NEi Nastran

Advanced Finite Element Analysis

25 Key Features

2. **Composite Analysis**

3. **NEi Editor**
   Design Optimization
   Fatigue Analysis

4. **Surface Contact**
   Edge Contact

6. **Advanced Nonlinear Analysis**

25 Key Features of NEi Nastran

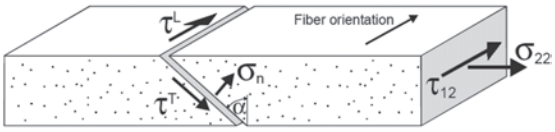
NEi Nastran is a powerful, general purpose Finite Element Analysis (FEA) tool with an integrated graphical user interface and model Editor which is used to analyze linear and nonlinear stress, dynamics, and heat transfer characteristics of structures and mechanical components. It represents the latest in FEA technology with one of the fastest iterative solvers on the market along with accurate solutions that have been trusted for nearly 20 years by companies in all industries. It is available on a wide variety of platforms including 32-bit and 64-bit Windows and Linux operating systems.

1. Micromechanics Based Composite Laminate Failure Analysis

- Higher level accuracy in correlating to real world composite failures, as compared to smeared lamina approaches (Tsai-Wu, Hill, Hoffman, etc.)
- Exact failure mode can be obtained through accessing separate failure index calculations for fiber and matrix
- Proven technology through test-analysis correlations of aerospace applications
- Based upon the well documented MultiContinuum Theory (MCT) which requires only standard orthotropic material information
- Developed for both unidirectional and woven composite materials
- Having access to constituent stress/strain leads to a direct link to composite fatigue life prediction, by coupling MCT with kinetic theory of fracture

2. Advanced Composites Ply Failure Criteria

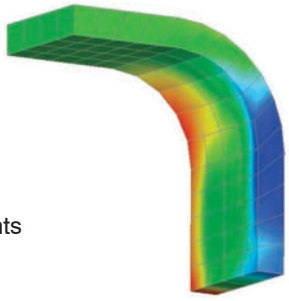
- LaRC02 set of criteria
- Developed by NASA
- First Ply Failure
- Physically based
- Hashin, Puck approach
- No additional material properties required compared to Tsai-Wu
- Failure mode taken into account
- Extensive validation (WWFE)



Advanced Failure Criteria for Composites

3. Nonlinear Progressive Ply Failure Analysis (PPFA™)

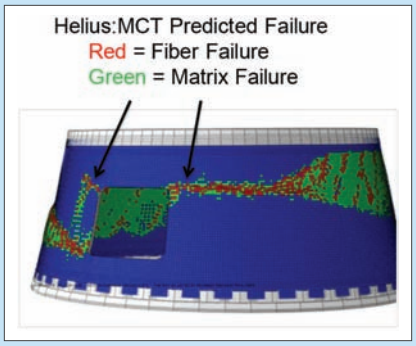
- Proper representation of reduced stiffness of failing composite parts
- Simulation of incomplete failure
- More accurate damage assessment
- Enables substantially more optimized composite designs when designing to ultimate load, by considering Last Ply Failure as well as First Ply Failure



