

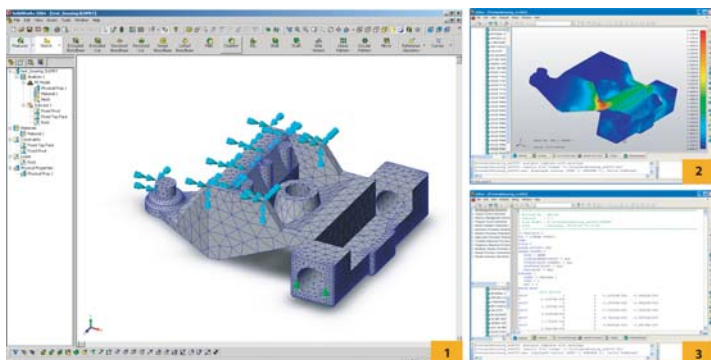
NEiWorks V1.12 Overview

High Performance Embedded Nastran FEA for SolidWorks.

NEiWorks: Nastran Finite Element Analysis (FEA)

As a SolidWorks® user, you appreciate the ease-of-use, integration, productivity and communication tools that let you produce high quality, accurate designs and get them to market faster. Similarly, you want your FEA software to provide the same benefits in the area of engineering analysis and simulation. You want to gain insight into different aspects of your design -- structural, thermal, dynamic, and verify critical design elements on your desktop before you commit resources to expensive and time consuming prototypes and test programs.

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. Nastran has achieved this level of acceptance because of its accuracy, precision, and fidelity of its analysis results. However, for most of its history, Nastran FEA has been a tool confined to high end analysis in very large corporations because of the cost and sophistication of the hardware and software. Now Noran Engineering, Inc. (NEi) has developed a product that allows Nastran FEA technology to move in a different direction making it affordable, easier to use, and accessible to a much wider engineering and design community.

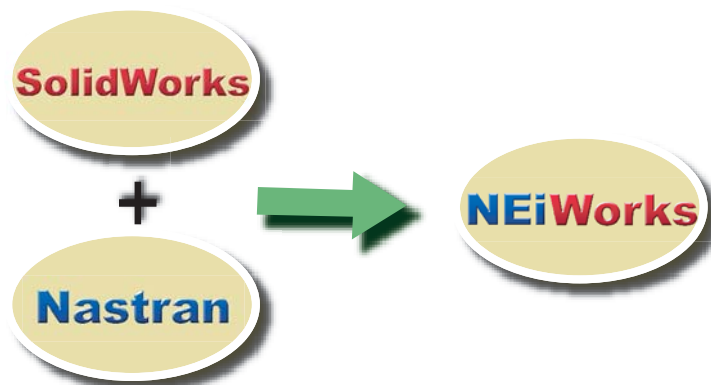


NEiWorks gives SolidWorks designers easy-to-use embedded Finite Element Analysis (1 and 2), highly accurate simulation results, and is unique and forward looking in its ability to share this data with the world of NASTRAN analysts (e.g. NEi, NX, MSC) via the Bulk Data File (3).

NEiWorks: Recognized, Powerful, Affordable Nastran FEA

Introduced in December 2004, NEiWorks received recognition from *NASA Tech Briefs* magazine, being named as a finalist in its Product of the Year Competition. *Desktop Engineering* magazine also picked NEiWorks in its Editor's choice. NEiWorks has qualified for SolidWorks Corporation's Gold Product status. This certification represents the highest level integration providing full associativity, sharing the same design data, and automatic updating across applications.

Most importantly, SolidWorks users will find NEiWorks extremely affordable because it can be purchased in a configuration that matches their work environment. For example, network installations can benefit from sharing solvers over multiple users with multi seat pricing. NEiWorks comes in Basic or Expert versions with affordable maintenance and upgrade options.



