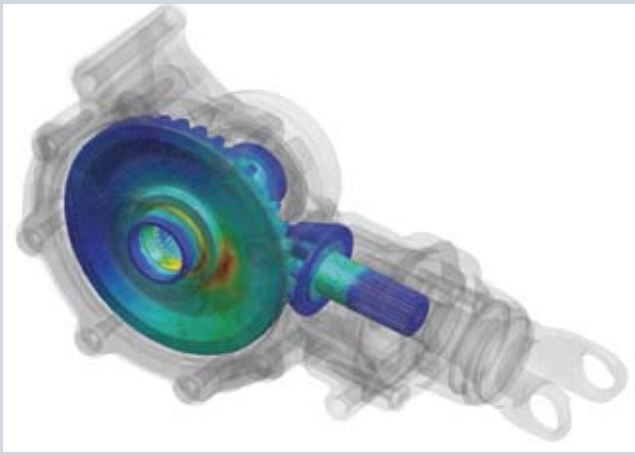
NX Advanced FEM

Benefits

- Build models faster with embedded tools for 3D geometry creation, editing and abstraction
- Make design changes easily with synchronous technology for quick what-if analysis
- Enable faster collaboration between analysts and design engineers with geometry associativity
- Provide access to design change knowledge
- Provide "on-demand" FE model updates based on design geometry changes
- Manage and share your CAE data with support for NX Manager and Teamcenter software for all created FE data sets
- Reduce modeling errors and model build time by providing your analysts with a familiar environment – NX environments are customized for the chosen solver
- Build models faster with a full range of tools for FE model generation including predefined constraint conditions and automated mesh mating conditions

Summary

NX™ Advanced FEM software is a comprehensive multi-CAD finite element modeling and results visualization solution that is designed to meet the needs of experienced CAE analysts. It includes a full suite of geometry creation and editing tools as well as FE pre- and post-processing tools that supports a broad range of product performance evaluation solutions. NX Advanced FEM provides 2-way association to design geometry, allowing users to rapidly iterate on design changes. Robust CAD translators, along with the embedded industry-standard Parasolid® 3D modeling kernel, enable non-native geometry to be easily imported for use within the NX Advanced FEM environment.



Basic functionality

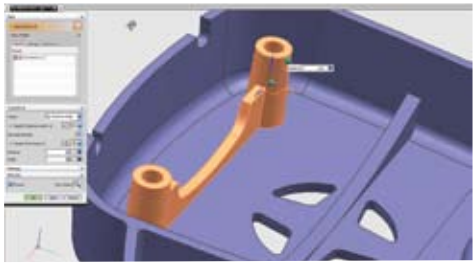
NX Advanced FEM provides fundamental modeling functionality for automatic and manual mesh generation, application of loads and boundary conditions and model development and checking. NX Advanced FEM includes Assembly FEM technology, a distributed model approach to handle large FEM assemblies. A robust set of visualization tools generates displays quickly, lets you view multiple results simultaneously and enables you to easily print the display. In addition, extensive post-processing functions enable you to review and export analysis results to spreadsheets and leverage extensive graphing tools to better understand your results. Post-processing also supports the export of JT™ data for collaboration across the enterprise with JT2Go and Teamcenter® for lifecycle visualization software.

NX Advanced FEM provides seamless, transparent support for a number of industry-standard solvers, such as NX Nastran®, MSC Nastran, Ansys, Abaqus and LS-Dyna. For example, when you create either a mesh or a solution in NX Advanced FEM, you specify the solver environment that you plan to use to solve your model and the type of analysis you want to perform. The software

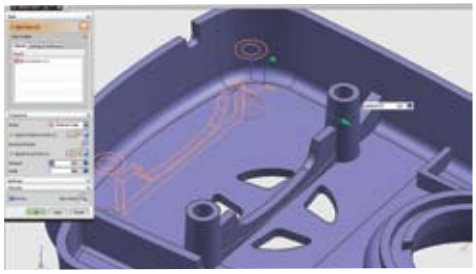
NX Advanced FEM

Benefits *continued*

- Reduce rework by verifying models before processing with a full set of graphical and mathematical tools that help check model suitability
- View analysis results quickly and easily with a dynamic visualization tool
- Gain new insights on design performance with extensive postprocessing tools to continue the iterative phases of analysis or to export/import information

