

NEi Nastran

Module NE-L1 (Linear Static, Steady-State Heat Transfer)

Overview

The NE-L1 package brings the extremely powerful linear capabilities of Nastran to your desktop. It contains the full array of Nastran element types and material models and serves as the foundation for the other available Nastran modules.

Solution Types:

- Linear Static Analysis
- Linear Steady State Heat Transfer Analysis
- Thermal Stress and Deflection Analysis
- Composite Laminate Analysis
- Mass Properties Analysis
- Inertial Relief
- Automated Inertial Relief (AIR)
- Linear Surface Contact Analysis

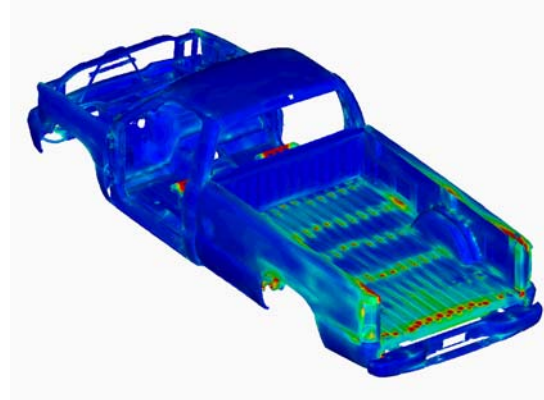
Capabilities:

Element Features:

- Material properties include: temperature and stress dependent isotropic, orthotropic, anisotropic, laminated composite, and concrete material models
- Robust shell and solid elements give accurate results even with high skew angles, large aspect ratios, and when used in a coarse mesh
- Linear and parabolic shell and solid elements
- Both element and grid point stresses can be output in any coordinate system with one simple command
- Automatic mid-side edge node option for solid elements
- Automatic vertex drill degree of freedom for shell elements
- Automatic correction for improperly defined rigid and interpolation elements
- Automatic correction of improperly defined parabolic solid elements
- Extensive checkout procedures

Element Library:

- Rod



- Tube
- Bar
- Pipe
- Tapered beam
- Spot weld (CWELD)
- Quadrilateral and triangular membranes, plates, and shells (five and six DOF/node up to eight nodes)
- Solids (four, five, and six-sided up to 20 nodes)
- Shear panels
- Plane stress, plane strain
- Axisymmetric (CTRIAX6)
- Spring, mass, and damper
- Coupled spring and damper
- Contact elements (gap, slide line, surface-to-surface)
- Surface-to-surface and edge-to-surface weld option for contact surfaces
- Offset weld for contact surfaces with significant separation
- Rigid general form, rigid rod, rigid bar, and rigid plate
- Interpolation
- Conduction
- Capacitance
- Boundary surface
- Laminated solid elements (CHEXA, CPENTA)
- General elements

Solution Features:

- Model size is only limited by available disk space
- Extremely fast sparse direct and iterative (PCGLSS) equation solvers reduces solution times from hours to seconds
- Automatic stiffness singularity detection
- Decomposition mechanism detection and correction
- Solution error measure
- Multiaxial Fatigue Analysis*
 - Life and damage results measures
 - Strain-life and stress-life

Loads and Boundary Conditions:

- Nodal forces and moments
- Pressure loads
- Bolt preload
- Gravity and centrifugal
- Rotational acceleration and velocity
- Single and multipoint constraints
- Symmetric, antisymmetric, axisymmetric, cyclic symmetric boundary conditions
- Multiple loading and boundary condition subcases
- Thermal loading and stress recovery
- Isotropic and anisotropic thermal conductivity
- Temperature dependant internal heat generation
- Temperature dependent heat transfer coefficient
- Radiation and convection loads
- Surface normal heat flux
- Grid point nodal power
- 2D or 3D interpolation of input, temperature, displacements, forces, moments, and pressure loads

Material Properties:

- Isotropic
- 2D and 3D Orthotropic
- 2D and 3D Anisotropic
- Temperature-dependent
- 2D and 3D composite laminate
- Isotropic and orthotropic plane strain

Performance and Control Features:

- Uses the most current advances in finite element technology
- Extensive error checking at all stages of execution



Direct Matrix Input Grid (DMIG) Support:

- Stiffness matrix import and export
- Conductivity matrix import and export
- Mass matrix import and export
- Load vector import and export

Model Reduction:

- Static condensation
- Export reduced stiffness matrix using DMIG format
- Automated model reduction tools and correction

Global Matrix Output:

- Stiffness matrix
- Conductivity matrix
- Mass matrix

Output Features:

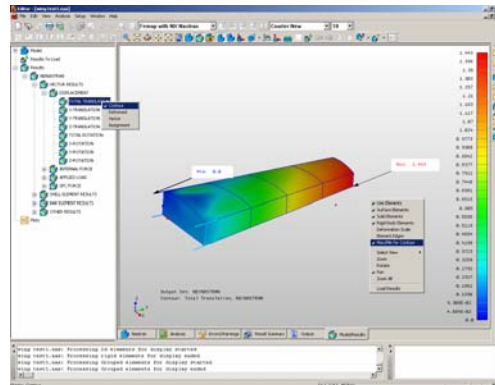
- Direct support for Femap results import using either Binary or ASCII Femap neutral file interface
- Results import support for other FEA and CAD applications using the Nastran Output 2, Nastran XDB or Patran 2.5 neutral file interface
- Detailed tabular results output file

* Additional license required

- Results measures include: energy, stresses, strains, forces, reactions, displacements, velocities, accelerations, heat fluxes, thermal gradients, temperatures, enthalpies, grid point force balance, global stiffness, mass, and damping matrixes, solution and mesh error estimates, element and grid point results
- Automatic and user definable measure sorting for handling multiple load case results
- Powerful grid and element set generator for generating sets that can be used to control output, define measure coordinate systems, generate grid point temperatures, and define measure sort commands
- Shell and solid element corner stress and strain output
- Grid point weight generator with complete mass properties
- Stress discontinuity / convergence error calculation
- Element and grid point stress, strain, heat flux, and thermal gradients can be output in any coordinate system including: material, grid point, basic, and global
- Intermediate bar and beam element output
- von Mises stress output for bar, beam, shell, and solid elements
- Composite failure index and strength ratio output
- Composite sandwich material stability index
- Automatic generation of structural temperatures for direct modeler import or structural analysis
- Heat flow into heat boundary elements
- Normalized mesh convergence error measures
- Real-time grid point and element results x-y plotting and MS Excel Comma Separated Variable (.CSV) file generation
- Inertial relief rigid body acceleration vector output
- Thermal, mechanical and total strain output
- Contact force and stress output for linear contact analysis
- Global ply through PCOMPG

Editor:

- Fully integrated and customizable Nastran Editor controls program operation and provides results summary data through an easy to use GUI
- Full post-processing and results query
- Tabbed windows to give immediate access to all input and output files
- Field markers make manual editing simple and increase productivity dramatically
- Complete online documentation and context sensitive help
- Permits batch queuing of jobs for sensitivity and configuration trade studies
- Special real time controls allow changing solution parameters while running
- Real time 2D x-y plotting and 3D deformed shape and contour plotting with optional MS Excel Comma Separated Variable (.CSV) file output
- Fully automated report generator (linear static, normal modes, and buckling solutions)
- Custom x-y plotting of results
- Shear flow vector plots
- Automatically generated dynamic plots during nonlinear analysis
- Miscellaneous results display:
 - Surface contact results
 - Rod, bar, beam, shell, and solid element results
 - Grid point stress and strain
 - Polar and rectangular complex data results
 - "TOTAL" calculated results for all Tensor6 x, y and z components in Model tree



- Optimization*:
 - Mapped input file for easy right-click definition of optimization variables
 - Optimization control dialog for editing, defining, and deleting objectives, constraints and variables
 - Integrated optimization result launcher for access to optimization evaluations

Accurate and proven answers with NEi Nastran:

- Over 20 years of use by industry
- Comprehensive verification program and documentation set
- Over 3000 test problems verified for each release
- Extensive built-in diagnostics verify accuracy of each analysis

Unparalleled support:

- Leader in outstanding customer support
- Onsite and offsite training courses taught by experienced professional engineers
- Phone and email support staffed by a team of FEA specialists
- Optional consulting services available

* Additional license required

NEi Software, Inc. is aggressively focused on commitment to the customer. Detailed documentation, customized on-site training, and comprehensive technical support ensures that you will see immediate return on your investment.

For more information about our company or our products, please contact:

Headquarters:

NEi Software, Inc.

5555 Garden Grove Blvd., Suite 300
Westminster, CA 92683-1886
USA
Phone: +1.714.899.1220
Fax: +1.714.899.1369
Email: info@neisoftware.com
Website: www.NEiSoftware.com

Europe:

NEi Software EMEA Office

The Old Barrel Store
Draymans Lane, Marlow
Buckinghamshire, SL7 2FF
United Kingdom
Phone: +44.0.1628.400.645
Fax: +44.0.1628.891.701
E-mail: emea@neisoftware.com
Website: www.NEiSoftware.com/emea

Asia:

NEi Software Asia Office

Shinjuku Park Tower
N30th Floor 3-7-1 Nishi-Shinjuku
Shinjuku-ku, Tokyo, 163-1030
Japan
Phone: +81.03.5326.3062
Fax: +81.03.5326.3001
E-mail: asia@neisoftware.com



NEi Nastran for Windows
From NEi Software, Inc.