SolidWorks becomes your FEA Platform

By embedding high accuracy, industry proven Nastran FEA solvers directly in SolidWorks, your familiar CAD environment and technology becomes your platform for creating simulation models, launching various analyses, and viewing the results. Integration is assured as part geometry is accessed directly through SolidWorks’ API. All the familiar tools and graphical power of Inventor are available for converting your 3D CAD model to the FEA domain. You apply loads, boundary conditions, material properties and mesh your model using the familiar layout of Inventor’s menus, toolbars, tree structure, and windows. Once your model is solved, results like stress contour plots and animations of deformation are displayed using Inventor as your post-processor.

Nastran is your FEA Engine

Using NEi Nastran solvers puts over 20 years of simulation technology development and experience to work on your engineering problem. You get the benefit of NEi Nastran’s proven field use in every major industry -- aerospace, automotive, maritime, defense, oil and gas, machinery, consumer, and medical. The solver technology has demonstrated its capabilities for reliable structural and thermal engineering results in demanding applications. Numerous examples and case studies have been accumulated over decades that demonstrate Nastran’s ability to capture and simulate real world physical behavior. You will find Nastran in high-profile projects like spacecraft, composite business jets, Formula1 race cars, naval ships, offshore platforms, Tour de France bicycles, and implantable medical devices, as well as their less visible components like, pumps, valves, derricks, housings, and brackets. Nastran is trusted by engineers who specialize in finite element analysis based on its reliable results in extensive real-world applications. For more on FEA solver technology check out Intel’s Parallel Universe magazine Issue #13 “Transforming Product Engineering: Fast, Accurate Finite Element Analysis (FEA)”. http://software.intel.com/en-us/intel-parallel-universe-magazine

NEi Nastran Key Features

**NEi Nastran Solver Bundles.** A comprehensive package of the most used and needed simulations for mechanical engineering provides affordable professional FEA.

**NEi Nastran Nonlinear.** Sophisticated capability to extend your analyses to the nonlinear realm can be added when you need this level for real world fidelity.

**NEi Nastran Materials.** The full spectrum of material models are covered -- metals, plastics, elastomers, nonlinear shape-memory alloys, bio-medical, and composites.

**NEi Nastran Composites.** Special features and tools provide both ease-of-use and comprehensive technical treatment to get the most performance out of composites’ anistropic properties.
Capabilities NEi Nastran in-CAD for SolidWorks

Model Geometry Access:
• Part geometry data is accessed directly through SolidWorks API
• Data accessed for finite element mesh generation and application of loads and boundary conditions
• Supports SolidWorks type surfaces, such as mid-surfaces and sheet metal
• Supports assembly analysis

Meshing:
• Global and local controls applied to part geometry with grid filter sizing
• Mesh control on arbitrary user defined regions
• Sketch line or curve meshing
• Free surface meshing: quads or triangles
• Continuous shell (quad or tri) meshing
• Auto mesh, loads and constraints update automatically with geometry changes
• Mesher status window and progress bar
• Display/hide shell element normals
• Reverse normals for shell elements
• Mesh validation checks - distortion, Jacobian, and skew
• Display/hide beam element direction and orientation
• Reverse beam element direction
• Display/hide beam element and shell element cross section
• 1D element cross section property definition
• Combined solid (3D) and shell (2D) meshing
• Combined solid (3D) and beam (1D) meshing
• Combined shell (2D) and beam (1D) meshing
• End release for 1D elements
• Display free edges
• Drilling DOF for shell elements (CQUAD4/CTRIA4)
• Mesh table, an intuitive tool to both globally and locally edit the mesh definition of parts within an assembly

Assembly Connectors:
• True surface contact
• Automatic contact
• Thermal contact resistance
• Self contact (NEi Explicit)