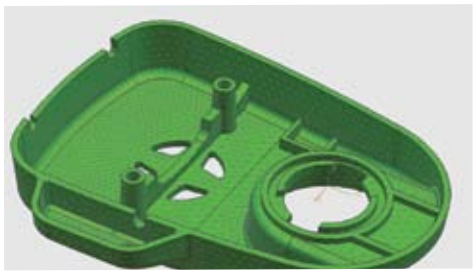


3. Selecting geometry that you want to move.



4. Dragging geometry to new position.

5. New design ready for analysis.

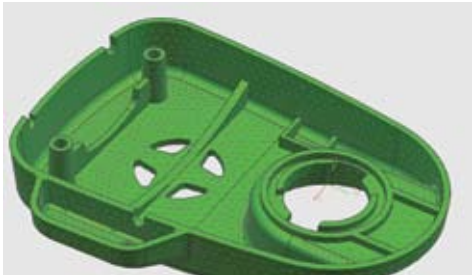


6. Updated analysis model ready for simulation.

then presents all meshing, boundary conditions and solution options using the terminology or "language" of that solver and analysis type. Additionally, you can solve your model and view your results directly in Advanced FEM without having to first export a solver file or import your results.

- Advanced FEM features data structures, such as the separate Simulation (.sim) and FEM (.fem) that help facilitate the development of FE models across a distributed work environment. These data structures also allow analysts to easily share FE data to perform multiple types of analyses.
- The NX data model is designed to be fast and handle large amounts of data. Duplication of data is avoided and data is placed where it belongs – either on the mesh or on the elements.
- NX automatically organizes your model in a logical manner. Meshes are placed in mesh collectors, and your model entities can also be placed in groups. Visibility of these entities can be easily managed in the Navigator.
- NX provides a distributed model approach for analyzing assemblies whereby the Assembly FEM model does not contain the component FEM models, but instead holds pointers to these models. An assembly FEM contains occurrence and position data for multiple component FEMs as well as connection elements that join the component FEMs into a system. The assembly FEM can also contain material and physical property overrides on component FEM meshes.
- Advanced FEM offers world-class meshing capabilities. The software is designed to produce a very high quality mesh while using an economic element count. Advanced FEM supports a complete complement of element types (0D, 1D, 2D and 3D). Additionally, Advanced FEM gives analysts control over specific meshing tolerances that control, for example, how the software meshes complex geometry, such as fillets.

1. Plastic case initial geometry.



2. Plastic case initial analysis model.

- Advanced FEM includes multiple geometry abstraction tools that enable analysts to tailor the CAD geometry to the needs of their analysis. For example, analysts can use these tools to improve the overall quality of their mesh by eliminating problematic geometry, such as tiny edges or slivers.
- Advanced FEM also supports the new NX Thermal and NX Flow solutions
- NX Thermal is a fully integrated finite difference solver. It allows thermal engineers to predict heat flow and temperatures in systems subjected to thermal loads.
- NX Flow is a computational fluid dynamics (CFD) solver. It allows analysts to perform steady-state, incompressible flow analysis and predict flow rates and pressure gradients for movement of fluid in a system.

When used in combination, NX Thermal and NX Flow provide fully coupled treatment of convective heat transfer, enabling robust simulation of conjugate heat transfer problems.