NEiWorks: Easy to Use, High Accuracy, Flexible Configurations, Share Results

NEiWorks is a breakthrough product. Embedded in SolidWorks, NEiWorks integrates highly accurate Nastran FE analysis with SolidWorks' easy to use 3D solid modeling. NEiWorks retains the same look-and-feel, menu, and tree type structures of SolidWorks. Users experience the familiar environment and ease of use they have come to expect while getting high accuracy analysis capabilities. NEiWorks can provide a wide spectrum of analysis and simulation tasks depending on the solvers selected and the configuration package. You can start with basic linear statics or move up to the most sophisticated configuration which allows access to the full power of the Nastran solvers through other pre- and post-processors for high level virtual simulation of nonlinear transient analysis of structural, thermal, and dynamics problems. In this age of collaborative design, NEiWorks provides you with another important advantage. If you work in an industry where Nastran is an accepted standard, NEiWorks lets you share results with the entire community of Nastran users, like NEi, NX and MSC using text files for input data and results via standard OP2 format - an important consideration for communicating with customers, suppliers or design partners.

Noran Engineering, Inc.

USA HEADQUARTERS

Noran Engineering, Inc
5555 Garden Grove Blvd., Ste 300,
Westminster, CA 92683-1886, USA
Phone: 1.714.899.1220
Fax: 1.714.899.1369
E-mail: info@noraneng.com
Website: www.neinastran.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

UNITED KINGDOM

Epsilon Structural Analysis Ltd.
Suite 9, Premier House, Argyle Way
Stevenage, Herts SG1 2AD, UK
Phone: +44.0.870.190.9431
Fax: +44.0.870.190.9432
E-mail: info@epsilon-sa.com
Website: www.epsilon-sa.com

SCANDINAVIA

FEMComp Engineering
Nygårdsgatan 12
SE-722 19 VAESTERAS, SWEDEN
Phone: +46.21.35.00.45
E-mail: info@femcomp.com
Website: www.FEMComp.com

INDIA

WaveAxis
#4, 3rd Floor, K.R. Garden,
5th Main, Koramangala 8th Block
Bangalore, 560095, INDIA
Phone: +91.80.51308070
Fax: +91.80.51308071
E-mail: info@waveaxis.com
Website: www.waveaxis.com

JAPAN

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

✓ Model Geometry Access

- Part geometry data is accessed directly through SolidWorks API
- Supports assembly analysis
- Data accessed for finite element mesh generation and application of loads and boundary conditions

✓ Assembly Connectors

- True surface contact
- Thermal contact resistance

✓ Meshing

- Global and local controls for part geometry with default sizing
- Mesh control on arbitrary user defined regions
- Free surface meshing: quads or triangles
- Auto mesh, loads and constraints update with geometry changes
- Mesher Status Window

✓ Loads and Boundary Conditions

- Uniform pressure and force on faces, edges and vertices
- Directional pressure and force
- Acceleration loads (Gravity)
- Enforced displacement and rotations
- Temperature, default temperature and heat flux
- Symmetric, antisymmetric, axisymmetric boundary conditions
- Fixed constraints on faces, edges and vertices
- Directional and prescribed constraints
- Thermal constraints

✓ Element Library

- 3D solid: tetrahedron both linear or parabolic
- 2D shell: quadrilateral and triangular plates, membranes
- Rigid elements

✓ Material Properties

- Isotropic
- Orthotropic
- Nonlinear materials
 - Nonlinear elastic
 - Elasto-plastic
 - Plastic
- Hardening
 - Isotropic
 - Kinematic
 - Combined
- Yield
 - Von Mises
 - Tresca
 - Mohr-Coulomb
 - Drucker-Prager
- Custom stress-strain curve

✓ Material Orientation

- Vector projection
- Curve tangent
- Rotated curve tangent
- Translated curve tangent
- Surface U and V directions

✓ Properties

- Solid and plane
- Different plane property for each face
- Composite laminate with various failure theories:
 - Hill
 - Hoffman
 - Tsai-Wu
 - Max. stress
 - Max. strain
 - NASA LarcO2

✓ Surface Contact

- Automatic surface contact generation
- Free and welded contact types
- Static friction

✓ Coordinate Systems

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

✓ User Interface

- SolidWorks look and feel
- Seamless integration with SolidWorks GUI
- Modern tree view layout

