

NEiNastran

Features

Overview

Following is a complete list of NEiNastran's features and capabilities.

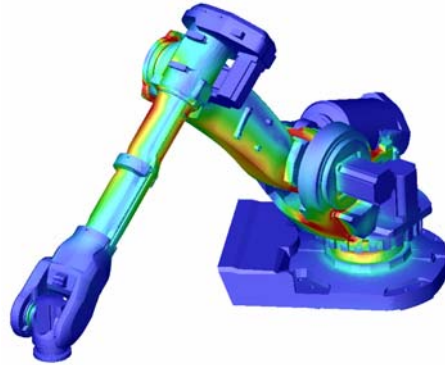
Features:

User Interface:

- Native Windows look and feel
- Multiple graphics windows
- Full, multi-level undo/redo
- On-line help with hypertext links
- Toolbars and custom tools toolbars to access frequently used commands
- Cut and paste images into Windows applications
- Dynamic highlight during selection operations
- Box, circle, polygon, front, depth and query picking of geometric and FEA entities
- Select entities by associativity (all elements connected to specified nodes, all elements of a specified property)
- Extensive material properties library with over 6000 different materials and alloys

Modeling Capabilities:

- Basic geometry creation
- Point, line, circle, spline, surface, volume
- Boolean and extrude/revolve solid modeling
- Mid-surface extraction
- Project curves onto surfaces
- Project nodes onto a plane or to a vector
- Intersect surfaces to create curves
- Define regions by projecting curves onto solids
- Create curves based on U-V space on surfaces
- Shell solids with constant thickness
- Rule, revolve, extrude and loft surfaces
- Stitch surfaces into solids
- Break, trim, extend, join and fillet, offset and copy geometric entities
- Group operation Booleans



- Parametric design language to define model in terms of variables and equations
- Import or export DXF and IGES points and curves, Stereolithography (SLA) data, ACIS (.sat) and Parasolid (.x_t) parts or assemblies
- CATIA import model files and Express files from CATEXP (CATIA v4.1.x or v4.2), VDA import (up to v2.0), IDEAS import (MS8), Pro/E import (v16 to v20), Solid Edge import, Unigraphics NX import (v11 through NX v1)
- Import model files from all popular commercial FEA codes
- Line, plate, and solid element types
- Mass and general stiffness matrices
- Contact lines and surfaces, and slide lines
- Generate plates from lines and solids from plates
- Quad-dominant automatic surface meshing, including multiple internal boundaries such as holes
- Automatic and mapped meshing, including biasing
- Tetrahedral and hexahedral mesher
- Mesh refinement and smoothing
- Geometry or finite element based loads and boundary conditions
- Unlimited number of grid points
- Composites layup dialogs
- Data surfaces to define complex load definitions

- Geometry thicken to add thickness to surfaces

Groups and Layers:

- Easily subdivide your model for visualization or post-processing purposes, group by; Coordinate Clipping
- Automatically add new entities to active or user-specified group
- Group by ID, Associativity, Material, Property, and Type
- Automatic group creation based on properties, materials, and geometric constraints

Results Processing:

- Deformations, animations, and vector displays
- Single and multi-load set animations
- Filled color contours and criteria displays
- Isosurface and cutting planes, with dynamic control
- Shear and bending moment diagrams
- Error estimates
- Results across composite laminates
- Extensive result sorting capabilities
- x-y plots with multiple curves
- Text reports: standard and user-customized
- Interactive data query with mouse
- Freebody displays, including Grid Point Force Balance support for grouped entities
- Import/export in comma separated tables
- Internet publishing with VRML support
- Save animations with AVI support

Graphics:

- Dual Windows GDI (Vector-Based) and OpenGL graphics
- 3D dynamic pan, zoom and rotation
- Hidden line and wireframe display
- Free edge and free face display
- Light source shading and transparency
- Complete beam and plate displays, including orientation, axes, offsets, etc.