Features

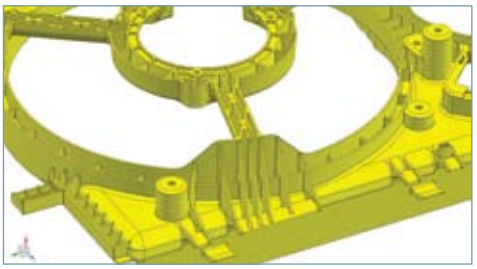
- Unique and powerful geometry manipulation and abstraction tools
- Fully associative CAD modeling tool set for feature removal and idealization
- Automatic CAE abstraction tools based on desired element size that further refine the geometry to ensure high quality mesh generation with limited user interaction
- CAD software interface support for Parasolid software, JT, STEP and IGES data formats
- Intuitive interface with the Simulation navigator for quick and easy access to FE data entities
- Complete material definitions support for isotropic, anisotropic, orthotropic and hyperelastic materials
- Material catalog with 60 predefined materials
- Complete set of load and constraint definitions to support a wide variety of solutions
- Support for NX Nastran desktop and enterprise solvers
- Support for additional CAE solutions for fluid flow analysis, advanced thermal analysis, advanced durability analysis and design optimization
- Support for a variety of external third-party solvers through a unique environment selection based on PLM XML that enables customization of the UI to match the selected solver for element definition through solution setup
- Integrated design parameter-based optimization

FE modeling tools

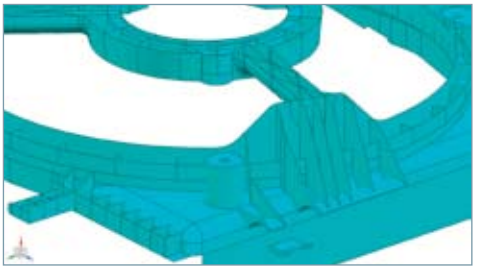
Wire-frame, surface and solid geometry from other CAD systems can be accessed through embedded standards-based interfaces (IGES, STEP AP203, STEP AP214, Parasolid and JT) or optional direct CAD interfaces for Catia and Pro/Engineer.
A complete set of geometry creation and modification tools is provided to work directly with native and non-native geometry. Often design geometry must be modified to build an effective model. Details may need to be suppressed or eliminated; additional geometry may be required to control mesh density, or surfaces not present in the geometry may be needed for meshing. NX Advanced FEM is powered by synchronous technology, which enables users to modify geometry by easily moving or deleting individual faces or features such as bosses or ribs. Synchronous technology empowers analysts to make simple changes to the idealized geometry to support what-if analyses thereby speeding up design-analysis iterations. Furthermore, this technology works with native and imported geometry, both with or without history.

NX Advanced FEM provides extensive model editing capabilities, including the ability to:

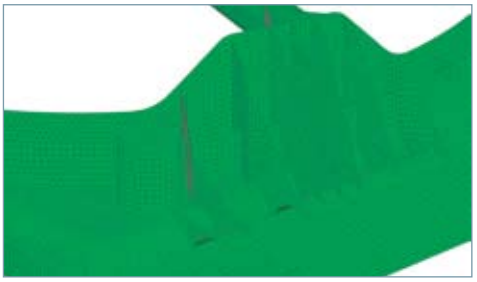
- Interactively suppress features defined within the NX part
- Perform sensitivity analysis using design parameters as defined in the CAD model
- Remove fillets and holes automatically using the idealize command set on both native and non-native geometry
- Add, modify or delete entities (sheet body, solid body)
- Extract the mid-surface representation directly from the solid body for modeling thin walled components; surface thickness is mapped from the solid to the 2D representation
- Facilitate automated switching, which reduces manual interaction when surface geometry is switched together for thin-walled components
- Leverage powerful commands to partition and split solid bodies that also predict if a partition can be easily meshed
- Automatically support the relationships between the component CAD model and the FE model within NX Manager and Teamcenter



1. Solid geometry of thin-walled part.



2. Mid-surface generated from solid geometry.



3. Shell mesh on mid-surface geometry.

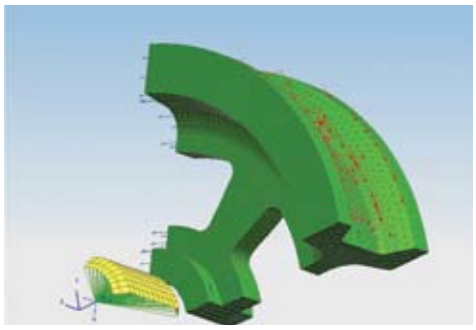
Abstracting CAD for FE meshing

Often, CAD topology contains details that are of no use to the analyst. Sliver surfaces, detailed embosses (e.g., “Made in U.S.A.”), small fillet radii and small holes are examples of details the analyst may not wish to mesh. There are multiple tool sets to abstract and idealize the geometry for model preparation.

