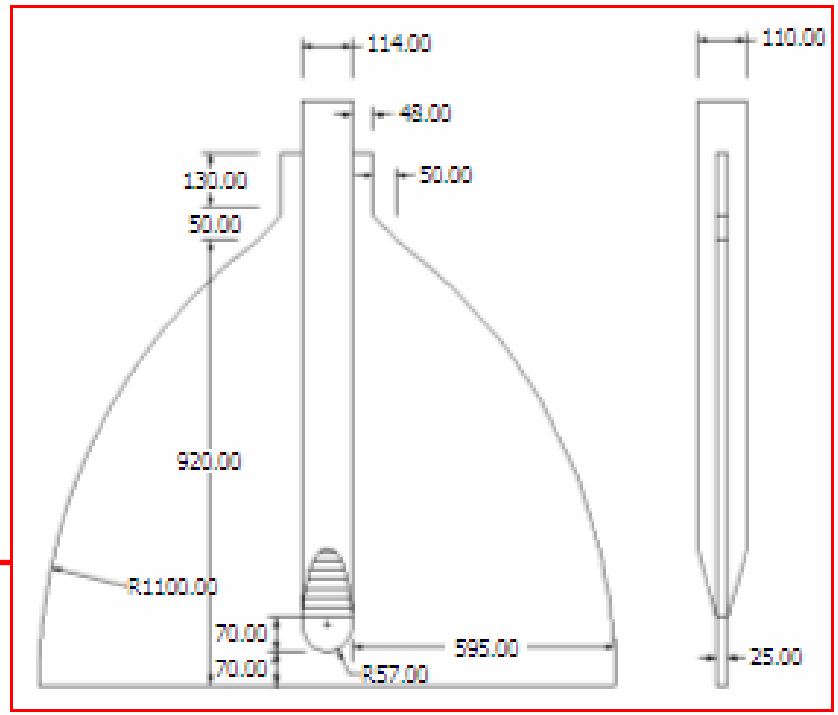




- Element results are computed on the given stations (1D elm.), corner node positions (2D elm.), vertex positions (3D elm.).
- Since only nodal displacements are stored on the output file, the derivatives of the solution are derived before visualization.

