# 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?

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.

Your 3D CAD System becomes your FEA Platform

NEi Nastran in-CAD embeds Nastran FEA technology in your 3D CAD system allowing its solid modeling capabilities to provide FEA pre- and post-processing. Part geometry data is accessed directly through your system’s API. You get compatibility and integration, as well as the same look-and-feel, menu, and tree-type structures of your CAD system when you develop your FEA model. Similarly, you are in a familiar working environment when you review the results of the FE analysis. Using your 3D CAD system as your FEA platform reduces your learning curve, maintains your productivity, eliminates any compatibility issues arising from importing CAD models into different FEA platforms, and lowers your cost for professional level FEA software.

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.

Composites require tools for simplifying ply information and extracting important technical insights.

Composites

We break out composites from other materials in NEi Nastran analysis capabilities because these materials have a number of significant and unique requirements. These manifest themselves in most steps of the FEA process from model creation to meshing and post-processing. Specialized tools in NEi Nastran in-CAD make your work with composites both productive and technically insightful. Progressive Ply Failure Analysis (PPFA™), modern failure analysis indices like Puck and LaRC02, and Multi Continuum Theory assure thorough technical treatment and understanding of your design's performance. Ease-of-use comes with features that simplify definition of material properties, ply layups, orientation, and the ability to locate problem areas down to specific plies. NEi Nastran in-CAD's package of comprehensive features allows you to get the most from designs using composite materials.

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 the Materials heading in the Capabilities section.
each of the FEA domain categories. NEi Nastran solvers’ reputation for accuracy is assured by comprehensive validation with each new release and decades of real-world application history. Computational accuracy is foremost and fundamental in Nastran solver design. Robust software design, development methodology and architecture avoid compromises and preserves solver precision.

Sharing Results

In addition to colleagues, consider others that will need access to your results – design partners, management, suppliers, and customers. Post-processing, graphs, images, and 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 its 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.

- **Linear Statics**: Linear statics is one of the most common types of analysis. Determine stress, strain, and deformation resulting from applied static loads and imposed constraints.
- **Composites**: 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.
- **Linear Steady State Heat Transfer**: Using the principles of conduction and convection heat transfer, engineers can examine designs for equilibrium temperature distribution.
- **Prestress Static and Normal Modes**: 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.
- **Normal Modes**: Determines the undamped natural mode shapes and frequencies of structures allowing designers to explore and resolve problems with noise and vibration.
- **Thermal Stress**: NEi Nastran in-CAD supports the analysis of structures subjected to thermal loads.
- **Buckling**: Allows designers to examine structures for sudden failure modes caused by compressive forces.
- **Assembly Modeling with Contact**: 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.
**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.

- **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.

- **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.

- **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.

<table>
<thead>
<tr>
<th>Multi-axial Static Fatigue</th>
<th>NEi Explicit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculates fatigue life based on stress-life (S-N) theory and strain-life (E-N) theory. Multiple time history loads are input along with material stress versus cycle, or strain versus cycle data. Life expectancy and accumulated damage is determined.</td>
<td>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.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Vibration Fatigue</th>
<th>MultiContinuum Theory (MCT)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculates 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.</td>
<td>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.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Progressive Ply Failure Analysis (PPFA™)</th>
<th>Helius/MCT Predicted Failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>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.</td>
<td>Red = Fiber Failure, Green = Matrix Failure</td>
</tr>
</tbody>
</table>
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