

NEiFusion

Features

Overview

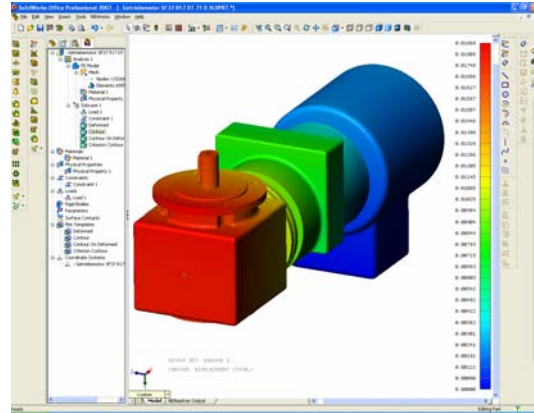
NEiFusion combines a FEM Modeler with comprehensive pre- and post-processing capabilities, and NASTRAN Solvers. Parts and assemblies can be analyzed for a wide spectrum of static, dynamic, and thermal loading. NEiFusion features true geometry associativity, composite elements, custom coordinate systems and nonlinear analyses for plasticity and true surface to surface contact.

With NASTRAN being one of the most widely used solutions, NEiFusion users can now communicate their data to most standard pre- and post-processors through support of the NASTRAN file format. This provides versatility to a product which is already easy to use and backed by the renowned NASTRAN solution.

Capabilities:

General:

- Full single-window integration between solid modeling and analysis
- Full support of Windows® functions such as drag-and-drop, point-and-click, and cut-and-paste
- Dynamic viewing (zoom, pan, rotate, sectioning) by mouse or advanced 3D pointing devices
- Toolbars for fast access to main functionality
- Flexible model coloring and transparency control (parts, assemblies, single or groups of faces, etc.)
- 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
- Copy and paste of features



- Powerful configuration management for easy “what if” design variations (geometrical and physical)
- Comprehensive support for bi-directional CAD data exchange with most major CAD packages
- Comprehensive, context-sensitive HTML-based help system and tutorials
- OpenGL graphics taking advantage of the latest Computer Graphics chips

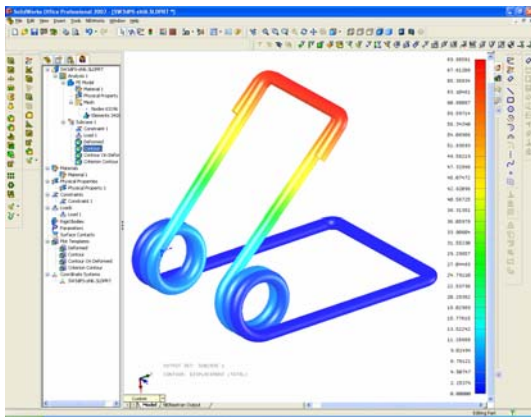
CAD Interoperability:

- 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)
- Support of seamless integrated third-party bi-directional file translators, e. g. for CATIA®, Pro/ENGINEER®, etc.
- 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.)

- In-place editing
- Integrated sketching (dynamic referencing)
- Extrudes, revolves, feature patterns, holes, etc.
- Advanced 3D operations, e.g. lofting, sweeping, complex blending, filleting, etc.
- Advanced shelling, midsurfaces
- Multi-body support
- 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
- Multiple sub-assembly support
- Design-in-the-context of an assembly (references to other geometry, associative relationships, direct/indirect constraints)

- 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

Meshing:

- Global and local controls for part geometry with default 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 with geometry changes
- Mesher Status Window

Assembly Connectors:

- True surface contact
- Automatic contact
- Thermal contact resistance

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
- Thermal body loads
- Initial temperature conditions
- Custom colors and sizes for loads and constraints

Element Library:

- 1D line (CBEAM and CBAR)
- 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
- 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

Properties:

- 1D beam (PBEAM) and bar (PBAR)
- 2D plate (PSHELL) and composite (PCOMP)
- 3D solid (PSOLID)
- Contact (BSCONP)

Surface Contact:

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

Analysis Types:

- Linear statics
- Normal modes
- Linear buckling
- Nonlinear stress
- Thermal stress
- Prestress static
- Composite
- Contact analysis in assemblies
- Linear steady state heat transfer
- Optimization
- Modal Transient Response

- Direct Transient Response

Composite Analysis:

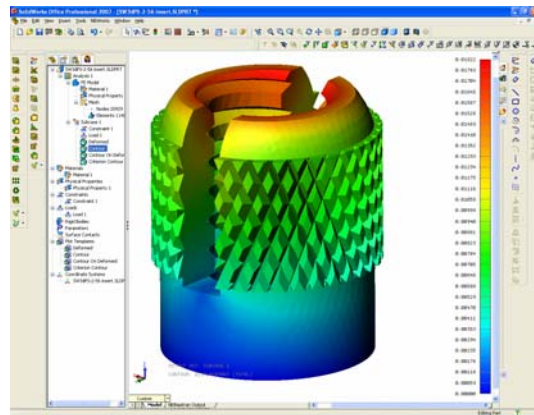
- Various failure theories supported:
 - Hill
 - Hoffman
 - Tsai-Wu
 - Max. stress
 - Max. strain
 - NASA LARC02