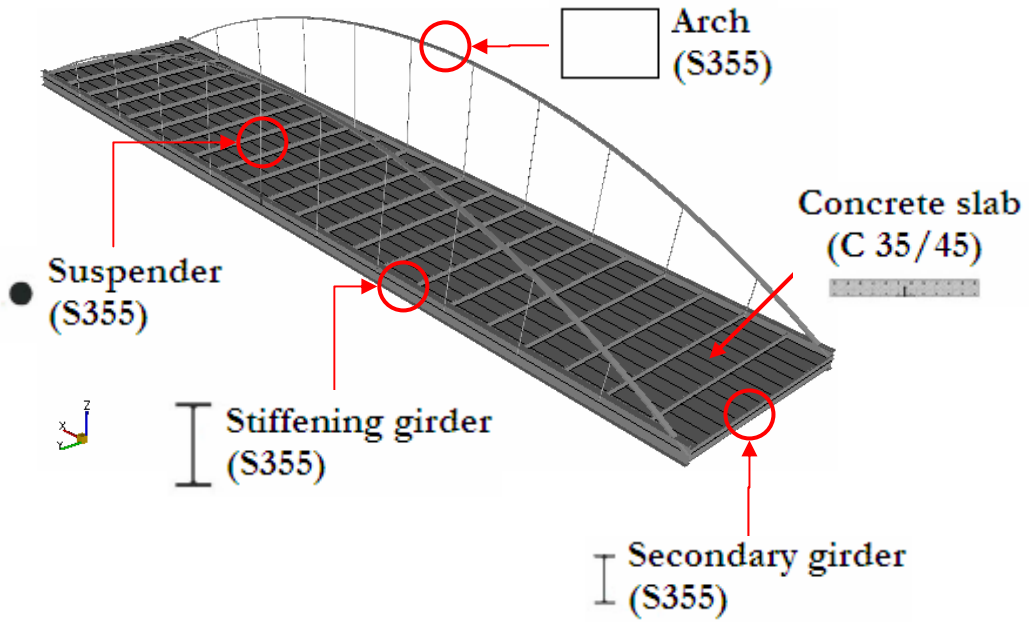
HANGER PLATES OF AN ARCHED BRIDGE



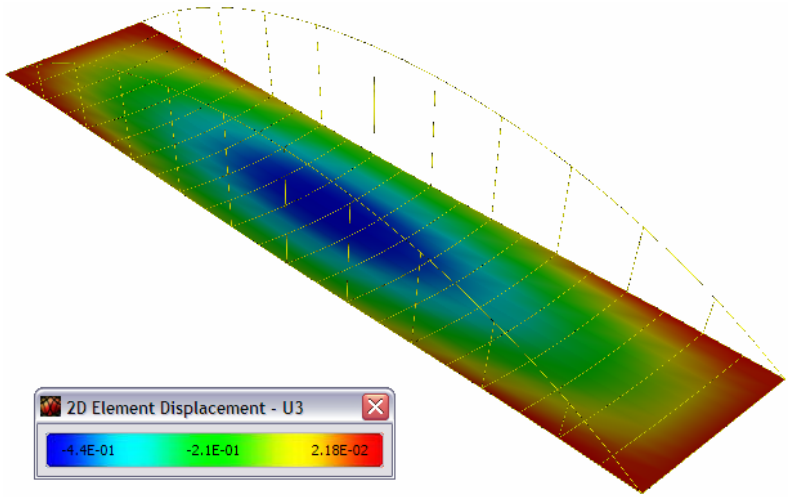


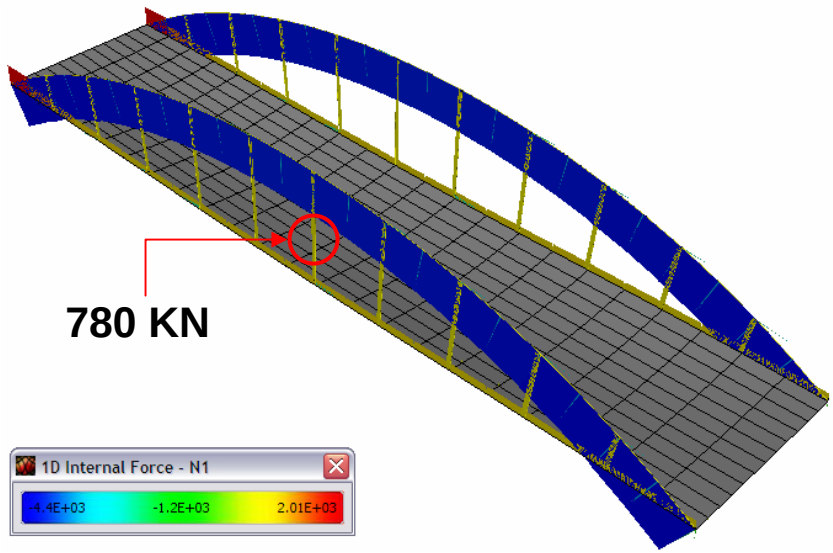
OBJECTIVE : LIFETIME ESTIMATION OF VERTICAL BRIDGE TIE RODS EXPOSED TO WIND-INDUCED VIBRATIONS