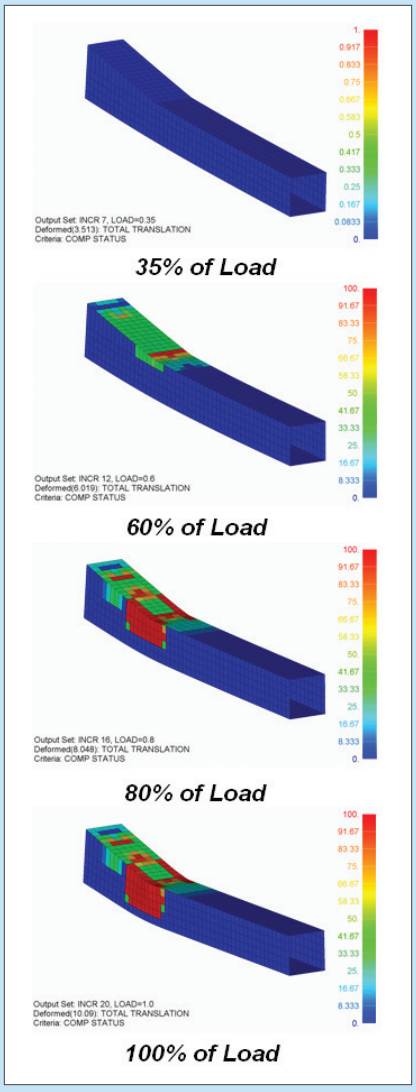
Solid Composites

4. Advanced 3D Layered Composite Analysis

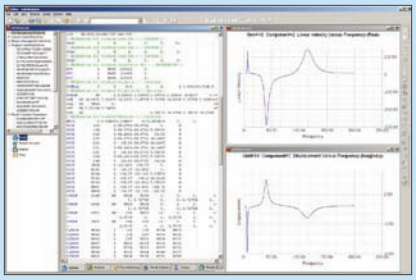
- Composite layups can be directly applied to solid elements
- CHEXA and CPENTA support layered material definitions using the PCOMP Bulk Data entry
- Overcomes the limitations of Classical Laminate Theory, which include the failure to take into account local design details such as jointed regions and T-connections
- Transverse shear and the through thickness normal stresses are included as a direct result of the triaxial stress state in the fully 3D orthotropic material
- Mapping of 3D ply layup from existing 2D PCOMP data
- Conversion of MAT8 (2D orthotropic) to new MAT12 (3D orthotropic)



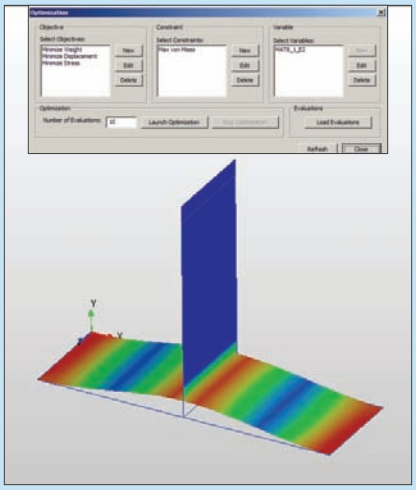
Composite Laminate Failure Prediction



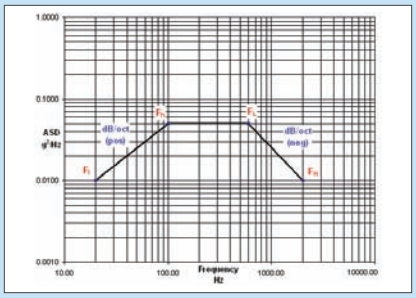
Progressive Ply Failure Analysis



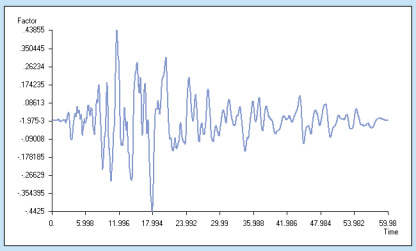
NEi Editor Model Results



Design Optimization in the NEi Editor



Vibration Fatigue



Comprehensive Fatigue

5. NEi Nastran Editor GUI

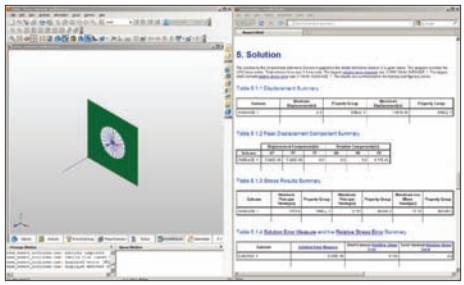
- Fully integrated and customizable Nastran Editor controls program operation and provides results summary data via an easy-to-use GUI
- Bulk data files can be modified in an interactive environment
- Special real-time controls allow solution parameters to be changed while running
- Post-processing and results query
- Tabbed windows to give immediate access to all input and output files
- Field markers simplify manual editing and increase productivity
- Complete online documentation and context-sensitive help that actually helps
- Permits batch queuing of jobs for sensitivity and configuration trade studies
- Real time 2D x-y plotting and 3D deformed shape and contour plotting with optional MS Excel Comma Separated Variable (.CSV) file output