Loads and Boundary Conditions:
• Uniform pressure and force
• Directional pressure and force
• Acceleration loads (gravity)
• Enforced motion: acceleration, velocity, displacement (rotational/translational)
• Temperature, default temperature and heat flux
• Symmetric, antisymmetric, axisymmetric boundary conditions
• Fixed constraints on faces, edges and vertices
• Directional and prescribed constraints
• Thermal constraints
• Thermal body loads
• Initial conditions: temperature, velocity, displacement (rotational/translational)
• Custom colors and sizes for loads and constraints
• Loads defined using geometric entities
• Convection
• Conduction
• Radiation
• Heat generation
• Rotational velocity/acceleration
• From output (thermal)
• Load variation using arbitrary 3D scale factors
• Load variation with interpolation method as quadratic or linear with bi-directional definitions
• Total load magnitude
• Distributed load as force or moment
• Temperature load supporting in structural analysis with body selection

Element Library:
• 1D line (CBEAM, CBAR, CPIPE)
• 2D linear shell (CQUAD4 and CTRIA3)
• 2D linear shell with drilling DOF (CQUAD4 and CTRIA4)
• 2D parabolic shell (CQUAD8 and CTRIA6)
• 3D linear and parabolic tetrahedron (CTETRA)
• Composites with plates and shells
• Surface to surface contact with manual or automatic recognition of surfaces
• Edge to surface contact with manual or automatic recognition of surfaces
• Concentrated mass
• Connectors:
  • Spring (CUSHI)
  • Rigid elements (RBE2 and RBE3)
  • Rod (CROD)
  • Nonlinear cable (CCABLE)
  • Bolts
  • Capscrew

Materials:
• Isotropic
• Anisotropic (2D & 3D)
• Orthotropic (2D & 3D)
• Nonlinear materials
• Nonlinear elastic
• Elasto-plastic
• Plastic
• Hardening
  • Isotropic
  • Kinematic
  • Combined
• Yield
  • von Mises
  • Tresca
  • Mohr-Coulomb
  • Drucker-Prager
• Custom stress-strain curve
• Hyperelastic
  • Neo-Hookean
  • Mooney-Rivlin
  • Ogden
  • Yeoh
• Generalized polynomial
• Experimental data function
  • Simple tension/compression
  • Equibiaxial tension
  • Simple shear
  • Pure shear
  • Pure volumetric compression
• Temperature dependent property support
• Failure Theory
  • von Mises stress
  • Principal stress
• Support for NEi Explicit materials
  • Rigid (isotropic, orthotropic 2D)
  • Brittle
  • Concrete
  • Crushable foam
• Fatigue
  • S-N data
  • E-N data
• PFFA stiffness reduction factors (isotropic, orthotropic 2D and 3D)

Material Orientation:
• Vector projection
• Curve tangent
• Rotated curve tangent
• Translated curve tangent
• Surface U and V directions

Properties:
• 1D beam (PBEAM/PBEAML) and bar (PBAR/PBARL)
• 1D tapered beam (PBEAML)
• 2D plate (PSHELL) and composite (PCOMP) and (PCOMP/G)
• 3D solid (PSOLID)
• Contact (BSCONP)

Surface Contact:
• Automatic surface contact generation
• Manual surface to surface, edge to surface
• Self contact (NEi Explicit)
• General, welded, slide, rough, offset weld and RBE3 element contact types
• Static friction, penetration surface offset and max activation distance
• Advanced contact parameters in both manual and auto surface contacts
• Weld damage model capabilities
• Contact delete (NEi Explicit)
• Tools to shift entities from master to slave

Coordinate Systems:
• Cartesian, cylindrical and spherical coordinate systems
• Referencing global assembly, part or custom coordinate systems
• Display toggles

User Interface:
• Seamless integration with SolidWorks GUI
• Menu support for all features
• Toolbar shortcuts
• Modern tree view layout
• Query display of real time information on nodes and elements
• Highlight specific nodes and elements on the model
• Total number of nodes/elements displayed in assembly tree
• Section view for parts and assemblies
• Dynamic update of loads, constraints, and rigid bodies
• Redesign physical properties and constraints dialog

