



# Fusion™

3D CAD Modeler + Nastran FEA

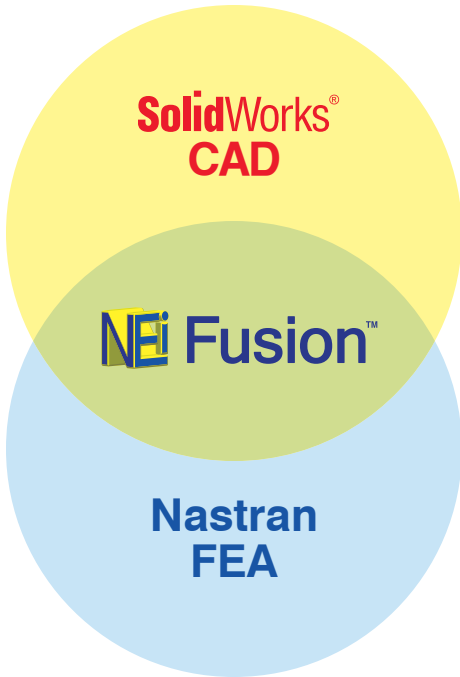
**2. Modeler Powered  
by SolidWorks +  
Nastran FEA**

**3. Features and Benefits  
Highlights**

**4. Analysis Types**

**7. Capabilities**





NEi Fusion joins two powerful technologies — SolidWorks parametric CAD and high accuracy Nastran solvers.

## A New Approach to Analysis and Simulation

NEi Fusion represents a new approach to engineering analysis and simulation software. NEi Fusion joins two powerful technologies — SolidWorks 3D feature-based, parametric CAD for model creation, and high accuracy, industry-proven, Nastran solvers for solution generation. This combination provides a modeling environment that is familiar and user friendly to your entire product development team and gives professional level results acceptable to the community of Nastran analysts as well as your customers, vendors, and design partners. NEi Fusion can virtually test parts for a wide range of static and dynamic structural and thermal conditions quickly in a CAD environment that is friendly to changes and exploring design alternatives. The combination of parametric CAD and Nastran is a first in the industry and at its precedent-setting price point will enable wide use throughout your organization. In addition, NEi Fusion includes a comprehensive suite of features for evaluation, presentation, and reporting of simulation results and the Nastran foundation insures you will have a continuous consistent migration path to higher end analysis if needed.

## Can You Afford to Omit Simulation from Your Development Process?

Why wait until prototypes are built, expensive test fixtures developed or actual field usage to find out how your design will perform? NEi Fusion lets you apply forces, pressure, thermal conditions, temperature, vibration, impact loads and more. You see the deformation, stresses, strains, heat transfer, and modal shapes that your design will experience. Through a variety of highly visual images, contour plots, animations, graphs, and output data, you get the engineering insight you need to innovate and optimize, so you can achieve the best quality, lowest manufacturing costs, and fastest time to market in today's hyper competitive global marketplace.

## Ease in Exploring Design Alternatives, Confidence in Results

NEi Fusion combines an FEA Modeler, comprehensive pre- and post-processing capabilities, and Nastran solvers to create an analysis package for small and medium size companies, consultants, and any engineer who needs affordable, professional level simulation for product development, virtual testing, design validation, and quality assurance. Parts and assemblies can be analyzed for a wide spectrum of static and dynamic structural and thermal loading.

## NEi Fusion Modeler

The NEi Fusion Modeler is built on SolidWorks' industry-proven, Windows-based, fully associative, parameterized, feature-based, modeling engine that provides a full set of advanced and powerful tools for fast, easy, intuitive, and robust model creation. A wide range of CAD data import capabilities complements the 3D modeling tools to help you build your models quickly when existing CAD files are available.

## NEi Fusion Pre-Processor

A comprehensive element library, sophisticated meshing capabilities, and extensive material library insure you get real world fidelity and professional level simulation.