6. Shape and Parameter Design Optimization

- Parameter optimization integrated into the NEi Editor
- Optimization GUI designed to manage and define optimization constraints, variables and objectives
- Simple right-click definition of optimization variables from within the NEi Editor

7. Automated Analysis Report Generation

- Analysis summary
- Group definitions
- Contact definitions
- Element initial distortion
- Load vector resultant
- Reaction vector resultant
- Displacement data
- Peak displacement
- Stress result summary
- Report wizard for easy generation



Automated Analysis Report Generation

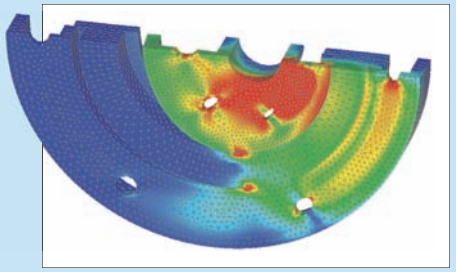
8. Multiaxial and Vibration Fatigue

- No third party products or costly modules for fatigue and vibration fatigue
- Fatigue life and fatigue damage due to random vibration calculated based on Steinberg's three banded technique
- Allows for quick assessment and predictability of outcome of standard industry vibrational testing
- Greatly improves upon predicting structural failure due to vibration based on RMS (root mean square) outputs and response PSDs
- Fatigue capability typically found in costly specialized fatigue packages in core NEi Nastran
- Stress life (high cycle) and strain life (low cycle) methods included
- Fatigue life and fatigue damage output from uniaxial and complex multiaxial loadings
- Rainflow counting for variable amplitude loadings

25 Key Features of NEi Nastran

9. Linear Surface Contact

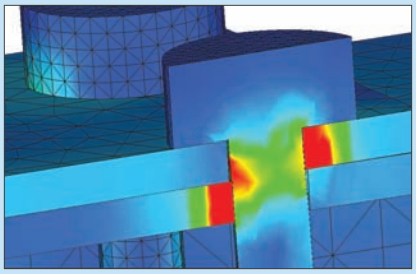
- True 3D surface contact with closing and opening of gaps in core Nastran linear statics (SOL 101)
- Iterative strategy saves hours of nonlinear run time
- Works with Automatic Surface Contact Generation (ASCG)
- Makes linear gap elements obsolete
- Ideal for large contact models that are initially in contact
- Fast trade studies for bearing assemblies analyzed in minutes



Linear Surface Contact

10. Nonlinear Surface Contact

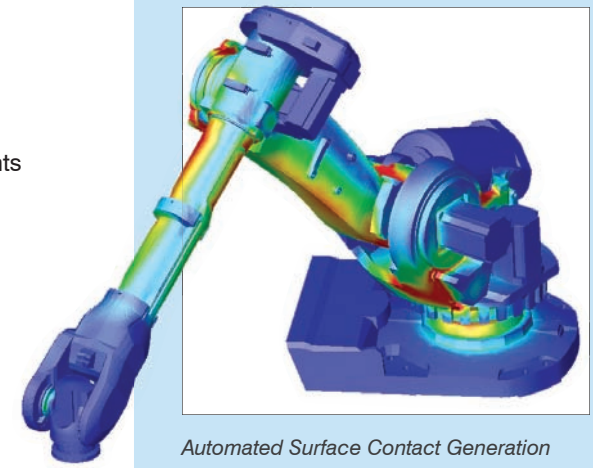
- Free edges contacting surfaces now identified automatically
- Edge to face welding carried out automatically
- No element congruence required
- Offsets and gaps between edges and faces are permissible
- Rapid meshing of complex shell structures now possible
- Ideal for large thin shell fabrications such as ship, aircraft, and automotive structures