✓ Analysis Types

- Linear Statics
- Normal Modes
- Linear Buckling
- Nonlinear Stress
- Thermal Stress
- Prestress Static
- Prestress Normal Modes
- Composite
- Contact Analysis in Assemblies
- Linear Steady State Heat Transfer

✓ 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

✓ 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

✓ Editor

- Fully integrated and customizable Nastran Editor controls program operation and provides results summary data through an easy to use GUI
- Features tabbed windows to give immediate access to all input and output files
- Field markers make manual editing simple and increase productivity dramatically
- Complete online documentation and context sensitive help
- Permits batch queuing of jobs for sensitivity and configuration trade studies

✓ Advanced Features (available through the Editor)

- Spring mass and damper
- 1D elements: rods, tube, bar, pipe, tapered beams
- Tension-only plate and cable
- Laminated solids (CHEXA, CPENTA)
- Spot weld (CWELD)
- Gap, slideline and surface contact
- Anisotropic and temperature dependent materials
- Initial strain analysis
- Inertial relief analysis
- Tabular results listing
- Detailed HTML report customization
- Single and multi-load set animations
- Interactive data query with mouse
- Parameter setup and control
- View nonlinear analysis while running
- Queue jobs to run consecutively for sensitivity analysis
- Simple optimization studies

✓ 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.

✓ Language Support

- GUI: English, Japanese, Italian, French, others upon request
- Technical documentation: English

NEiWorks V1.12 Basic

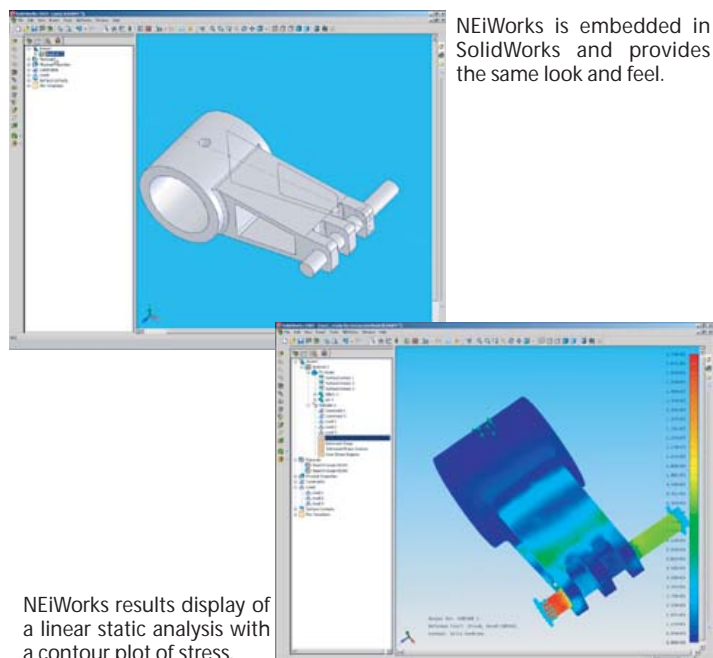
High Performance Embedded Nastran FEA for SolidWorks.

NEiWorks Basic, embedded Nastran FEA for SolidWorks®

NEiWorks Basic is a true Nastran Finite Element Analysis (FEA) Program embedded into the SolidWorks solid modeling program. You can carry out FEA on SolidWorks parts and assemblies from within the SolidWorks environment. NEiWorks Basic is fully integrated, uses the SolidWorks geometry engine, provides the same look and feel as the host SolidWorks program, via an identical menu and tree type structure, and has SolidWorks' Gold Product status. NEiWorks Basic accesses NEiNastran's L1 and L2 Solver modules to provide you with high accuracy solutions to a range of linear static, heat transfer, and modal analyses.

Easy to learn and use, covers the major analyses types

One of the most common types of analysis needed by design engineers is linear static analysis. By far, linear static structural analysis represents the majority of the analyses performed. NEiWorks Basic makes this type of analysis easy to do. Starting with your SolidWorks part, loads and boundary conditions are applied using a series of pull down menus to define the force or pressure on the structure and the direction. Similarly, constraints are defined. The material is then selected from a material library or you can define a material by entering appropriate properties, including orthotropic materials and composites. The part is meshed automatically with provisions for manual control. Results can be displayed in a wide variety of formats from contour plots of stress, strain and displacement, to graphical outputs, tabular data listings, and animations. Context sensitive Help is available to assist you at every step.