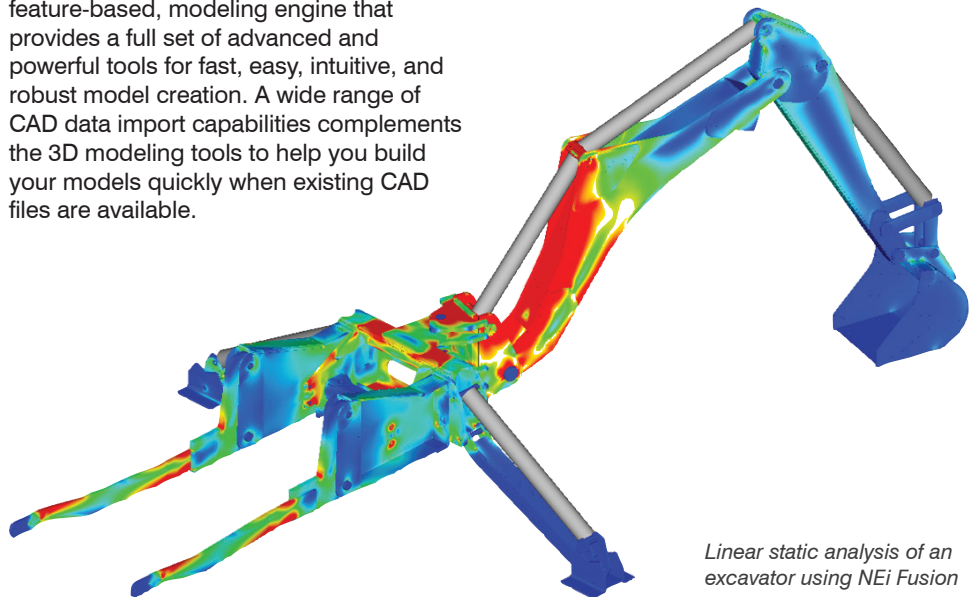
## NEi Fusion Nastran Solvers

NEi Fusion employs the same industry regarded NEi Nastran Solvers found in all of our analysis products. That means you can expect solutions that are accurate, precise, and reliable. Nastran assures that you achieve the best possible real world representation along with extremely fast turn around time on solutions.

**"NEi Fusion employs the same industry regarded NEi Nastran Solvers found in all of our analysis products... you can expect solutions that are accurate, precise, and reliable."**

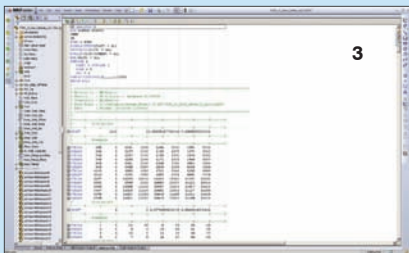
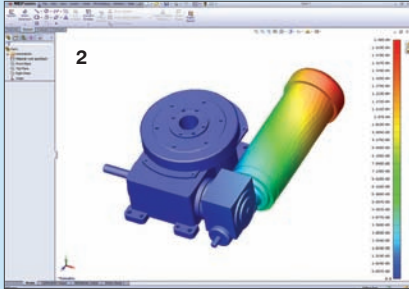
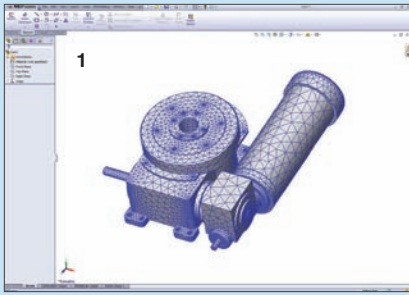
## NEi Fusion Post-Processor

NEi Fusion post-processing not only provides a wide variety of images, graphs and data with which to view your simulation results, but also is equipped with wide file sharing and import/export capabilities. You can easily share your FEA models and results with vendors, design partners, suppliers, customers, and other segments of your organization — an important consideration in today's highly collaborative work environment.

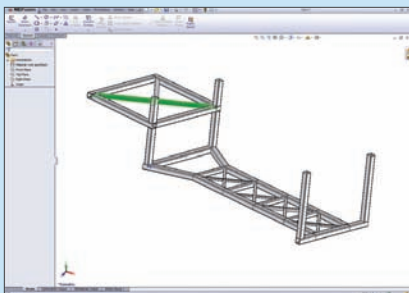


Linear static analysis of an excavator using NEi Fusion

# Features and Benefits Highlights



NEi Fusion gives designers easy-to-use SolidWorks modeling (1) with Nastran Finite Element Analysis and highly accurate simulation results (2); it is unique and forward-looking in its ability to share this data with the world of Nastran analysts (e.g. NEi, NX, MSC) via the Bulk Data File (3).