Idealize commands are provided to allow the user to remove and suppress design features like holes and fillets. This can be done on NX native or non-native 3D CAD geometry. This set of powerful tools allows the user to change the design geometry without the need to own the original geometry.

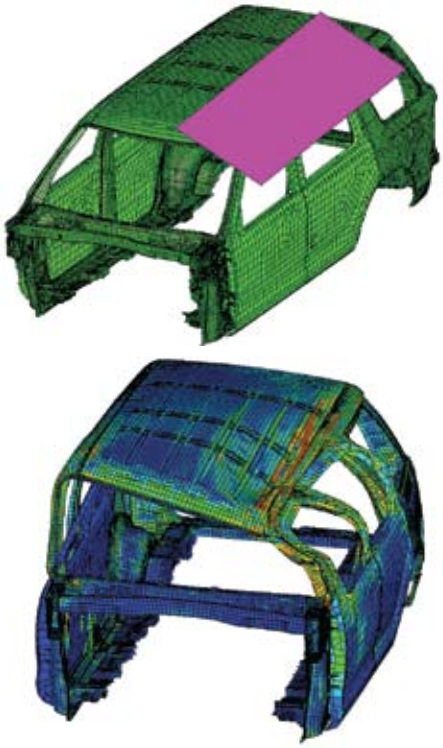
Loads and boundary conditions
NX Advanced FEM provides extensive capabilities to define loading and boundary conditions to correctly simulate operating environments:

- Loads can be defined on and associated with geometry. The creation of the load will ensure a node is placed at the location during automatic meshing.
  - Mesh point
  - Face
  - Edge
  - Curve
- Restraints defined on and associated with geometry
  - Mesh point
  - Face
  - Edge
  - Curve
- Support for surface-to-surface contact definitions
  - Surface-surface glue contact
  - Automatic face pairing
  - Coupling
  - Automatic coupled DOF
  - Manual coupled DOF
- Constraints and restraints, including nodal displacement
- Structural loads
  - Nodal forces and temperatures
  - Element face and edge pressures
  - Acceleration (gravity, translation, rotation)
  - Ambient and reference temperatures
- Heat transfer loads
  - Nodal and distributed heat sources
  - Face and edge fluxes, convection and radiation
- All loads and restraints displayed with unique graphical symbols
- Associativity of geometry-based loads and restraints that is maintained through design geometry changes
- Definition of time-varying loading and boundary conditions to correctly simulate nonlinear loading conditions
- A powerful "field" utility, which can be used to define loads by expression, array, or table input. Spatial-varying loads can be quickly and easily defined via expression or by ascii file import. The same is true for time, frequency, and temperature dependent loads.



Complete model checking tools
Analyzing a model with errors can be time-consuming and expensive, and errors are often not detected even after analysis. NX Advanced FEM provides a full set of graphical and mathematical tools to help verify that a model is complete and correct before you submit it for solution.

- Coincident node and element checks eliminate duplications.
- Free-edge and face checks avoid unwanted cracks in a model.
- Shrink element display verifies that elements are located properly.
- Element shape checks (distortion, warping, etc.) verify that elements do not violate limits and can produce accurate results.
- Element thickness display checks that shell thickness is correct.



Getting the best results from analysis
For the mixed science and art of analysis to impact design decisions, results must be presented in an understandable form. NX Advanced FEM provides extensive graphics and manipulation capabilities that focus on critical data and present it for review and action. A comprehensive and flexible methodology has been adopted to enable the user to act before, during and after FEM solutions are sought.