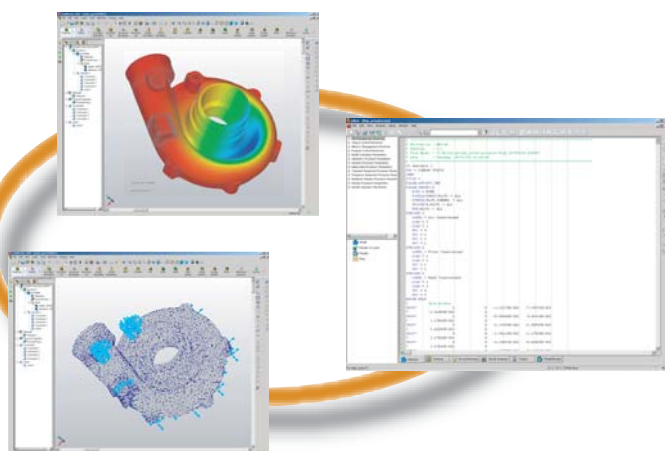
NEiWorks is embedded in SolidWorks and provides the same look and feel.

NEiWorks results display of a linear static analysis with a contour plot of stress.

In addition to structural analysis, NEiWorks Basic lets you perform heat transfer analysis providing temperature and heat flux plots. Modal analysis is also included which is used to reveal vibrations in structures. See the NEiWorks Product Chart for a complete listing of Analysis Types and Post Processing capabilities of NEiWorks Basic and NEiWorks Expert.

Saves Time and Money, Connects You with Customers and Design Partners

NEiWorks Basic lets you graduate from hand calculations, eliminate successive rounds of prototypes and testing, explore design alternatives, and verify your engineering judgments. Plus, a major feature of NEiWorks - you will be able to pass your work on to a large established base of analysts who use NEiNastran, NX Nastran and MSC.Nastran. This can be a key business consideration in allowing you to easily communicate and work with potential customers, suppliers, or design partners.



NEiWorks lets you pass your FEA results to all other Nastran users using common file formats.



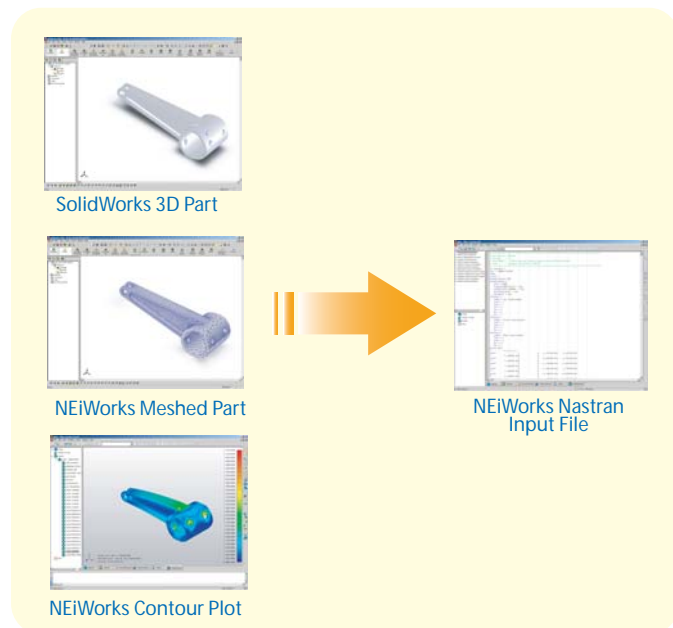
NEiWorks V1.12 Expert

High Performance Embedded Nastran FEA for SolidWorks.