Beam element enhancements have simplified model creation.

## Why NEi Fusion™?

NEi Fusion is the first design, validation, and optimization solution that “fuses” technologies such as:

- 3D feature-based, parametric CAD for easy model creation
- Industry-proven Nastran solvers for precise solution generation
- A full range of 1D, 2D, and 3D element types such as: beams, bars, springs (CBUSH), linear and parabolic plates and shells (quadrilateral and triangular), as well as linear and parabolic tetrahedral elements
- File sharing via the Nastran Bulk Data File allowing your product development team to collaborate with analysis departments, customers, vendors, and design partners
- Plus NEi Fusion comes at a precedent-setting, affordable price point that will enable wide usage throughout your organization

## Easy

NEi Fusion is a breakthrough product fusing easy-to-use, industry-leading SolidWorks 3D solid modeling interface with a comprehensive element library, sophisticated meshing capabilities, and extensive material library. This fusion assures design professionals of lower learning curves that translate into faster design cycles, and ultimately, faster time to market.

## Powerful

NEi Fusion’s robust feature set allows users to explore a range of analysis methods including: linear statics, steady state heat transfer, normal modes, buckling, prestress, dynamics, and nonlinear. Additionally, every NEi Fusion user has our extended capabilities with assembly analysis, composite materials, surface contact, optimization, drop testing, and the ability to output to a Nastran Bulk Data File. This greatly promotes efficient workflow within organizations, and increases opportunities for consultants looking to expand their services into the automotive, aerospace, and maritime industries.

## Precise

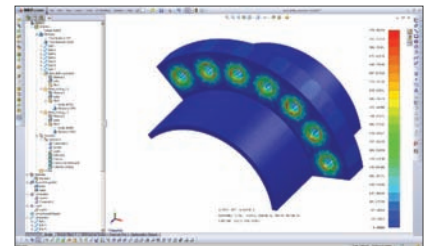
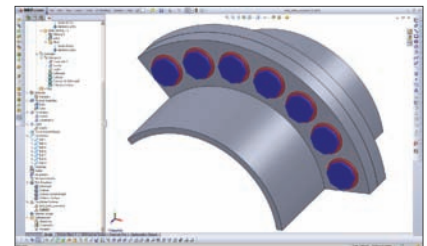
Nastran is the most widely used FEA software in the world. It has been proven over several decades and has become the established standard in aerospace, automotive and maritime industries. NEi Nastran has achieved industry acceptance because of the fidelity of its precise and accurate results.

## Efficient

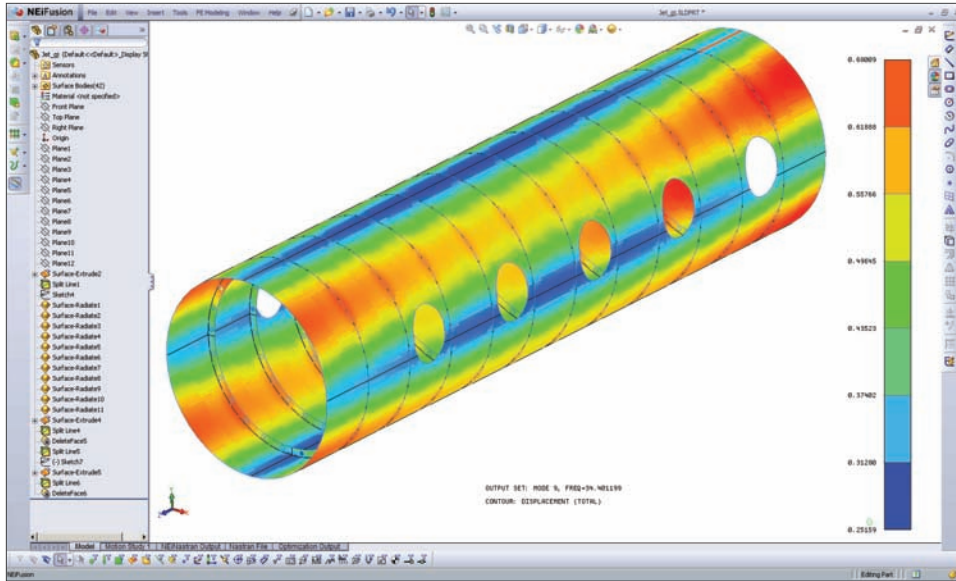
NEi Fusion is also unique because of its ability to create a collaborative work environment between designers and analysts, ultimately removing the traditional barriers between these groups. Designers can explore “what-if” scenarios easily with full associativity of FEA and CAD data and validate and optimize concepts. Models can be shared or passed off to analysts for more intensive examination using a portfolio of pre-posts if necessary.