Post-Processing:
• Stress, deformation plots
• Principal and directional stress plot
• Strain plot
• Resonant frequencies, mode shape plots
• Temperature, heat flux plots
• Iso-surfaces
• Results across composite laminates
• Partial results generation for modal and transient analysis types
• Export Nastran input deck to other FEA systems
• Output within SolidWorks view with sensitive Help and analysis control, such as pausing and solution termination
• Export results using Femap Binary Neutral file format (FNO)
• Customizable material library
• Single and multi-set animations
• Max/min labels
• Results processed on selected parts of assemblies
• Dynamic result data display during nonlinear analysis
• Loads and constraints shown on deformed plots
• XY plot capability
  • Single-set
  • Multi-set
• Section cut capability
• Updated to more user friendly and intuitive tab format
• Post-processing the connectors results as elements
• Single tree results node for multi-set results
• Pop-up menu access to multi-set animation settings
• Default plot templates are grouped under Results
• Thermal analysis default plot templates for temperature and heat flux
• Fatigue default plot templates are grouped under Fatigue Results, Damage and Life
• Color bar option controls
• Visibility options within plot display
• Ability to hide/show deleted elements for Explicit post-processing
• Beam diagram support for bar elements
• Min/Max display enhanced within Section view
• Vector Display options to resize the vector graphics
• Min/Max display updated in Section view mode.
• Vector Display Options is added to resize the vector arrow
Capabilities NEi Nastran in-CAD for SolidWorks

- Upgraded post-processing tool bar with previous and next result, result set and ply results
- Generate report and XY plot in post-processing tool bar
- Display underformed edges
- Contour on beam and shell cross section
- Output file format and Result File type

Report Generation:
- HTML formatted reports for linear static analysis
- Customizable report format
- Step by step wizard for report generation process
- Includes standard model data

Graphics:
- Advanced OpenGL graphics taking advantage of the latest Computer Graphics chips
- 3D dynamic pan, zoom and rotation
- Hidden line and wireframe display
- Light source shading and transparency

- Multi-view display of Part/Assemblies
- Option to adjust graphics settings for lower performing graphics cards

Compatibilities:
- Nastran input file can be sent to any Nastran FE Solver including NEi Nastran, NX Nastran, or MSC.Nastran
- Binary results file in OP2 format usable by all Nastran solvers and wide variety of post-processors
- Part and Assembly geometry is fully compatible with SolidWorks' Parts and Assemblies

System Requirements:
- Intel Pentium® 4 or AMD based PC as a minimum, Intel Core™ i7, Xeon, AMD Opteron recommended
- 1 GB RAM minimum, more recommended
- 3.63 GB free hard disk space for installation, more required for simulation models
- Microsoft Windows 7 and Windows 8
- Compatible with SolidWorks 2014

International Languages:
- GUI = English, Japanese, Italian, French
- Technical documentation: English

NEI Software 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

For information on NEi Software, find a contact near you at http://www.nenastran.com/fea/locations
Table of Contents

1. How to Select FEA Software?
2. A Checklist for FEA Software
3. NEi Nastran in-CAD Basic
4. NEi Nastran in-CAD Expert
5. Additional Modules
How to Select FEA Software?

NEi Nastran in-CAD™

Your 3D CAD System + Nastran FEA = NEi Nastran in-CAD

You know the powerful benefits finite element analysis (FEA) technology can bring to your product development process. But, how should you evaluate and choose the right FEA software for your needs? What criteria should you apply? The product choices are wide and varied. In addition to offerings from a number of dedicated FEA software vendors, each CAD vendor offers its own brand of simulation software. Which route is best?

NEi Nastran in-CAD was developed to capture the technology advantages inherent in both 3D CAD and FEA systems. NEi Nastran in-CAD has the deep FEA technology base Nastran solvers have acquired from demanding engineering simulations in decades of field use, while keeping the familiarity, ease-of-use, integration, and power of your existing 3D CAD system.

Nastran FEA Solver Technology

NEi Nastran in-CAD uses NEi Nastran FEA solvers. NEi Nastran has over 20 years of field use with an established record of accuracy, proven results, well-developed capabilities, and wide industry acceptance. NEi Nastran has been used in a wide range of projects covering aerospace, automotive, maritime, medical, and consumer product industries. In addition, you will find NEi Nastran in the full spectrum of company sizes from well-known, global, Fortune 500 companies to small, independent, consulting engineers. Similarly, NEi Nastran can be found in high-profile, major engineering projects like SpaceShipTwo, Virgin Galactic’s commercial spacecraft venture, as well as practical, cost-effective detail design work like weight-saving brackets for aircraft interiors. NEi Nastran generates engineering solutions for the full spectrum of mechanical engineering problems providing virtual test results for structural deformation, stress, mode shapes, dynamics, impact, fatigue, contact, and thermal. For more on speed and accuracy in solver technology visit www.nenastran.com/fea/solver.php
A Checklist for Selecting FEA Software