NEiWorks Expert

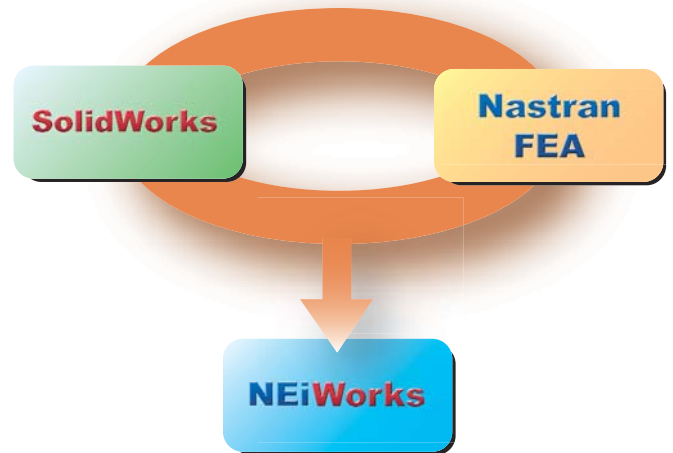
NEiWorks Expert, the power of Nastran FEA Solvers combined with SolidWorks® ease of use

NEiWorks Expert brings high end Nastran FEA to SolidWorks users and to experienced Nastran analysts who are looking for the next generation in innovative pre- and post-processing capabilities. NEiWorks Expert integrates the solid geometry capabilities of SolidWorks with the accuracy, speed, and power of NEiNastran solvers to give users an FEA environment that eliminates time consuming and frustrating associated tasks like geometry cleanup and remeshing for part changes allowing more time for true analysis and simulation work. Plus, a major feature of NEiWorks - you can pass your work on to the large established base of analysts who use NEiNastran, NX Nastran and MSC.Nastran. This can be a key business consideration in allowing you to easily communicate and work with potential customers, suppliers, or design partners.



A full complement of high end analysis types and features

NEiWorks Expert builds on the linear, modal, and steady state heat transfer capabilities of NEiWorks Basic by adding linear dynamics and nonlinear capabilities for stress, strain, buckling, and event simulation. For thermal problems, temperature dependent materials, radiation, convection, and heat power are included. Plus NEiWorks Expert includes Drop Test analysis and Model Reduction capabilities. See the NEiWorks Expert Product Chart for a complete listing of Analysis Types and Post Processing capabilities.



NEiWorks Expert also includes a number of features that will appeal to experienced Nastran FEA analysts. The 3D Surface Contact capabilities greatly simplify this type of commonly needed analysis. The industry unique Editor feature boosts productivity, saves time and frustration in setting up solutions and simplifies problem set-up by providing control of program operation, model editing, trade off study generation, and results analysis. NEiWorks' Cause, Action, Remarks formatted error and warning messages make it easy to diagnose and quickly correct model errors. It is a welcomed change from cryptic and obscure error messages that can make software frustrating and unnecessarily difficult.

Connection with the Nastran community, at an affordable price

NEiWorks Expert will give you entry to the large established base of analysts who use NEiNastran, NX Nastran and MSC.Nastran allowing you to more easily communicate with potential customers, suppliers, or design partners. This is a great way to break down barriers between design and analysis functions, improve communication, shorten design cycles and connect with valuable business partners. Additionally, NEiWorks is highly affordable. You get the most FEA for your budget with special pricing for multiple seat installations, the ability to share solvers in network installations, and the ability to run SolidWorks simultaneously with the NEiNastran solver.



NEiWorks V1.12

High Performance Embedded Nastran FEA for SolidWorks.

