MARTEC™

TRIDENT FEA 2004
Technical Specification

Trident FEA is an industry leading advanced finite element analysis software package for naval architecture and marine engineering applications.

Visualization

- Produces wire mesh, fill hide and hidden line removal plots from any view and/or perspective.
- Provides colour display based on specified verification parameters.
- Produces plots generated of the entire model or portion. Model can be viewed by erase/restore based on module, element type, material, property, number, value, attached elements, cursor identify, coordinate location, external/internal, structure/fluid, global/local
- Provides capability for interactive data query of input and result data
- Produces tables of detailed model input and result data
- Creates colour contour plots of pressures, element attributes, displacements and stresses
- Creates vector plots to verify loads and normals
- Generates XY plots of input and analysis result data
- Creates animation files in AVI format for display of deformations and mode shapes
- Provides superposition capability to plot stresses on deformed model and to impose deformed model over original

Primitive Geometry

- Create points, lines, arcs, surfaces, solids and intersections
- Import CAD geometry in IGES format (DXF and Step format under development)
- Various entity generation techniques
- Module definition

Mesh generation and refinement

- Mesh generation using different meshing algorithms including parametric and unstructured mesh generation
- Direct generation of individual element
- Special templates for meshing 3D beams, stiffened and corrugated panels, plate intersections and penetrations, and tubular joints
- Global and local controls with default sizing
- Mesh density control feature
- Define element size or spacing with bias
- Element conversion (Higher-lower order element and 2D-3D) capability
- Capability to create fluid elements by converting structural elements
- Top-down analysis capability
- Mesh refinement (element and special templates)
- Capability to translate, rotate and mirror image model (Append or retrain base model)
- Capability to create models of ship structures using the Ship Modeler resource to build modules from structural components (panels, strakes, girders), import CAD line entities, and automatically mesh and assemble modules

Element Types

- Rod, spring and beam elements
- Plate and shell: quad and triangle elements
- Membrane: quad, triangle and stiffened elements
- Solid: tetra, wedge and brick elements
- Axisymmetric: solid, shell and beam elements
- Transition elements
- Shear web elements
- Fracture elements
- Fluid elements

Property Library

- Plate
- Cross Section
- Material:
  - Isotropic, orthotropic
  - Elastic-plastic with isotropic and kinematic hardening and multilinear stress-strain curve
- Fracture
- Thermal
- Formulations:
  - Hyper-elastic
  - Frequency-dependent
  - Complex Stiffness

Loads, Boundary Conditions and Lumped Masses

- Concentrated and pressure loads
- Hydrostatic pressure loads
- Sectional forces
- Self weight
- Inertia loads
- Load combinations
- Prescribed displacements, velocities, and accelerations
- Time or frequency dependent loads
- Out-of-phase time dependent loads
- Support Motions
- Follower forces
- Multi-point constraints
- Fixed and flexible translation and rotational constraints
- Temperature

Application Interfaces

- External FE Model data translation (import/export) capability: (Ansys, Dyna3D, Nastran, Patran, Marc, Abaqus, Sesam, Adina)
- Hypermesh: Model data translation (import/export)
- CAD: Geometric data translation using CAD data standards for IGES (DXF and Step format under development)
- Heat transfer analysis: Solves steady state temperature and heat flux conditions due to convection, conduction and radiation. Uses Trident Heat Transfer Solver VASTF (Field solver)
- Seakeeping: Hydrodynamics loads via integrated interface. Uses FD and TD WaveLoad solver. (Time Domain and Frequency Domain)
- Fatigue I: Basic fatigue analysis capability using Miners Sums and Fatigue Diagrams methods
- Fatigue II: Simulation of crack initiation and propagation under deterministic (constant amplitude) and spectral loads. Uses Trident

LIFE3D program. (Linear Incremental fatigue evaluation system for 3 dimensional models)

Graphical Model Verification

- Colour display
  - Integrity tests: Aspect ratio test, Minimum and maximum corner angles verification, Common boundaries, Element warping, Node and element duplication tests, Minimum element side length test, Plate element thickness to side length ratio test
  - Element attributes: formulations, properties
- Vector Display
  - Element normals
  - Boundary conditions: Free, Fixed, Flexible, Prescribed
  - Mass: element and lumped masses
  - Load cases: Element and concentrated loads, translational accelerations, angular velocities, angular accelerations and centre-of-rotation
- XY plots
  - Time function
  - Frequency function
- Tabular Reports
  - Summary: Dimensions, Nodes and elements, Loads, Boundary conditions, Masses, Skew Coordinates, Formulation, Function, Primitives, Properties, Crack Lines and Tips, and Fluid mass
  - Loads global summary

Results Display

- Color-filled and color line contour, vector, and X-Y plots of Stress, Strain, Displacements, Vibration and Buckling modes
- Animation
- Superposition

Solution methods

- Direct:
  - Sparse matrix and skyline-based
- Eigensolvers:
  - Direct and subspace iteration

Analysis modules

- *Linear static analysis*
  - Produces displacements, strains, stresses, forces, and error estimates as results of the analysis under a variety of loading conditions
- *Natural frequency analysis*
  - Provides several eigenvalue analysis techniques to evaluate natural frequencies and their corresponding mode shapes. Both in-air and fluid modes can be calculated (latter requires a fluid mass representation)
- *Buckling analysis*
  - Predict the loads at which the onset of structural instability will occur through eigenvalue analysis
- *Dynamic analysis*
  - Performs dynamic response analysis due to time history loading, response spectra input, steady state harmonic input, and random vibration excitations

- *Complex eigenvalue*
  - Complex natural frequencies are calculated for system with complex stiffness characteristics
- *Nonlinear analysis*
  - Performs nonlinear static and dynamic analysis, including large displacements, plasticity, hyper elasticity, and post buckling analyses
- *Thermal stress analysis*
  - Predicts thermal-induced linear elastic stress and deformations

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia
Canada B3J 3J8

tel. 902.425.5101
fax. 902.421.1923
HInfo@martec.comH
Hwww.martec.comH