In addition to the advantages of a tight coupling between 3D CAD and FEA systems, there are additional capabilities to consider in evaluating FEA software. Asking the right questions and making a thorough comparison of capabilities and performance metrics is the best way to determine that the FEA package you select will work for your product development process. Here are some fundamental considerations that will help you develop your checklist:

- What types of engineering analyses do I need to perform for my product?
- How do the materials I use in my designs affect my FEA choices?
- What FEA capabilities are necessary to get reliable simulation results?
- Who will I be sharing my analysis results with and what formats would be most productive?
- Is FEA affordable and what is the financial justification?
- What is important in support and service?

Types of Analysis

You know your product design process best, and the physical testing that is done to insure performance, quality, reliability, and safety. Listing the key structural, dynamic and thermal issues that you typically examine should be the basis for determining the capabilities you will want in your FEA software. Virtual testing with FEA software should help answer the pressing design questions you typically face.

NEi Nastran in-CAD analysis types come in two bundled packages – Basic and Expert. Basic contains the analyses that are most often used and needed in mechanical engineering and design. Plus it includes capabilities for contact and composite materials. Page 4 provides an easy, visual illustration and synopsis. Expert extends the capabilities in Basic from the linear to the nonlinear realm, and adds additional dynamic analyses, as well as powerful automated impact and drop testing. Page 5 summarizes Expert. In addition, several special analysis types are available individually as add-on modules. These include fatigue, explicit FEA for high speed and extreme deformation impact, and advanced techniques for composites. See page 6.

Support for wide spectrum of materials allows “what-if” studies.

Materials

Review the materials that you typically use in your designs, as well as the ones that may be under consideration in the future. Do you use metals? Plastics? Elastomers? Composites? Highly nonlinear materials like shape-memory alloys? Custom materials with specific performance data like bio-medical materials? Your analysis software needs to have modeling capabilities to handle the materials you work with. While most metals are easily handled by analysis packages because the material is isotropic and well-defined, other materials with predominantly nonlinear properties or idiosyncratic failure characteristics, like composites need to have capabilities in the software to accommodate their behavior.

NEi Nastran in-CAD has capabilities for a full range of engineering materials. See the Materials heading in the Capabilities section for a full itemized listing.

Achieving Reliable Simulations

Creating simulations that reflect real world behavior depend on multiple factors – appropriate loads and boundary conditions, a relevant element library, quality meshing, and solver accuracy. As with good CAD design, FEA requires attention to detail. Define the kind of parts you are going to analyze and the conditions they will see. Next, review the FEA software’s capabilities for adequate pre-processing to the FEA domain (i.e. loads, constraints, elements, mesh).

NEi Nastran in-CAD specifications are itemized in the Capabilities section. Professional modeling for a wide variety of structures and conditions is possible because of the extensive selection under
animations should be available for these audiences. A report generation feature is useful for both productivity and record keeping requirements.

NEi Nastran in-CAD Post-Processing, Report Generation, and Compatibilities are headings listed in the Capabilities section. The familiar environment and power of your 3D CAD system will further enhance post-processing and results sharing capabilities. Plus, the Nastran pedigree lets you share with the rest of the Nastran community – NEi, NX and MSC for wider collaboration.

Financial Justification and ROI
The ability to virtually test designs typically provides companies with cost savings by enhancing the existing design process. Design cycles become faster and less expensive by reducing the number of iterations, prototypes and physical tests. In addition, simulation provides technical insights that help detect problems earlier and makes physical testing more targeted and effective. Better products mean reduced warranty claims and avoidance of recalls. Innovation and faster time to market improves margins and revenue.

From a cost perspective, NEi Nastran in-CAD is made affordable by using your existing 3D CAD system as an FEA platform, and bundling analysis types that cover most mechanical engineering needs. Professional level entry, the flexibility to add advanced analysis capabilities when needed, and a familiar CAD system as a working environment further contribute to lower overall costs by providing a faster learning curve and removing the need to change FEA software systems as your ability and needs grow.