	Basic (L1-L2)	Expert (L1-L5)		Basic (L1-L2)	Expert (L1-L5)
Analysis Types			Editor		
Linear static	✓	✓	Tabular results listing	✓	✓
Normal modes	✓	✓	Detailed HTML report customization	✓	✓
Linear buckling	✓	✓	Single and multi-load set animations	✓	✓
Nonlinear stress		✓	Interactive data query with mouse	✓	✓
Thermal stress	✓	✓	Parameter setup and control	✓	✓
Prestress static	✓	✓	Real time control of solution parameters	✓	✓
Composites	✓	✓	Real time 2D xy plotting and 3D deformed shape and contour plotting	✓	✓
Contact analysis in assemblies	✓	✓	Batch job queuing system	✓	✓
Linear steady state heat transfer	✓	✓	Graphical nonlinear convergence form displays nonlinear work, load, and displacement convergences in percent complete bar format		✓
Optimization	+	+	Configuration trade study generator automatically generates queued models with user specified design variable changes such as thickness or dimension for design sensitivity analysis	✓	✓
Loads and Boundary Conditions			Real time deformed shape results contour displays with automatic updating for nonlinear static and transient solutions		✓
Uniform pressure and force on faces, edges and vertices	✓	✓	Real time results x-y plot support at min/max and user specified models locations with automatic updating for nonlinear static and transient solutions		✓
Directional pressure and force	✓	✓	Export x-y plots to MS Excel Comma Separated Variable (.CSV) file format	✓	✓
Acceleration loads (gravity)	✓	✓	3D vertical bar plot support for Modal Assurance Criterion (MAC) and Modal Cross Orthogonality (MXO)	✓	✓
Enforced displacements and rotations	✓	✓	User defined settings can be customized and saved for different solution types	✓	✓
Temperature, default temperature and heat flux	✓	✓	Special input forms for classified DDAM data allows models to run in an unclassified environment		✓
Symmetric, antisymmetric, axisymmetric, boundary conditions	✓	✓	Parabolic shell to linear shell element converter	✓	✓
Fixed constraints on faces, edges and vertices	✓	✓			
Directional and prescribed constraints	✓	✓			
Thermal constraints	✓	✓			
Assembly Connectors					
True surface contact		✓			
Thermal contact resistance	✓	✓			
Element Library					
3D Solid: tetrahedron both linear or parabolic	✓	✓			
2D Shell: quadrilateral and triangular plates	✓	✓			
Rigid elements	✓	✓			
Conduction	✓	✓			
Materials					
Isotropic	✓	✓			
Orthotropic	✓	✓			
Nonlinear materials: nonlinear elastic, elasto-plastic, plastic		✓			
Hardening: isotropic, kinematic, combined		✓			
Yield: Von Mises, Tresca, Mohr-Coulomb, Drucker-Prager		✓			
Custom stress-strain curve		✓			
Material Orientation					
Vector projection	✓	✓			
Curve tangent	✓	✓			
Rotated curve tangent	✓	✓			
Translated curve tangent	✓	✓			
Surface U and V directions	✓	✓			
Customizable material library	✓	✓			
Properties					
Solid and plane	✓	✓			
Different plane property for each face	✓	✓			
Composite laminate with various failure theories: Hill, Hoffman, Tsai-Wu, Max Stress and Strain, NASA LARC02	✓	✓			
Surface Contact					
Automatic mate dependent contact pair generation	✓	✓			
Free and welded contact types		✓			
Static friction		✓			
Coordinate Systems					
Cartesian, cylindrical and spherical coordinate systems	✓	✓			
Referencing global assembly, part or custom coordinate systems	✓	✓			
Display toggles	✓	✓			
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 to other FEA systems	✓	✓			
Output within the NEiWorks view with sensitive Help and Analysis Control	✓	✓			
Import results using FEMAP Binary Neutral file format (FNO)	✓	✓			

+ = Add On

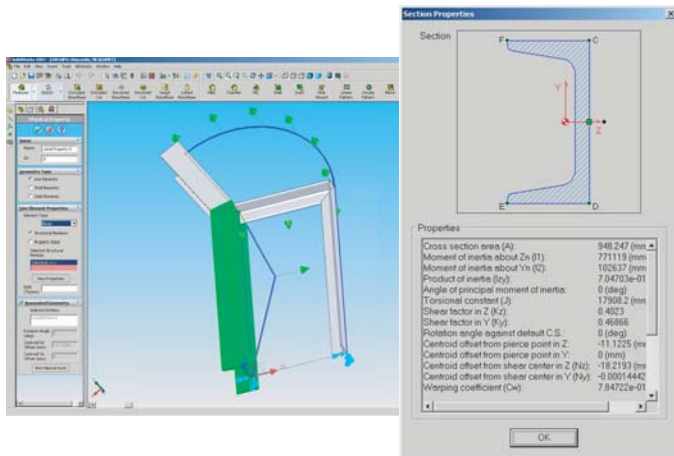
NEiWorks V1.2 New Features

High Performance Embedded Nastran FEA for SolidWorks.