Optimization Analysis:

- Design objectives to minimize, maximize or reach target values
- Optimize weight, stress, temperature and natural frequency
- Parametrically update geometry dimensions

Drop Testing Analysis:

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



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 Nastran input deck to other FEA systems
- Customizable material library
- Output within NEiFusion Modeler view with sensitive Help and analysis control, such as pausing and solution termination
- Import results using FEMAP Binary Neutral file format (FNO)
- Single and multi-set animations
- Max/min labels

Report Generation:

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

Direct Matrix Input Grid (DMIG) Support (Analyst version only):

- Stiffness matrix import and export
- Conductivity matrix import and export
- Mass matrix import and export
- Load vector import and export

Global Matrix Output (Analyst version only):

- Stiffness matrix
- Conductivity matrix
- Mass matrix

Model Reduction (Analyst version only):

- Static condensation
- Export reduced stiffness matrix using DMIG format
- Automated model reduction tools and correction
- Export reduced mass, stiffness, damping, and load matrices to DMIG or NASTRAN Output2 (.OP2)
- Craig-Bampton reduction (component modes synthesis)

Modal Correlation (Analyst version only):

- Modal assurance criterion (MAC) output and 3D plots
- Model cross-orthogonality output and 3D plots
- MS Excel comma separated variable (.CSV) and NEiNastran modal database (.MDB) input formats

- Automatic interpolation of input data to closest grid

Editor (Analyst version only):

- 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
- Tabular results listing
- Detailed HTML report customization
- Single and multi-load set animations
- Interactive data query with mouse
- Parameter setup and control
- Real time control of solution parameters
- Real time 2D x-y plotting and 3D deformed shape and contour plotting
- Batch job queuing system
- Graphical nonlinear convergence form displays nonlinear work, load, and displacement convergences in percent complete bar format
- Configuration trade study generator automatically generates queued models with user specified design variable changes such as thickness or dimension for design sensitivity analysis
- Real time deformed shape results contour displays with automatic updating for nonlinear static and transient solutions
- Real time results x-y plot support at min/max and user specified models locations with automatic updating for nonlinear static and transient solutions
- Export x-y plots to MS Excel Comma Separated Variable (.CSV) file format
- 3D vertical bar plot support for Modal Assurance Criterion (MAC) and Modal Cross Orthogonality (MXO)
- User defined settings can be customized and saved for different solution types
- Special input forms for classified DDAM data allows models to run in an unclassified environment
- Parabolic shell to linear shell element converter

Compatibilities:

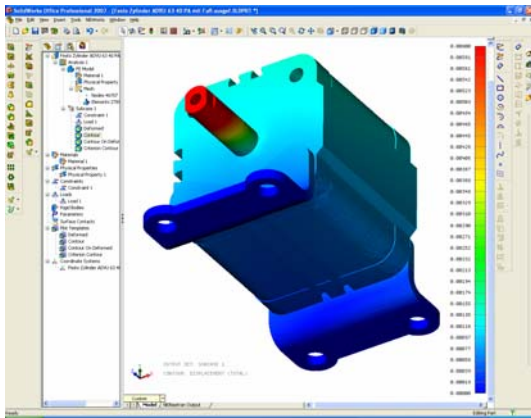
- Nastran input file can be sent to any Nastran FE Solver including NEiNastran, NX Nastran, or MSC.Nastran
- Binary results file in OP2 format usable by all Nastran solvers and wide variety of post-processors
- Part and Assembly geometry is fully compatible with SolidWorks' Parts and Assemblies

User Interface:

- Menu support for all features
- Toolbar shortcuts
- Modern tree view layout

International Languages:

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



System Requirements:

- Intel Pentium® 4 or AMD Athlon™ based PC
- 512 MB RAM minimum, more recommended
- 1.5 GB free hard disk space for installation, more required for simulation models
- Microsoft Windows XP® Professional, Windows XP x64 or Windows 2000®

Noran Engineering, Inc is committed to the success of our customers. Detailed documentation, customized on-site training, and comprehensive technical support ensures that you will see immediate return on your investment.

For more information about our company or our products, please contact:

Headquarters:

Noran Engineering, Inc

5555 Garden Grove Blvd., Suite 300
Westminster, CA 92683-1886
USA

Phone: 1.714.899.1220

Fax: 1.714.899.1369

Email: info@noraneng.com

Website: www.NENastran.com

Europe:

SmartCAE

Piazza della Gualchierina, 9
59100 Prato
ITALY

Phone: +39.0.574.404.642

Fax: +39.0.574.401.265

E-mail: info@smartcae.com

Website: www.smartcae.com

Asia/Pacific:

Digital Solutions

Kyoei Nakasuji Bldg., 3-7-18
Nakasuji, Asaminami-ku
Hiroshima 731-0122
JAPAN

Phone: +81.82.831.1190

Fax: +81.82.831.1193

E-mail: post@digital-sol.co.jp

Website: www.digital-sol.co.jp



NEiFusion

From Noran Engineering, Inc.