➢ DETERMINATION OF TENSION LOADS ON TIE RODS

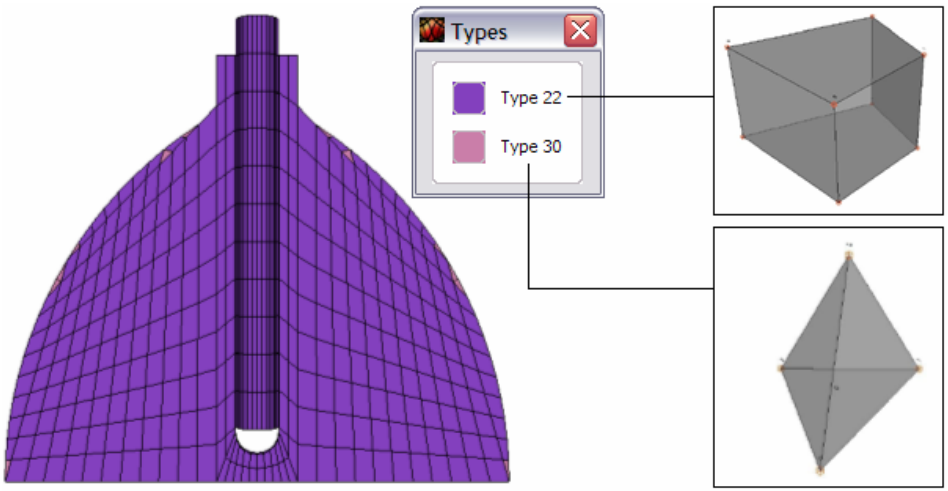


- ➢Linear static analysis
- ➢Dirichlet boundaries
  - •UX, UY, UZ restrained (x=0 edge)
  - •UY, UZ restrained (x=85 edge)
- ➢Neumann Boundaries
  - •Self weight
  - •Traffic loads
  - •Wind loads (distributed on arch and stiffening girders)