Beam Elements

Beam capability in NEiWorks enables users to simulate and analyze beam models.

- Curve meshing supports lines and arcs including those used for constructing weldments (structural members).
- All joints are merged and a continuous mesh is created for the lines/arcs selected in meshing.
- Physical properties are available for beam and bar elements:
 - Cross sections properties can be defined by either a Structural Member option or a Direct Property Input option.
 - Section properties and cross section shape are available.
- Load and constraints can be created on a point or along a line/arc for beam analysis.
- Deformation and other results at each stress recovery point can be displayed graphically.

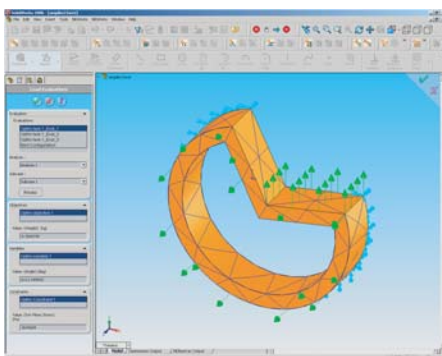


NEiWorks Beam and Cross-Section Display

Optimization Analysis

HEEDS Optimization is integrated within NEiWorks.

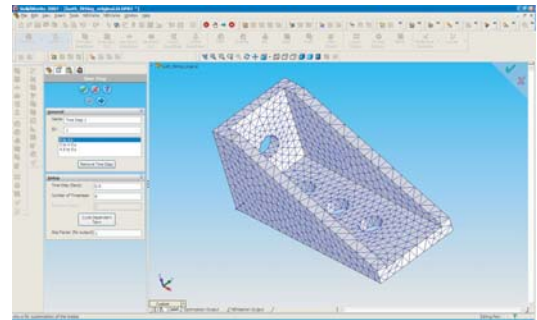
- The user will be able to assign design objectives to minimize, maximize or target the following:
 - Geometric data values
 - Weight
 - Eigenvalues



Dynamic Analysis

Phase 1 for Dynamic Analysis supports:

- Pre-stiffened Normal Modes
- Normal Modes Analysis enhancements
- Modal Transient Response Analysis
- Direct Transient Response Analysis



Thermal Analysis Upgrades

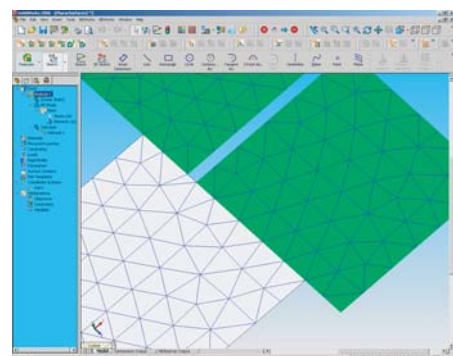
- Body Temperature: The user can apply temperature load to the entire body using this option.
- Initial Temperature: Typically this will be an additional body load. If it is not specified, the solver will just use the reference temperature defined on the material.

Automated Impact Analysis Wizard

Impact analysis such as drop test and projectile impact studies is now fully automated using NEiNastran V9.1 Automated Impact Analysis (AIA) capability. Using a simple input form to define projectile initial velocity and acceleration, the NEiNastran solver will automatically setup surface contact between the projectile and part and calculate the time steps needed to capture an accurate nonlinear transient event simulation.