The Nastran Bulk Data File provides maximum efficiency in sharing, and administration of software resources. This efficiency is extended to any design partner such as customers or vendors who utilize the Nastran standard.

**“NEi Fusion is also unique because of its ability to create a collaborative work environment between designers and analysts, ultimately removing the traditional barriers between these groups.”**



Bolts and cap screw joints can be automatically created by selecting appropriate geometry



FEA analysis of a jet fuselage using NEi Fusion

## The Perfect Package to Get Started

NEi Fusion is the perfect package for getting started in Finite Element Analysis (FEA) because it combines two outstanding technologies at a fraction of the cost of high-end FEA offerings. The highly-regarded, industry-proven Nastran FEA solver is a platform that can grow with your simulation needs, while SolidWorks insures a familiar working environment that combines ease of use with the latest high performance tools for 3D CAD model creation. NEi Fusion gives product development professionals the most widely used and needed analysis capabilities — linear statics, steady state heat transfer, normal modes, buckling and prestress, plus additional capabilities for handling composite material analysis, performing optimization analysis, and modeling assemblies with contact.

Check out these analysis capabilities to see how NEi Fusion can bring engineering insight to your design process and help you spot problem areas, optimize performance, and virtually test before building expensive prototypes and fixtures. In short, save time and money while building more innovative, higher quality products.

### Linear Statics

Linear statics is one of the most common types of analyses needed by design engineers. You apply loads and constraints to your parametric part and the NEi Nastran solver provides results which can be displayed in a wide variety of formats showing stress, strain, and deformation.

### Linear Steady State Heat Transfer

Using the principles of conduction and convection heat transfer, engineers can examine designs for equilibrium temperature distribution.

### Normal Modes

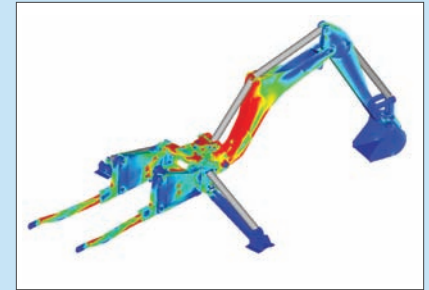
Determines the undamped natural mode shapes and frequencies of structures allowing designers to explore and resolve problems with noise and vibration.

### Buckling

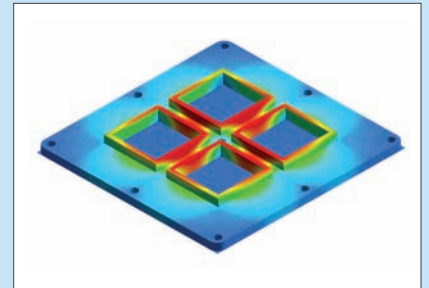
Allows designers to examine structures for sudden failure modes caused by compressive forces.

### Prestress

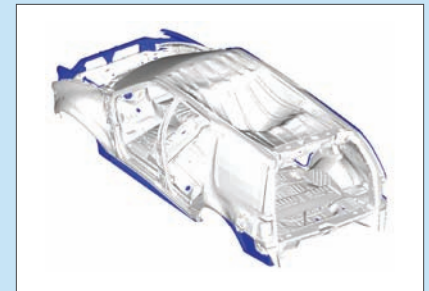
Allows the user to induce an initial stress state on structures such as rotor blades or taut strings.



