NEiNastran

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

Capabilities:

Element Features:

- Material properties include: temperature and stress dependent isotropic, orthotropic, anisotropic, and laminated composite 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:
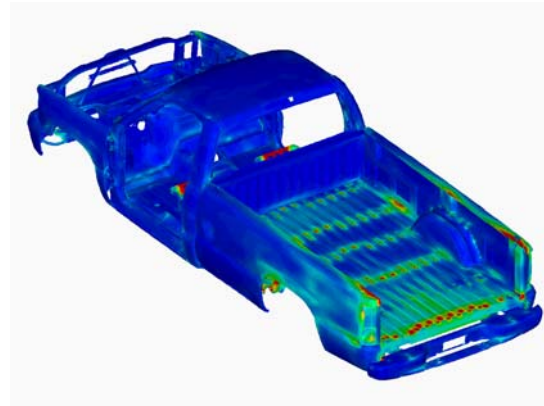
- Rod
- Tube
- Bar
- Pipe
- Tapered beam
- Cable
- Spot weld (CWELD)
- Quadrilateral and triangular membranes, plates, and shells (five and six DOF/node up to eight nodes)
- Solids (four, five, and six-sided up to 20 nodes)
- Shear panels
- Plane stress, plane strain
- Spring, mass, and damper
- Coupled spring and damper
- Contact elements (gap, slide line, surface-to-surface)
- Weld option for contact surfaces
- Rigid general form, rigid rod, rigid bar, and rigid plate
- Interpolation
- Conduction
- Capacitance
- Boundary surface
- Laminated solid elements (CHEXA, CPENTA)

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

Loads and Boundary Conditions:
- Nodal forces and moments
- Pressure loads
- 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
- Orthotropic
- Anisotropic
- Temperature-dependent

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
- 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
- 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

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 solutions)
- Custom x-y plotting of results
- Shear flow vector plots

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
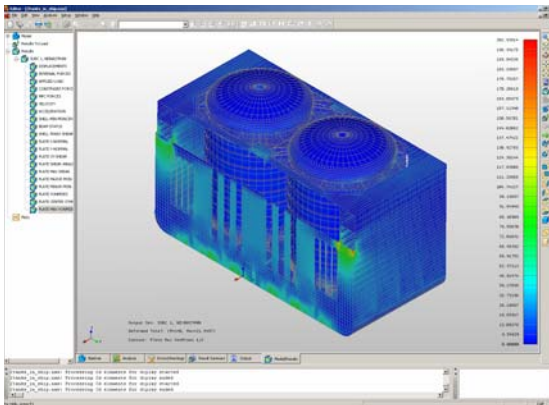
Accurate and proven answers with NEiNastran:

- Over 15 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

Noran Engineering, 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:
Noran Engineering, Inc
5555 Garden Grove Blvd., Suite 300
Westminster, CA 92683-1886
USA
Phone: 1.714.899.1220
Fax: 1.714.899.1369
Email: info@noraneng.com
Website: www.NENastran.com

Europe:
SmartCAE
Piazza della Gualchierina, 9
59100 Prato
ITALY
Phone: +39.0.574.404.642
Fax: +39.0.574.401.265
E-mail: info@smartcae.com
Website: www.smartcae.com

Asia/Pacific:
Digital Solutions
Kyoei Nakasuji Bldg., 3-7-18
Nakasuji, Asaminami-ku
Hiroshima 731-0122
JAPAN
Phone: +81.82.831.1190
Fax: +81.82.831.1193
E-mail: post@digital-sol.co.jp
Website: www.digital-sol.co.jp

NEiNastran *for Windows*
From Noran Engineering, Inc.