Nonlinear Surface Contact

11. Automated Surface Contact Generation (ASCG™)

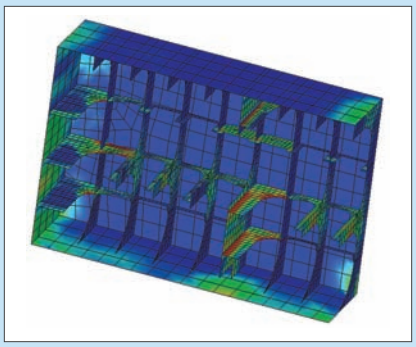
- Solver will search for element faces in contact and automatically set up the contact between discontinuous bodies
- Can be used for automatically welding parts together or automatically defining nonlinear surface contact in complex regions
- No geometry is required
- Legacy Nastran models supported – ability to upgrade from legacy GAPs to surface contact with a single Case Control Command
- Ability to weld parts together through a small gap automatically, which prevents manual definition of multiple master-slave regions with an offset
- Self contact is now available using ASCG



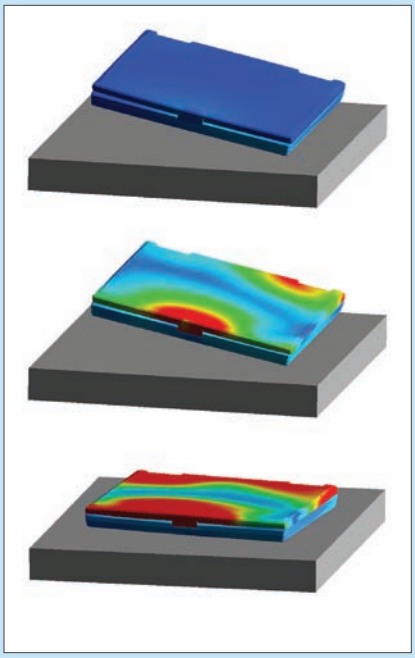
Automated Surface Contact Generation

12. Automated Edge Contact Generation (AECG™)

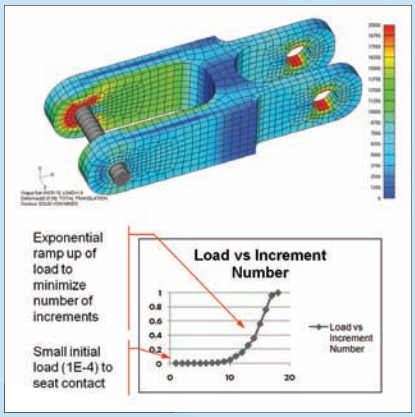
- Free edges contacting surfaces now identified automatically
- Edge to face welding carried out automatically, offsets and gaps between edges and faces are permissible
- No element congruence required allowing for dissimilar meshes
- Offset weld contact – new contact type, allows for welded connections with significant separation between contact edges/surfaces
- User Input
  - Choose type of contact to be generated (general, welded, bidirectional sliding, rough, or offset weld)
  - Designate which elements should be considered for each contact pair generated
  - Specify near tolerance for objects to be considered in contact with each other
  - Advanced contact options for friction, contact stiffness, contact activation distance, allowable penetration, and surface offset available
- Ideal for complex shell structures
  - Rapid meshing now possible, large amount of labor intensive meshing removed
  - Large thin shell fabrications such as ship structures
  - Mid-surfaced solid models ideal candidates, not essential to close gaps