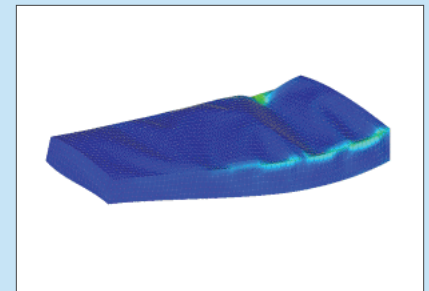
Linear Statics



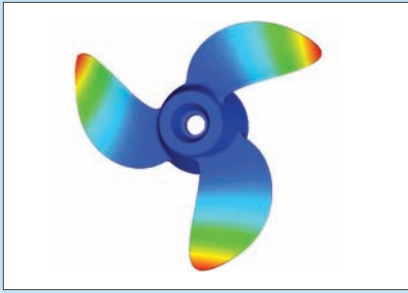
Linear Steady State Heat Transfer



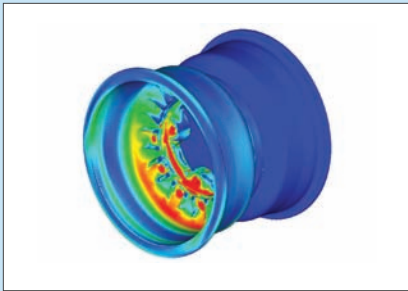
Normal Modes



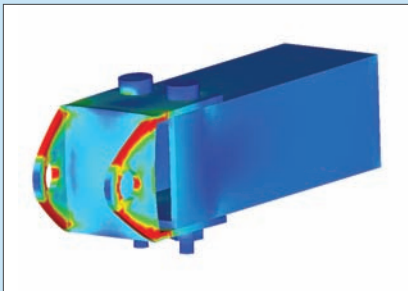
Buckling



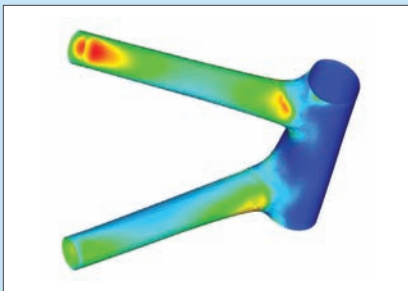
Prestress



Optimization



Assembly Modeling with Contact

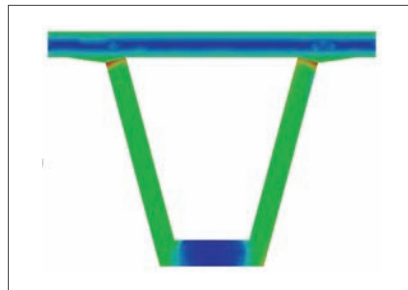


Composites

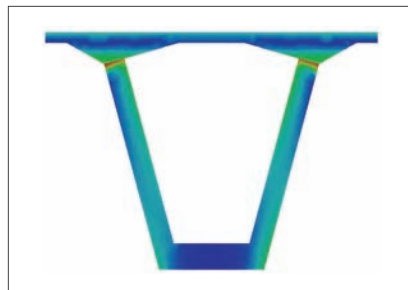
## Optimization

Designers need to be equipped with the latest and best technology to meet the demands of global competition. One of the many ways NEi Fusion meets this need is with Optimization Analysis. This optimization module, developed by Red Cedar Technology, provides computer aided engineering help for stubborn design problems where conflicting parameters make it difficult to find solutions.

- Support for Parameter and Shape optimization using continuous or discrete variables
- Optimization constraints can be implemented for temperature, laminate and shell thicknesses, material properties, von Mises stress, weight, eigenvalues, and displacement
- Optimization objectives for temperature, laminate and shell thicknesses, material properties, von Mises stress, weight, eigenvalues, and displacement can be minimized, maximized, or targeted
- Model parameters and model dimensions can be made as optimization variables
- Optimization can be multi-objective with multiple subcases within all solutions
- Support for 2D (shells) and 3D (solids) elements



Bridge structure before optimization



Bridge structure after optimization

## Assembly Modeling with Contact

NEi Fusion allows designers to go beyond analyzing individual parts. Assemblies with different kinds of contact can be modeled including sliding, friction, and welded, allowing simulations to attain real world fidelity. This sophisticated nonlinear modeling capability is typically an expensive add-on to most solvers, but with NEi Fusion, it comes standard.

**“This optimization module provides computer aided engineering help for stubborn design problems where conflicting parameters make it difficult to find solutions.”**

## Composites

In addition to a library of materials, NEi Fusion contains a suite of tools designed to bring engineering insight to the analysis of composites and laminated products. These tools help make the process straightforward and less time consuming.