Technical Expertise
All NEi Nastran technical support is provided by degreed engineers with FEA industry experience. They handle telephone and email support as well as conduct training classes and mentoring sessions. Customer testimonials confirm the level of service, commitment and emphasis this portion of our software business receives, and why it is highly regarded in the industry. The development record and history of new capabilities, features, and enhancements in each release of NEi Nastran is an excellent indicator that you can expect NEi Nastran in-CAD software to keep your simulations technically advanced so your product development process maintains in competitive edge.

Customer Testimonials

"I've been one of the lead structural analysts/designers at Scaled Composites for the last 15+ years which included most of the analysis for the recent SpaceShipOne. Your good work in your product is already being used to help us design the first commercial man carrying spaceship. In other words, expect us to be bugging you a lot!"

Dan Kreigh
Lead Structural Analysts

"We have chosen NEi Nastran after an extensive and detailed internal benchmark, comparing the results, the performance, and the features of the solver developed by NEi Software, Inc. with those of our former FEA platform. Full compatibility, accuracy, along with the professionalism and quick turnaround of the tech support from NEi Software and SmartCAE were the main reasons why we selected NEi Nastran as our FEA software for the future."

Paolo Marabini
Structures and Calculation, Chief Engineer
**NEi Nastran in-CAD™ Basic**

**BASIC: A Portfolio of the Most Used and Needed Analyses**

NEi Nastran in-CAD Basic is an ideal way to strengthen your engineering analysis and product development capabilities through FEA simulation. A bundle of the most often used and needed simulations provides a cost effective way to get comprehensive technical treatment for the mechanical engineering design questions you routinely face. The inclusion of Assembly Modeling with Contact and Composites gives you a package of exceptional utility and equips you to handle a full spectrum of materials.

Assembly Modeling with Contact. You can go beyond analyzing individual parts. Assemblies with different contact can be modeled using sliding, friction and welded options. This sophisticated nonlinear modeling capability enables simulations with real world fidelity. This is typically an expensive add-on with most other solvers.

Composites. In addition to a library of materials, this suite of tools makes data entry, model creation and results analysis straightforward and less time consuming. Advanced features provide the visualizations and engineering data needed for high performance part design.

<table>
<thead>
<tr>
<th>Linear Statics</th>
<th>Composites</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear statics is one of the most common types of analysis. Determine stress, strain, and deformation resulting from applied static loads and imposed constraints.</td>
<td>NEi Nastran in-CAD’s suite of tools provides easy, straightforward handling of complex ply data. Analysis based on latest failure indices, like Puck and LaRC02, means reliable and insightful results.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Linear Steady State Heat Transfer</th>
<th>Prestress Static and Normal Modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Using the principles of conduction and convection heat transfer, engineers can examine designs for equilibrium temperature distribution.</td>
<td>Enables engineers to analyze structures subjected to initial stress, and model the effect of the initial stress state on the structures’ displacements, stresses, and modes.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Normal Modes</th>
<th>Thermal Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>Determines the undamped natural mode shapes and frequencies of structures allowing designers to explore and resolve problems with noise and vibration.</td>
<td>NEi Nastran in-CAD supports the analysis of structures subjected to thermal loads</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Buckling</th>
<th>Assembly Modeling with Contact</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allows designers to examine structures for sudden failure modes caused by compressive forces.</td>
<td>Go beyond analyzing individual parts. Real world simulation of assemblies is possible with sophisticated modeling of different kinds of contact interactions including sliding, friction and welded contact types.</td>
</tr>
</tbody>
</table>
EXPERT: Nonlinear Capabilities and Dynamics Take Simulation to the Next Level

NEi Nastran in-CAD Expert takes your analysis skills to the next level by providing nonlinear analysis capability to all the simulations in Basic. In addition, it adds several types of dynamic analyses along with the Automated Impact Analysis and Drop Test (AIA™). The analysis types in the NEi Expert bundle are itemized and shown in the graphic below.