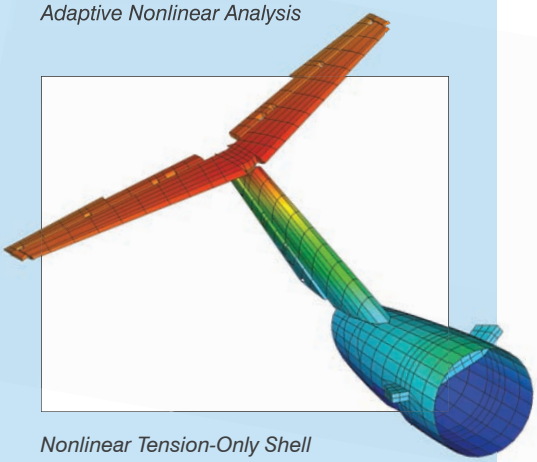
Automated Edge Contact Generation



Laptop Drop Testing



Adaptive Nonlinear Analysis



Nonlinear Tension-Only Shell

13. Automated Impact Analysis (AIA™)

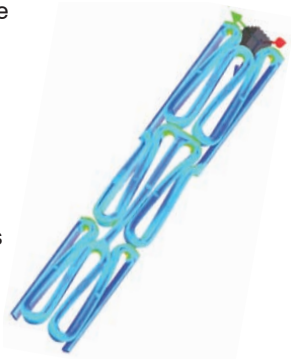
- Routine drop testing so there is no manual configuration with every step
- Arbitrary object and target shapes
- Automatic object repositioning
- Uses Automatic Surface Contact Generation (ASCG) to automatically define contact regions
- Automatic natural frequency calculations of object and target allows optimum calculations of time steps and impact duration
- Nonlinear Transient Analysis – all data calculations and data transfer carried out
- Automatic real-time output of x-y data at impact points during analysis

14. Adaptive Nonlinear Static and Nonlinear Transient Analysis

- Adaptive load increment/time step method for efficient nonlinear solutions
- Number of increments is minimized, based on solution convergence
- Only maximum and minimum number of load increments need to be set
- Nonlinear solutions can be started at a very small load for better stability, without having to run a large number of load increments

15. Advanced Nonlinear Material Models

- Advanced nonlinear materials for higher fidelity simulation models
- For applications such as biomedical stents and deployable space structures, Nitinol material model can be employed to obtain shape memory effects and superelasticity
- For civil applications, concrete material model can be employed to simulate the cracking behavior in tension and crushing behavior in compression
- Improved standard nonlinear elastic and plastic material model to approximate certain specialized nonlinear effects
- Mooney-Rivelin, Neo-Hookean, Ogden, Yeoh, and generalized polynomial material models for compressible and incompressible material behavior
- Temperature dependent material properties for thermoelastic and thermoplastic nonlinear material behavior



Shape Memory Materials (Nitinol)

16. Special Nonlinear Elements

Nonlinear Tension-only Shell Element

- Tension-only membrane reverts to a shear panel in compression
- Designed for analyzing semi-monocoque aircraft structures
- Controllable compression stiffness scale factor
- Currently used by Cessna Aircraft for production design work

Nonlinear Tension-only Cable Elements

- Allows initial slack or preload
- Ultimate tensile stress can be specified above which the cable fails
- Supports temperature dependent material properties

25 Key Features of NEi Nastran

17. Nonlinear Database Management and Restarting

- A unique nonlinear database system has been created to allow very efficient storage and retrieval of data during a nonlinear analysis
- Restart capability for nonlinear static analysis is provided via PARAM, NLINDATABASE and the NLINDATFILE directive
- Multiple restart database files may be generated at user defined load increments then used as restart points with different loading, boundary conditions, model parameters, or material properties
- One application is nonlinear buckling (SOL 180) where the user can predict the load region where instability is expected and confirm it with a nonlinear buckling analysis
- A nonlinear static database can also be used as an initial condition for a nonlinear transient response solution (SOL 129)

18. Modal Database Management and Modal Filtering

- Modal Base filtering option based on modal effective mass
- Modal Data save and re-use option
- Very big savings in CPU time possible with all Modal based response method avoiding re-calculation of modal data
- Simple interface: best usability

19. Integrated Modal Assurance Criteria (MAC) and Modal Cross-Orthogonality (MXO)

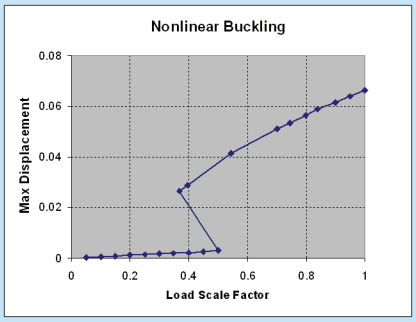
- Correlation carried out by NEi Nastran in a normal modes analysis
- No third party software required
- External or test data can be in CSV, DMIG or MDB format
- Non-MDB (Modal Database) data will be automatically converted to MDB for re-use
- Automatic mapping of test to model grid positions
- Interpolation of test eigenvalue shape to model
- MAC and MXO correlation supported
- Text output in the Model Results Output File
- Graphical support is included in the NEi Nastran Editor
- Future developments include FRF correlation