Customization:

- Record, edit and playback macros
- BASIC Scripting Language
- Neutral file: fully documented binary and ASCII file formats

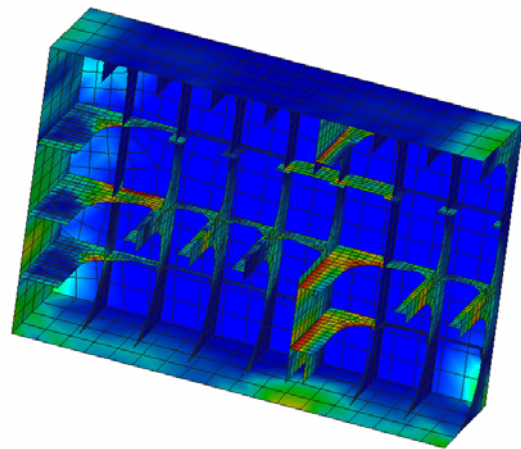
Element Library:

- Tapered beam
- Rod and bar
- Tube
- Pipe
- 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
- Cable element
 - Initial slack or tension
 - Failure (snap)
- 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
- Spot weld (CWELD)
- Rigid general form, rigid rod, rigid bar, and rigid plate
- Interpolation
- Conduction
- Capacitance
- Boundary surface
- Laminated solid elements (CHEXA, CPENTA)
- General elements

Solution Capabilities:

- Linear Static Analysis
- Inertial Relief
- Thermal Stress and Deflection Analysis
- Nonlinear Static Analysis
 - Geometric nonlinear
 - Large displacement/rotation (updated Lagrangian)
 - Follower forces
 - Snap-through analysis
 - Displacement control
 - Material nonlinear
 - Elastic
 - Elastic-plastic
 - Perfectly plastic
 - Nonlinear elastic
 - Thermo-elastic
 - Creep
- Tension-only cables and shell elements with multiple loading condition

- Gap, slide line, and surface-to-surface contact with friction
- Temperature dependent materials in all solutions
- Linear and Nonlinear Buckling Analysis
- Natural Frequencies and Mode Shapes
 - Constrained, unconstrained, and rigid body
 - Modal participation factors
 - Modal reaction forces
 - Effective mass and percent effective mass
 - Extremely fast blocked Lanczos and subspace methods
- Linear and Nonlinear Direct Transient Response Analysis with adaptive time stepping
- Linear Surface Contact Analysis
- Direct Frequency Response
- Modal Transient and Frequency Response
- Modal Response and Shock Spectrum Generation
- Dynamic Design Analysis Method, DDAM
- Single and multi-phase DDAM operation capability
- Modal Summation (NRL, SRSS, ABS, and CQC methods)
- Random Vibration
 - Correlated and uncorrelated input
 - Power spectral density
 - RMS
 - Autocorrelation input
 - Force, reaction, displacement, velocity, and acceleration output
 - Optional maximum and minimum principal stress/strain and von Mises results
 - Results output for bush elements
 - Multipoint constraint force output
- Vibration Fatigue Analysis
 - Life and damage results measures and biaxiality ration from vibration fatigue analysis
- Response Spectra Generation
- Complex Eigenvalue Analysis
- Enforced Motion
- Damping supported in all dynamic solutions
 - Modal damping
 - Equivalent viscous
 - Fraction of critical
 - Quality factor
 - Structural damping
- Material damping
- Discrete viscous damping elements: CBUSH, CVISC, CDAMP
- Raleigh and proportional damping
- Nonlinear and frequency dependent damping
- Option to force modal damping to be treated as structural damping
- Nonlinear shock and vibration mount
- Linear and Nonlinear Prestress
 - Dynamic solutions
 - Static analysis
 - Buckling analysis
 - Modal analysis
 - Transient and frequency response analysis
 - Random vibration
 - Response/shock spectrum generation
 - Modal summation
- Linear and Nonlinear Steady State Heat Transfer
- Linear and Nonlinear Transient Heat Transfer
- Composite laminate materials supported in all solutions
- Mass Properties Analysis
- Direct Matrix Import and Export (DMIG) for all solutions
- Craig-Bampton and Guyan Reduction
- Static condensation
- Automated Surface Contact Generation (ASCG)
- Automated Impact Analysis (AIA)



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
- 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
- Automated internal superelement generation
- Automated surface contact and surface weld generation

Solution Features:

- Model size limited by available disk space only
- Extremely fast sparse direct and iterative (PCGLSS) equation solvers reduces solution times from hours to seconds
- Automatic mass and stiffness singularity detection and correction
- Decomposition mechanism detection and correction
- Solution error measure for static solutions, orthogonality loss and error norms for eigenvalue solutions
- User definable restart capability for nonlinear static analysis

Material Properties:

- Isotropic
- 2D and 3D Orthotropic
- 2D and 3D Anisotropic
- Temperature-dependent
- Stress-dependent
- Creep
- Large strain hyperelastic
- Large strain isotropic
- 2D and 3D composite laminate
- Isotropic and orthotropic plane strain

Performance and Control Features:

- Uses the most current advances in finite element technology
- Written in fast, optimized 32-bit Visual Fortran 95 and C/C++
- Complete program control is provided allowing user definable program execution sequence
- Multiple model initialization files can be created, each customized for a particular job
- Extensive error checking at all stages of execution
- Program execution can be tailored to system resources for optimum performance
- Support for model input files over 10 million lines in size

Output Features:

- Direct support for FEMAP results import using either ASCII or Binary FEMAP neutral file interface
- Results import support for other FEA and CAD applications using the NASTRAN Output 2 or PATRAN 2.5 neutral file interface
- Support for NASTRAN XDB results neutral files
- A detailed tabular model data/results file eliminates the cryptic nature of standard NASTRAN output files by explaining all entities in the model database
- Results measures include
 - Energy
 - Stresses
 - Strains
 - Forces
 - Reactions
 - Displacements
 - Velocities
 - Accelerations
 - Heat fluxes
 - Thermal gradients
 - Temperatures
 - Enthalpies
 - Grid point force balance
 - Global stiffness matrix
 - Global conductivity matrixes
 - Mass and damping matrixes
 - Solution and mesh error estimates
 - Element and grid point results

- Nonlinear results include
 - Equivalent stress
 - Effective plastic, nonlinear elastic, and creep strain
 - Gap, slide line, and contact surface displacement
 - Contact forces and stresses
- 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
- Strain energy output for composite shell elements
- Von Mises strain output for linear solutions
- Automatic generation of structural temperatures for direct modeler import or structural analysis
- Element and grid point thermal gradients and heat flux
- Heat flow into heat boundary elements
- All directional results measures can be output in a user definable coordinate system either before or after an analysis is run
- PUNCH file support for real element results data
- Support for strain curvature output
- Bush element linear strain output
- Normalized mesh convergence error measures

- Real-time grid point and element results x-y plotting and MS Excel Comma Separated Variable (.CSV) file generation
- Inertia relief rigid body acceleration vector output
- Thermal, mechanical and total strain output

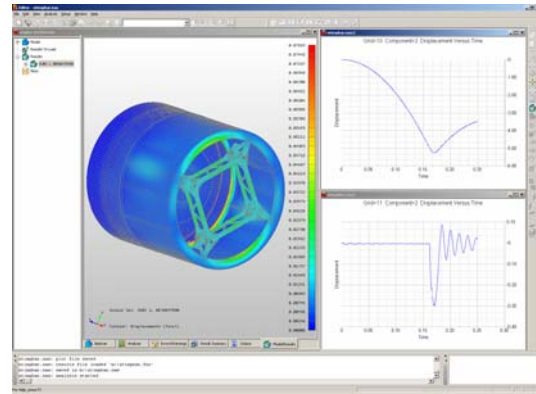
Loads and Boundary Conditions:

- 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
- Temperature dependent conductivity
- Isotropic and anisotropic thermal conductivity
- Temperature dependant internal heat generation
- Temperature dependent heat transfer coefficient
- Temperature gradient dependent heat transfer coefficient
- Radiation and convection loads
- Nonlinear functional forms
- Surface normal heat flux
- Grid point nodal power
- 2D or 3D interpolation of input, temperature, displacements, forces, moments and pressure loads
- Specified constant temperatures for steady state analysis
- Specified time-dependent temperatures for transient analysis
- Initial starting temperatures for nonlinear steady state analysis
- Initial starting temperatures for all transient analyses
- Inertial relief loading

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
- Tabled 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)
- Shear flow plots

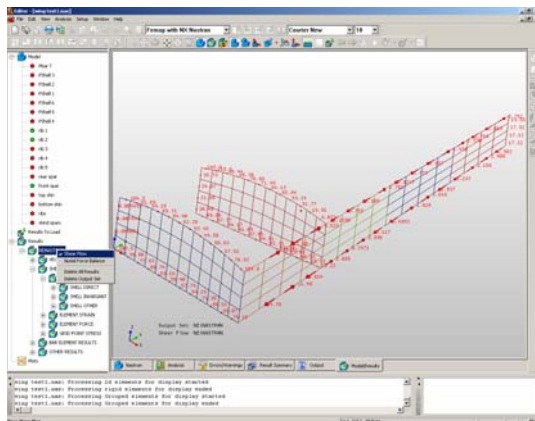


Accurate and proven answers with NEiNastran:

- 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

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



- Assignment plots
- Secure DDAM data input form
- 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
 - Shear flow and nodal force balance
 - Rigid body elements results
 - "TOTAL" calculated results, for all Tensor6 x, y and z components in Model tree

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.