Using NX Advanced FEM, you can perform a wide variety of functions.
You can create:

- Animated, stepped or smooth-shaded displays
- Cutting plane, contour, element and arrow displays
- Templates of display options for repeated use
- Groups of elements can be created using search criteria e.g., top 30 stress values

You can control:

- How data is displayed (for example, data components and coordinate system to use averaged and unaveraged data)
- Text, headers and colors

You can view:

- Multiple results simultaneously
- Results in multiple viewports
- Deformed geometry

You can display and animate:

- Streamlines, ribbons and bubbles for flow results

You can reduce the display to only show:

- A group of elements

You can insert:

- 3D probe results annotations
- 3D and 2D text annotations

You can export:

- Displays for report-ready printing/ plotting
  - JT
  - JPEG
  - BMP
  - PNG
  - GIF
  - Animated GIF
  - TIFF
- Single or multiple result sets to spreadsheets or directly to Excel (Windows only) for further manipulation

You can generate:
- HTML customized reports for model data and results inspection

You can import:
- Nastran OP2
- Abaqus FIL
- Ansys RST
- I-deas® UNV and BUN software

Optional solver environments
The NX Advanced FEM user layout is driven by the solution environment or language chosen by the user. This environment is customized to immerse the user in the language of the selected solver. The solver language controls all element and loading terminology, as well as all appropriate loads and contestants that are available to the solver.

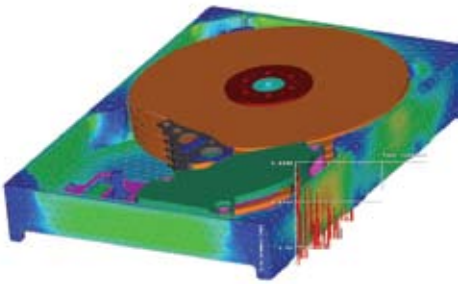
All environments are available to the user but import and export of FEA data is controlled by an optional solution environment add-on module for solvers such as Abaqus, Ansys, Nastran, LS-Dyna, etc. The Nastran Solution Monitor helps the user monitor the solver job status and track errors with the solve.

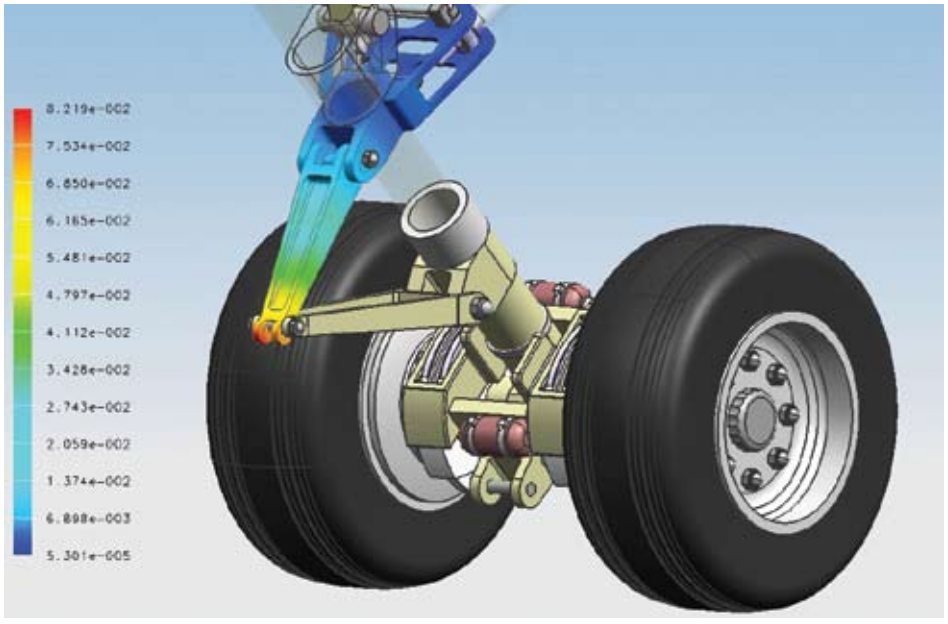
Solution processes
NX Advanced FEM also supports the definition of solution processes, which represent a multi-step solution and work with any of the supported structural solvers. The following processes are supported in Advanced FEM: adaptive, durability and optimization.