20. Virtual Fluid Mass Boundary Element

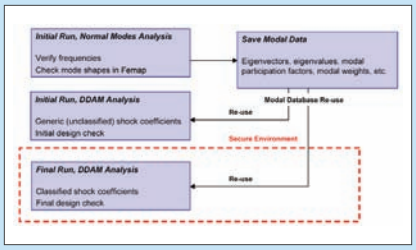
- Boundary Element Approach (BEA) to include Fluid Effects
- Air effects on aircraft structures can be determined
- Wet modes for submerged maritime structures can be calculated for more accurate dynamic representation
- Fluid mass is coupled to mode shapes and the added mass varies from mode to mode

21. Imported Results Data Interpolation

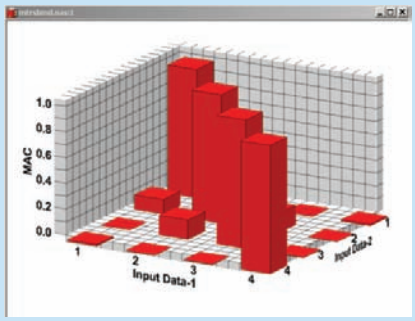
- 2D or 3D interpolation of input, temperature, displacements, forces, moments, and pressure
- Allows load transfer across different models / disciplines
- Accurate and user-controllable 2D / 3D polynomial interpolation through least-squares approach



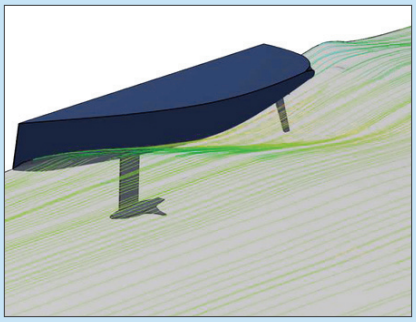
Nonlinear Static Restart Capability



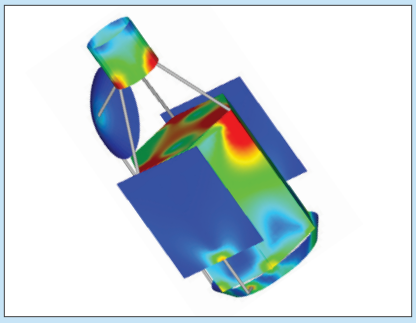
Application Workflow Example: DDAM



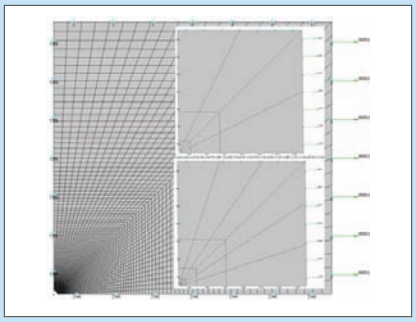
Integrated Modal Assurance Criteria



Virtual Fluid Mass Boundary Element

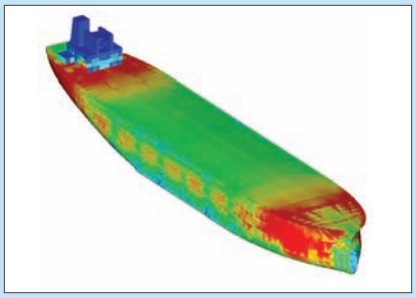


Automated Inertial Relief (AIR)



Robust and Accurate Element Library

SUV Modal Analysis: 1,230,000 DOF. Total Solution time: 6.8 min. (20 modes) and 17 min. (100 modes). Run on an Intel Core i7 2.8 GHz CPU with 8GB of RAM.