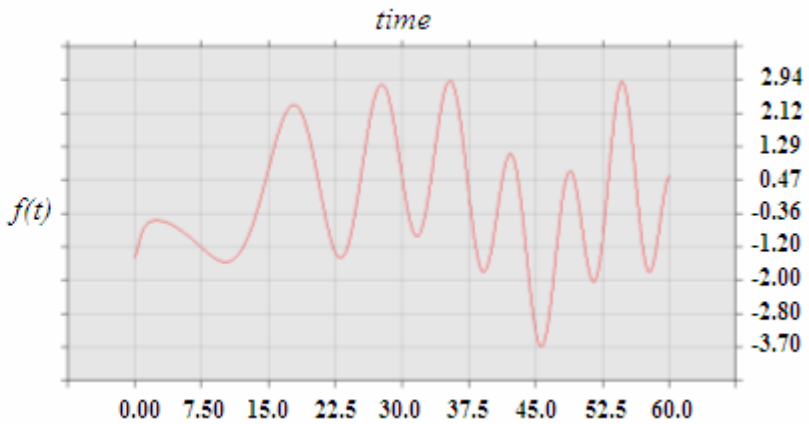


➢ MESH AND BC'S FOR THE ANALYSIS OF HANGER PLATES

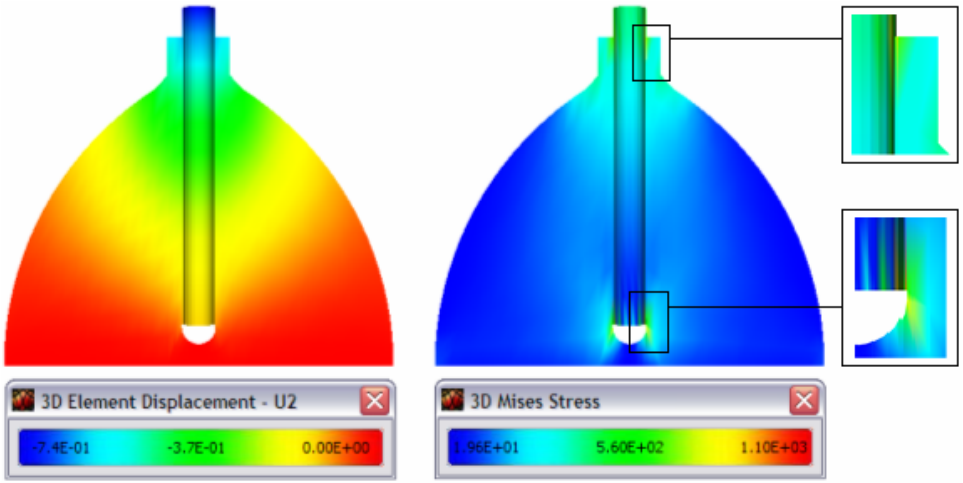


- ➢Eight-node hexahedral (*Element 22*) and four-node tetrahedral (*Element 30*) solid elements.
- ➢Assumed strain formulation used for overcoming possible locking behavior.
- ➢All nodes that lie at the bottom part of the plate section are *pinned*.
- ➢Variable amplitude Riemann-Siegel time function has been generated for wind vibration.

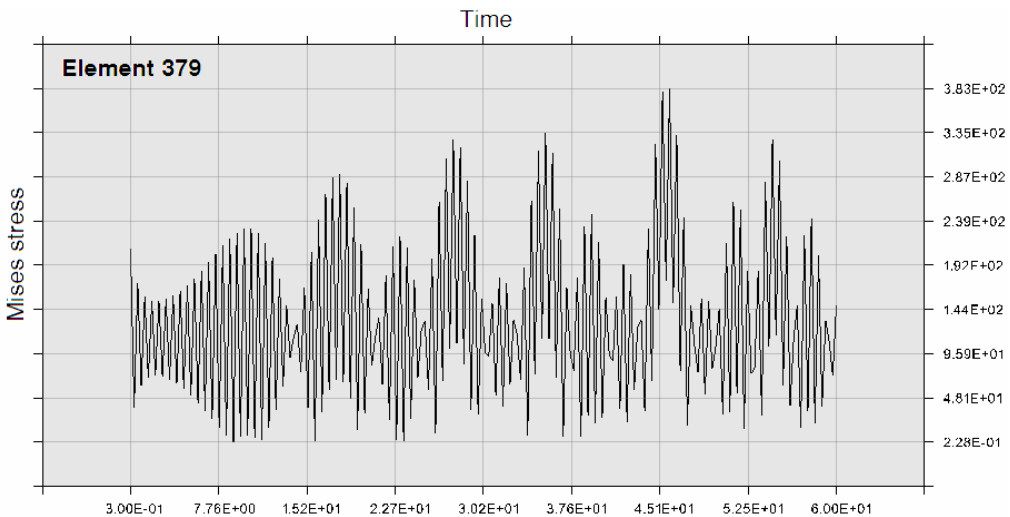
```
"create inbuilt Riemann-Siegel function";
func = RiemannSiegelZ[x];

"loop over demanded number of output steps";
For[i = 0, i < 201, i++,

  "print x-y values of function";
  If[i == 0,
   Print[0.0, ",", func /. x → 0.01],
   Print[0.3 * i, ",", func /. x → 0.3 * i];
  ];
];
```