- Clear, easy input of strength and stiffness terms with access to material libraries
- Ply lay-up definition, in which stacking sequences can be easily entered and modified, and symmetry can be used to reduce entries
- Easy orientation definition created by projecting a cylindrical or spherical coordinate system onto a structure to improve ply orientation
- Laminate property support for both 2D shell elements and 3D layered solid elements
- Post-processing visualizations that pinpoint problems at the ply level and logical paths from stress analysis of a structure to failure index results
- Support for advanced failure criteria including the most modern forms of Puck and NASA Langley LaRC02, as well as strength ratios for easy linear scaling and qualitative analysis
- Simple Material Property and Layup Editors that allow easy composite ply definition while providing advanced features for material property entry

## Taking Simulation to the Next Level

NEi Fusion also has the following analysis types: nonlinear statics, transient response, frequency response, Automated Impact Analysis (AIA), and nonlinear steady state and transient heat transfer. NEi Fusion is perfect for taking simulations to the next level since nonlinear is typically the next step in sophistication needed by design analysts. AIA also provides a widely needed simulation with applications ranging from projectiles to drop tests to packaging.

### Nonlinear Static and Nonlinear Transient Response

Allows material nonlinearity (material stress-strain data), contact (opening and closing of gaps and sliding), and large displacement and rotation (large deflection) effects to be captured in analysis models. Additionally, transient and inertia effects can be included.

“NEi Fusion is perfect for taking simulations to the next level since nonlinear is typically the next step in sophistication needed by design analysts.”

### Transient Response

Used to determine the response of a structure through a period of time under the influence of constant or time-dependent loads. Dynamic situations such as impulse loading can be modeled.

### Frequency Response

Determines the structural harmonic response based upon frequency-dependent loads or enforced motions such as displacements, velocities, or accelerations.

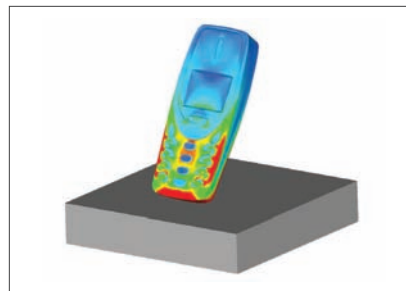
### Nonlinear Steady State and Transient Heat Transfer

Finds the steady-state solution to heat transfer models involving radiation, temperature dependent material properties, or other thermal boundary conditions such as convection or heat generation.

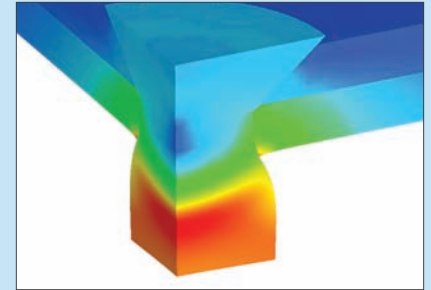
Nonlinear thermal boundary conditions can vary through time. Thermal pulsations in exhaust systems and power fluctuations upon microchips can be modeled.

### Automated Impact Analysis (AIA™)

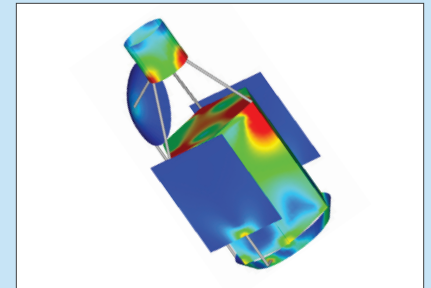
Ideal for performing projectile impacts and virtual drop tests, AIA provides an excellent introduction to the power that automated tools can bring to demanding simulation problems. AIA takes a very complex, time-consuming simulation task and simplifies and automates it. AIA requires a minimum of input data for the analysis — 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. 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 providing insight into dynamic, implicit, nonlinear behavior or real world impact problems.



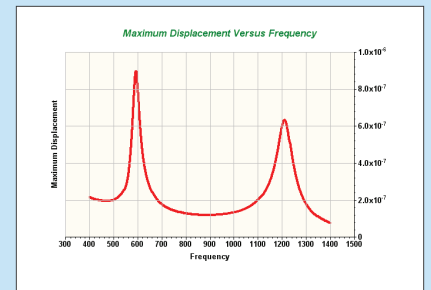
Automated Impact Analysis (AIA) can also be used as a virtual Drop Test as in the case of this cell phone.