Meshing Enhancements

- Continuous Meshing across surfaces

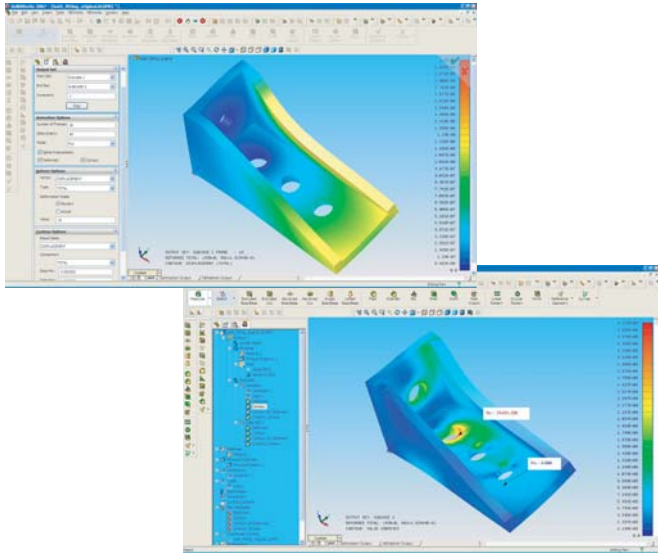


NEiWorks V1.2 New Features

High Performance Embedded Nastran FEA for SolidWorks.

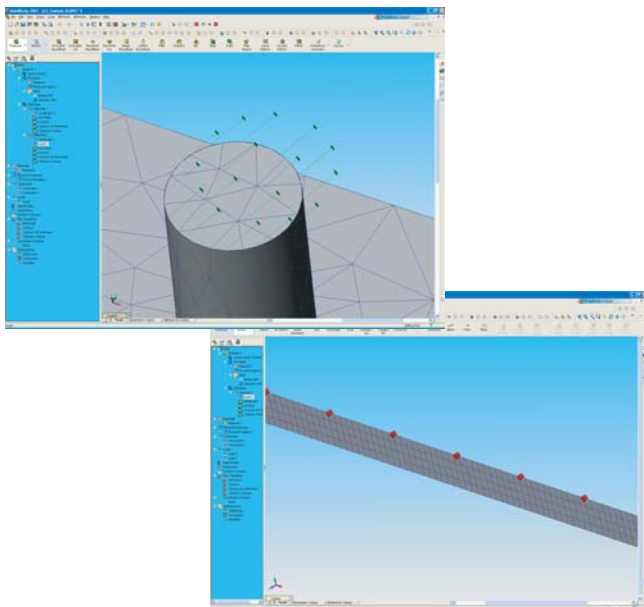
Post-Processing Enhancements

- Single Set Animation
- Multi Set Animation
- Max/Min Labels



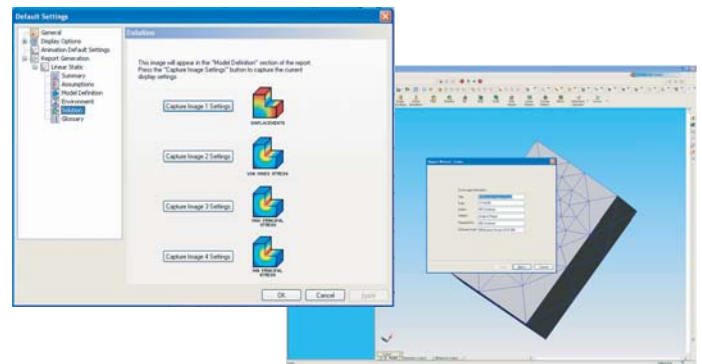
On-screen Display Enhancements and Customizations

- Load symbol display - can change size, color, image based on specific types of load
- Mesh control visualization



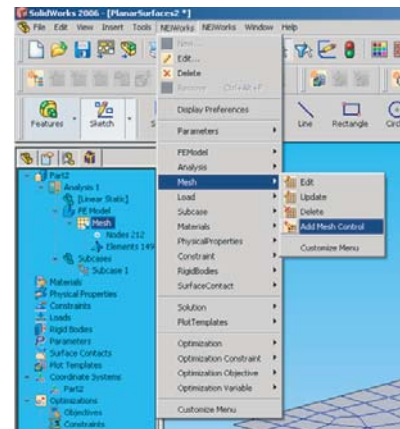
Output Report Writer

- HTML formatted reports can be generated for linear static analysis. Other solutions will soon be available.
- The HTML report contains a detailed summary of the analysis including:
 - Model dimension and mass properties by part and total.
 - Element geometry statistics.
 - Tabular and graphical results data.
- User customizable settings allow you to design your own report format.
- Simple, one button, access launches a wizard to guide you through the report generation process.



Other Features

- Menu support for all features



- Extended toolbar support



- Improved support of SolidWorks features