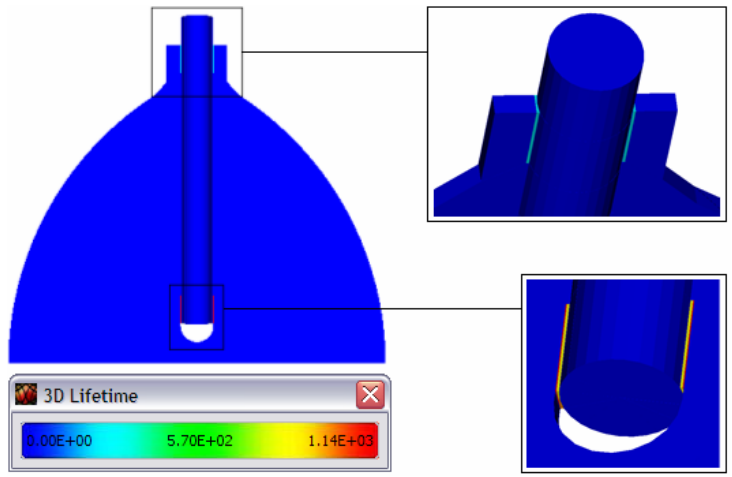


➢ LINEAR TRANSIENT AND FATIGUE ANALYSIS

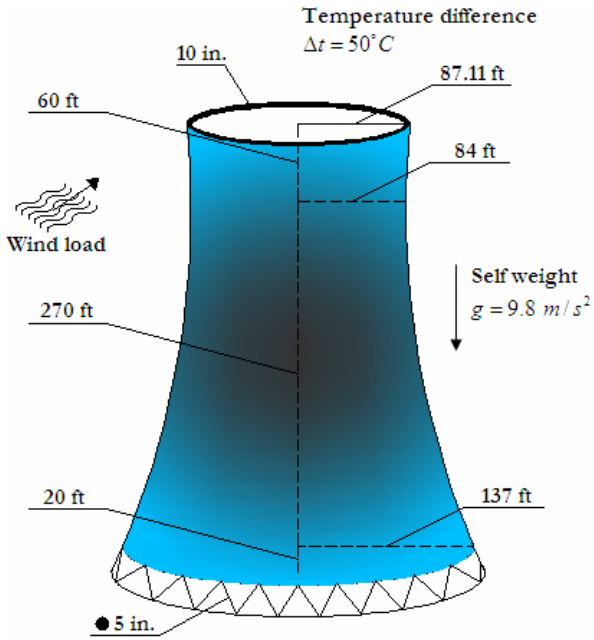


- ➢Total duration for transient analysis is 1 minute with 200 time steps.
- ➢Newmark direct time integration scheme used with damping effects neglected.
- ➢Mises stress component used for stress-range computation.
- ➢Shear Stress 100 strength curve used from Eurocode, Part 1.9 depending on the geometry of welding and loading.