Dynamic Design and Analysis Method (DDAM)

22. Automated Inertial Relief (AIR™)

- Easy method to analyze unconstrained or free-floating structures such as ship or aerospace vehicles
- All that is required is loads and PARAM, INREL, AUTO – no need for constraints or SUPORT entries or PARAM, GRDPNT
- Center of gravity is automatically calculated and internal stabilization elements automatically created

23. Robust and Accurate Element Library

- Higher order internal shape functions for quad and hex elements with geometric and material nonlinear capability
- Solid elements can be used to model thin plates and adhesive bond lines
- Element mass matrix formulations are more accurate than other leading solvers
- Robust results even with distorted elements
- Faster mesh convergence
- Extensive verification and comparison problem set that demonstrate superior accuracy
- Supported in all solution modules including nonlinear

24. High Performance, Large Capacity Linear and Eigensolvers

- PCGLSS Solver – Extremely fast iterative solver capable of handling models over 25 million degrees of freedom on a 64-bit Windows PC
- PSS Solver – Extremely fast parallel direct solver capable of handling models over 15 million degrees of freedom in 64-bit Windows PC
- VSS Solver – Based on NASA Vector Sparse Solver technology
- VIS Solver – Sparse iterative (Preconditioned Conjugate Gradient) solver
- Lanczos Eigensolver – Block Lanczos eigensolver capable of handling models over 4 million degrees of freedom on a Pentium PC
- Subspace Eigensolver – Subspace eigensolver based on VSS

25. Specialized Industry Features

- Dynamic Design and Analysis Method (DDAM)
- Component Mode Synthesis (CMS)
- Direct Matrix Input Grid (DMIG) Support

In addition to these features, NEi Software leads the CAE community in technical support and demonstrates this commitment by providing unique software features such as context sensitive help and innovative distance e-learning programs. Our in-house technical support team conducts training and provides mentoring and consulting services for FEA projects.

- Online support includes unlimited access to the NEi Software Knowledge Base Center
- Email support is a 24 hour turnaround service, Monday–Friday
- Telephone support is available 7AM–5PM PST, Monday–Friday
- Mentoring support provides advice to customers on finite analysis techniques, analysis strategy, meshing, model debugging, and more
- Free maintenance updates include documentation corrections and bug fixes
- Version updates provide significant enhancements and additional functionality
- Consulting services can be provided at an additional charge, and include complete project analysis from start to finish

About NEi Software

NEi Software is a world leader in Finite Element Analysis (FEA), engineering simulation, and virtual test software. The core product NEi Nastran is a powerful, industry-proven FEA solver that thousands of companies routinely use to perform linear and nonlinear structural stress, dynamics, and heat transfer analysis. In addition, NEi Software's portfolio includes products for impact, kinematics, fatigue, acoustics, optimization, aeroelasticity, and Computational Fluid Dynamics (CFD) with support for a full range of materials from composites to hyperelastic rubber. NEi Software covers the different needs of each stage of the product development process, from designers looking for affordable, easy-to-use, CAD-based simulation for validation and trade-off studies to dedicated FE analysts looking for high accuracy, productivity, and real world fidelity. The website features case studies in aerospace, automotive, maritime, military, civil, petroleum, medical, and consumer products with videos, webinars, tutorials, and options for evaluation.

Global Headquarters

5555 Garden Grove Blvd. Ste 300
Westminster, CA 92683-1886
United States

Phone: +1 (714) 899-1220
Fax: +1 (714) 899-1369
E-mail: info@neisoftware.com
Website: www.NEiSoftware.com

NEi Software EMEA Office

The Old Barrel Store
Draymans Lane, Marlow
Buckinghamshire, SL7 2FF
United Kingdom

Phone: +44 (0)1628-400645
Fax: +44 (0)1628-891701
E-mail: emea@neisoftware.com
Website: www.NEiSoftware.com/emea

NEi Software Asia Office

Shinjuku Park Tower
N30th Floor 3-7-1 Nishi-Shinjuku
Shinjuku-ku, Tokyo, 163-1030
Japan

Phone: +81-(0)3-5326-3062
Fax: +81-(0)3-5326-3001
Email: asia@neisoftware.com

BROCKEYF20100603