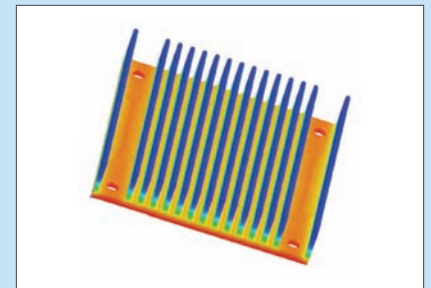
Nonlinear Static and Transient Response



Transient Response



Frequency Response



Nonlinear Steady State and Transient Heat Transfer

## General Capabilities

- Full single-window integration between solid modeling and analysis
- Direct use of CAD geometry for analysis
- Direct application of analysis input data to CAD geometry
- 3D visualization of analysis results on original CAD geometry
- FeatureManager™ for geometry, analysis and result visualization data
- Customizable analysis tree
- Dynamic editing of all geometric and analysis features
- Powerful configuration management for easy “what if” design variations (geometrical and physical)
- Comprehensive support for bidirectional CAD data exchange with most major CAD packages
- Comprehensive, context-sensitive HTML-based help system and tutorials
- Query displays real time information on model's nodes and elements
- Nastran Bulk Data File
- FE model file can be saved with only geometric data for smaller file size
- User specified solver path
- Highlight specific nodes and elements on the model

## CAD Interoperability

- Native file translators to and from nearly all mechanical CAD products on the market today: SolidWorks®, Pro/ENGINEER®, IPT (Autodesk Inventor®), Mechanical Desktop®, Unigraphics®, PAR (Solid Edge®), CADKEY®, IGES, STEP, Parasolid®, SAT (ACIS®), VDA-FS, VRML, STL, DWG, DXF™, TIFF, JPG, Viewpoint, RealityWave, HSF (Hoops), CATIA®
- Supported standards: ANSI, DIN, ISO, GOSJIS, GB and BSI

## Part Modeling

- Feature-based, fully associative, parameterized solid modeling
- FeatureManager™ dynamic design tree (e.g. re-order, drag & drop, etc.)
- Integrated sketching (dynamic referencing)
- Extrudes, revolves, feature patterns, holes, etc.
- Advanced 3D operations, e.g. lofting, sweeping, complex blending, filleting, etc.
- Advanced shelling, midsurfaces
- Advanced surface modeling: lofts and sweeps with guide curves, fill-in holes, drag-handles for tangency control, etc.
- Trimming, extending, filleting, and knitting surfaces
- Translating, rotating, copying, and mirroring surfaces
- Support for creating 3D models from existing 2D data, e.g. 2D-to-3D extrusion, etc.
- Multiple design variations with Configuration Management, DesignTables

## Assembly Modeling

- Fully associative: referencing of other parts and maintaining relationships when creating new parts
- Complete range of mating conditions, snap-to-fit SmartMates™
- Locating conflicting mate relationships with Mate Diagnostics
- Dynamic assembly visualization
- Real-time previewing of components, parts
- Easy designing and changing of parts and subassemblies from within an assembly
- Mirrored components to create new parts and assemblies based on existing designs
- Multiple assembly design variations with Configuration Management for easy “what if” design scenarios

## Assembly Connectors

- True surface contact
- Automatic contact

## Meshing

- Mesh validation checks: Distortion, Jacobian, Skew
- Global and local controls for part geometry with default sizing
- Mesh control on arbitrary user defined regions
- Combined shell and beam/bar meshing
- Sketch line or curve meshing

- Free surface meshing: quads or triangles
- Continuous shell (quad or tri) meshing
- Auto mesh, loads and constraints update with geometry changes
- Mesher status window and progress bar
- Display/hide shell element normals
- Reverse normals for shell elements
- Display/hide beam element orientations
- Display/hide 1D/2D cross-section

## Element Library

- 1D line (CBEAM, CBAR, CPIPE, PBARL, PBEAML)
- 2D linear shell (CQUAD4 and CTRIA3)
- 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
- Rigid elements
- Concentrated mass element
- Spring Element: transmits tension, compression also allows damping (CBUSH)
- Rod Element: transmits tension, compression and torsion
- Cable Element: transmits tension with optional bending stiffness