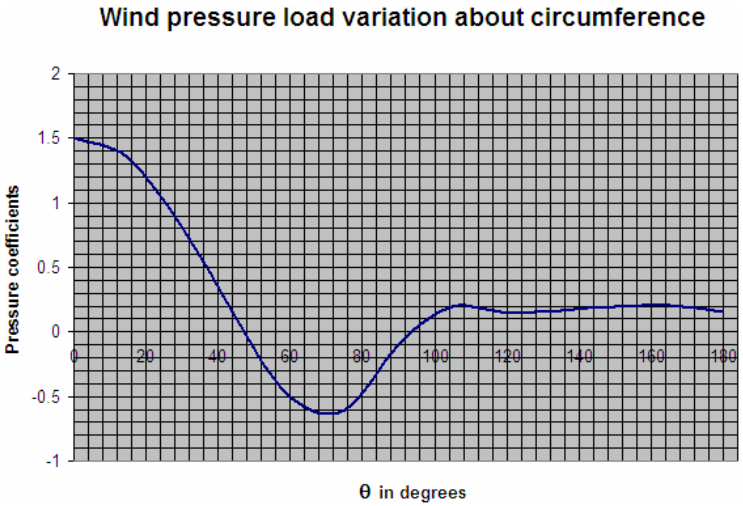
NATURAL DRAFT COOLING TOWER

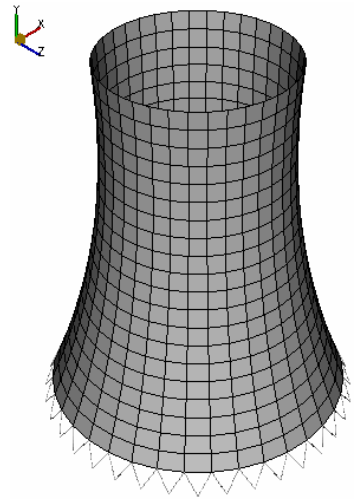


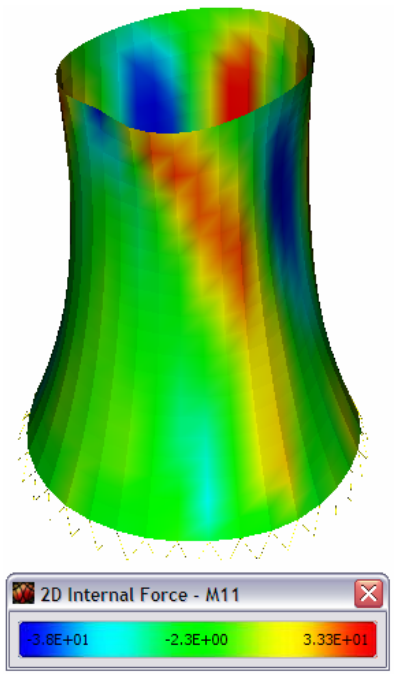
- Unsymmetrical wind load about the circumference.
- Temperature difference between the cold outside air and the hot humid air inside the tower.
- Self weight.

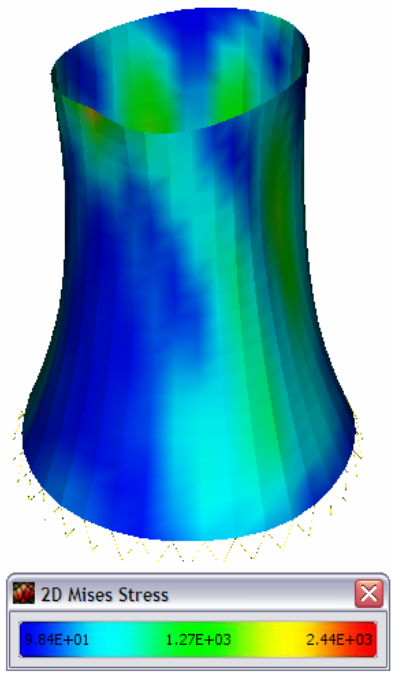
OBJECTIVE : OBTAIN DISPLACEMENTS AND STRESSES CAUSED BY THE DESIGN LOADS OF THE TOWER

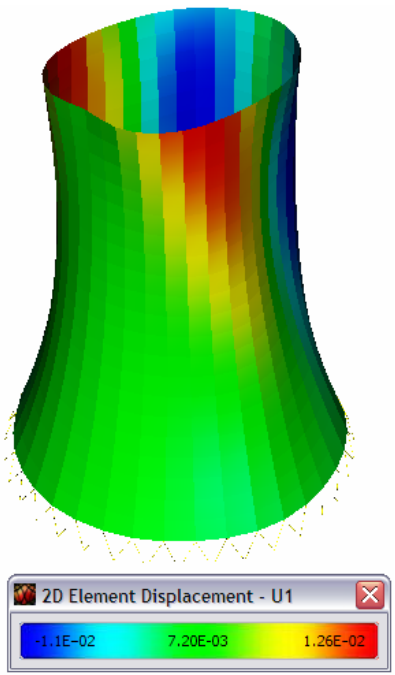
➢ WIND LOADING, MESH AND LINEAR STATIC ANALYSIS