Automated Impact Analysis (AIA™) provides an excellent introduction to the power that automated tools can bring to demanding simulation problems. AIA takes a very complex, time-consuming simulation task, simplifies and automates it. AIA requires a minimum of input data – projectile velocity and acceleration. AIA determines the time steps, duration, and complex contact interaction between projectile and target. AIA can provide a thorough and physically realistic simulation of impact because of this comprehensive treatment of the phenomenon. Note that this is much more useful and meaningful from an engineering standpoint than a simplistic imposition of force at a point found in other impact or drop tests. AIA provides insight into dynamic, implicit, nonlinear behavior in applications ranging from various projectile-target simulations, product drop tests, and virtual tests of packaging. Note that you can further your capabilities in impact analysis to high speed impact with NEi Explicit; see the section on Additional Modules.

Nonlinear Static
Provides capability for modeling material nonlinearity, contact, and large displacement and rotation, as well as transient and inertial effects.

Nonlinear Steady State Heat Transfer
Solves heat transfer problems with nonlinear thermal boundary conditions such as temperature dependent thermal properties.

Linear and Nonlinear Transient Response
Simulates the response of a structure through a period of time under the influence of constant or time-dependent loads e.g. impulse loading.

Random Response
Analysis of structural behavior in response to the imposition of random dynamic loads.

Frequency Response
Determines the structural harmonic response based upon frequency-dependent loads.

Nonlinear Transient Heat Transfer
Solves heat transfer problems with nonlinear thermal boundary conditions that vary through time e.g. power fluctuations.

Automated Impact Analysis (AIA™) and Drop Test
Sophisticated treatment provides realistic and meaningful impact and drop test simulations. The only inputs required are projectile velocity, and acceleration.

Advanced Nonlinear Material
Solver captures complex nonlinear phenomena such as plasticity, hyperelasticity, and shape-memory effect. This enables analysts to model a wide range of materials, from metals and shape-memory alloys to rubbers and soft tissue.
NEi Nastran in-CAD Additional Modules

**ADDITIONAL MODULES: Specific Analyses to Fit Your Needs**

NEi Nastran in-CAD is highly scalable in its range of simulations. In addition to the analysis types that are bundled in the Basic and Expert packages, the products below can be added individually when you find you need these specialized engineering capabilities.

Two fatigue packages, Multi-axial Static Fatigue and Vibration Fatigue address engineering for reliability in environments with repeated and cyclic loading conditions.

MultiContinuum Theory (MCT™) and Progressive Ply Failure Analysis (PPFA™) provide capabilities for extending the thoroughness and sophistication of your composite analysis which is often required for high performance applications.

NEi Explicit is a parallel explicit FEA solver that is completely integrated with the NEi Nastran environment. It gives you the power to do simulations of high-speed impacts like those in crash, explosion and ballistics scenarios. Similarly, extreme material deformation processes can be analyzed as are found in metal forming manufacturing processes.

---

**Multi-axial Static Fatigue**

Fatigue life based on stress-life (S-N) theory and strain-life (E-N) theory can be calculated. Multiple time history loads are input to a static analysis along with material stress versus cycle, or strain versus cycle data. Life expectancy and accumulated damage is determined.

---

**Vibration Fatigue**

Calculate damage and life from dynamic conditions using stress-life and strain-life material data as input. Examples of conditions that can be simulated include road vibration, wave cycles, engine vibration, and wind loads.

---

**NEi Explicit**

Simulation for problems involving high speed impact involving extreme deformation, multi-body contact, highly nonlinear material response, and conditions where there is penetration or tearing of the material e.g. bird strike used for aircraft safety studies and ballistics simulations.

---

**Progressive Ply Failure Analysis (PPFA™)**

Provides an understanding of the progression of events from first ply failure (FPF) to ultimate failure in composites. This advanced analysis allows designers to take advantage of residual strength and optimize size and weight for high performance applications.

---

**MultiContinuum Theory (MCT)**

MultiContinuum Theory, a micro-mechanics based approach to high performance composite analysis, achieves more accurate failure results by handling fiber and matrix properties separately as opposed to using an aggregated homogeneous representation.
About NEi Software

NEi Software is a leading innovator and global provider of Nastran 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.

Contact Information

NEi Software 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

For information on NEi Software, find a contact near you at http://www.nenastran.com/fea/locations