## Loads and Boundary Conditions

- Uniform pressure and force on faces, edges and vertices
- Directional pressure and force
- Acceleration loads (gravity)
- Enforced motions: acceleration velocity displacement (rotational and translational)
- Temperature
- Symmetric, antisymmetric, axisymmetric boundary conditions
- Fixed constraints on faces, edges and vertices
- Directional and prescribed constraints
- Thermal constraints
- Thermal body loads
- Initial temperature conditions
- Custom colors and sizes for loads and constraints
- Load defined using edge
- Load variation using arbitrary 3D scale factors
- Conduction
- Convection
- Heat Generation
- Heat Flux
- Rotational Velocity/Acceleration
- Radiation
- From output (thermal)

## Material Properties

- Isotropic
- Orthotropic 2D and 3D
- Solid anisotropic materials (MAT9)
- Hyperelastic materials: Neo-Hookean, Mooney-Rivlin, Ogden, Yeoh, Generalized Polynomial (nonlinear)
- Temperature-dependent property support
- Nonlinear materials: nonlinear elastic, elasto-plastic, plastic
- Hardening: isotropic, kinematic, combined
- Yield criteria: Von Mises, Tresca, Mohr-Coulomb, Drucker-Prager
- Custom stress-strain data

## Surface Contact

- Automated Surface Contact Generation (ASCG™)
- Automated Edge Contact Generation (AECG™)
- General, welded, slide, rough, offset weld and RBE3 element contact types
- Static friction

## Coordinate Systems

- Cartesian, cylindrical and spherical coordinate systems
- Referencing global assembly, part or custom coordinate systems

## Analysis Types

- Linear statics
- Normal modes
- Linear buckling
- Thermal stress
- Prestress static
- Contact analysis in assemblies
- Linear steady state heat transfer
- Nonlinear statics
- Modal and direct transient response
- Modal and direct frequency response
- Nonlinear steady state and transient heat transfer
- Nonlinear transient response

## Composite Analysis

- Various failure theories supported: Hill, Hoffman, Tsai-Wu, Max. stress, Max. strain, NASA LaRC02

## Optimization Analysis

- Design objectives to minimize, maximize or reach target values
- Parametrically update geometry dimensions
- Optimize weight, stress, material properties, temperature, eigenvalue, plate and laminate properties thickness

## Automated Impact Analysis (AIA™) and Drop Testing

- Automatic impact wizard
- Acceleration and contact direction input
- Time stepping automatically calculated based on natural frequency

## 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
- Export Nastran input deck to other FEA systems
- Customizable material library
- Single and multi-set animations
- Max/min labels
- Slice view provides section views of contour plot
- Post-process selected parts in an assembly
- XY plots can be viewed from tabular data
- Partial results generation for modal and transient analysis types
- Loads and constraints shown on deformed plots

## 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

- 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

## Compatibilities

- Part and Assembly geometry is fully compatible with SolidWorks' Parts and Assemblies
- 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

## Language Support

- GUI: English, Japanese, Italian, French
- Technical documentation: English

---

## About NEi Software

NEi Software is a world leader in 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.

### Global 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](mailto:info@neisoftware.com)  
Website: [www.NEiSoftware.com](http://www.NEiSoftware.com)

### NEi Software EMEA Office

The Old Barrel Store  
Draymans Lane, Marlow  
Buckinghamshire, SL7 2FF  
United Kingdom

Phone: +44 (0)1628-400645  
Fax: +44 (0)1628-891701  
E-mail: [emea@neisoftware.com](mailto:emea@neisoftware.com)  
Website: [www.NEiSoftware.com/emea](http://www.NEiSoftware.com/emea)

### NEi Software Asia Office

Shinjuku Park Tower  
N30th Floor 3-7-1 Nishi-Shinjuku  
Shinjuku-ku, Tokyo, 163-1030  
Japan

Phone: +81-(0)3-5326-3062  
Fax: +81-(0)3-5326-3001  
Email: [asia@neisoftware.com](mailto:asia@neisoftware.com)



NEi, NEi Works, NEi Fusion, and the NEi logo are trademarks of NEi Software, Inc. Nastran is a registered trademark of NASA. All other trademarks are the property of their respective owners. Copyright © NEi Software, Inc. 2010. All rights reserved.

BROCFUS120100430