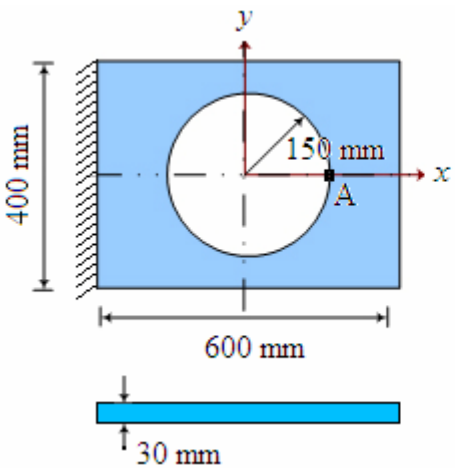


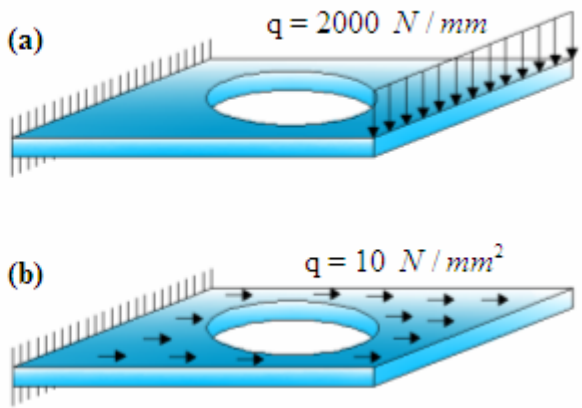


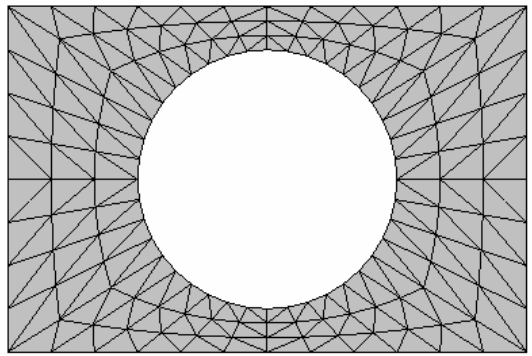




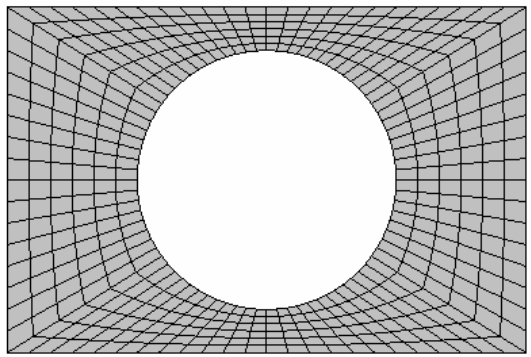
FATIGUE ANALYSIS OF A ROOF WITH HOLE







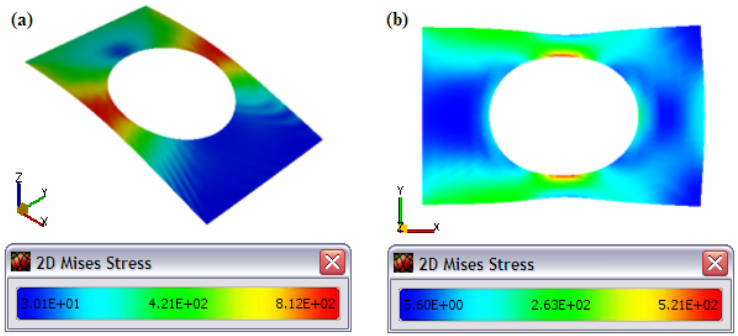
Mesh - a, Triangular plane stress elements



Mesh - b, Quadrilateral thick shell elements

OBJECTIVE : LIFETIME ESTIMATION OF PERIODICALLY LOADED ROOF WITH A HOLE FOR TWO DIFFERENT LOAD CASES

➢ LINEAR TRANSIENT, BUCKLING AND FATIGUE ANALYSES



*Values are millimeters.		u1	u2	u3	r1	r2	r3
Load case 1	Current study	0.000	0.000	-14.858	0.000	0.069	0.000
	SAP2000	0.000	0.000	-14.402	0.000	0.067	0.000
	MSC.Marc	0.000	0.000	-14.850	0.000	0.069	0.000
Load case 2	Current study (shell elements)	0.740	0.000	0.000	0.000	0.000	0.000
	Current study (plane elements)	0.751	0.000	-	-	-	-
	SAP2000	0.564	0.000	0.000	0.000	0.000	0.000
	MSC.Marc	0.725	0.000	0.000	0.000	0.000	0.000