Adaptive solution
Adaptive analysis is a linear statics solution option, available for all supported solvers, that uses a solver-independent h-adaptive analysis method to perform automatic mesh refinement during solve iterations. The main objective of adaptive analysis is to automate the lengthy and repetitive process of running multiple finite element analyses with different mesh densities. The mesh refinement is determined by error estimates associated with element stress discontinuity. Based on these error estimates, critical regions are identified on the model, which then become the target for further refinement of the mesh.

Once the critical regions are identified, the local refinement is implemented internally. An element sizing scheme is used to estimate the new size. Furthermore, element quality is checked along transition areas to eliminate false hot spots that may later influence the refinement process.

Adaptive analysis is supported for all 2D and 3D triangular elements. Hex, wedge and pyramid elements are not supported. During the adaptive solve, the refinement of existing mesh elements involves replacement by matching template patterns. Element nodes needing refinement are identified by the software and modified accordingly.





The right-hand table indicates the solution environments currently supported by NX Advanced FEM.

Optimization
Optimization is a process that helps the analyst arrive at the best solution for a given design goal. NX Advanced FEM allows the user to create an optimization solution process. The user can define a goal such as the mass of a part or component, a constraint such as maximum allowable Von Mises stress and the design parameter(s) to vary on the component. The optimization solution process will run based on the design criteria while varying the design parameters to enable the design engineer to determine if there is a better structural design alternative vs. the original baseline design.

Product availability
NX Advanced FEM is a core package in the suite of NX Advanced Simulation add-on applications available within the Siemens Lifecycle Simulation architecture.
It is a prerequisite for all other solution applications in the NX Advanced Simulation suite such as NX Nastran Desktop, NX Durability Wizard, NX Advanced Durability, NX Flow, NX Thermal, NX Electronic Systems Cooling, NX Space Systems Thermal, NX Response Simulation, NX Laminate Composites and the customized solver environments for Nastran, Ansys, Abaqus and LS-Dyna.

NX Advanced FEM is available on 32- and 64-bit Windows and also on 64-bit Linux systems.aliqua.

Solver	Analysis type	Solution type
Nastran	Structural	Linear statics (SOL 101) with surface-to-surface contact Normal modes (SOL103) Response simulation (SOL 103) Buckling (SOL 105) Nonlinear statics (SOL 106) Direct complex eigenvalue (SOL 107) Direct frequency response (SOL 108) Direct transient response (SOL 109) Modal complex eigenvalue (SOL 110) Modal frequency response (SOL 111) Modal transient response (SOL 112) Design optimization (SOL 200) Advanced nonlinear (SOL 601) Advanced nonlinear explicit (SOL 701) Linear and advanced nonlinear transient response (SOL 129)
	Thermal	Heat transfer (SOL 153)
	Axisymmetric structural	Linear statics (SOL 101) Nonlinear statics (SOL 106)
	Axisymmetric thermal	Heat transfer (SOL 153)
Abaqus	Structural Thermal Axisymmetric structural Axisymmetric thermal	General analysis Heat transfer General analysis Heat transfer
Ansys	Structural	Linear statics Modal Buckling Nonlinear statics
	Thermal	Thermal
	Axisymmetric structural	Linear statics Nonlinear statics
	Axisymmetric thermal	Thermal
NX Thermal and NX Advanced Thermal	Thermal	Finite difference thermal (based on TMG)
NX Flow	Fluid flow	Incompressible computational fluid dynamics (CFD)
	Coupled thermal/flow	Coupled
NX Advanced Flow	Fluid flow	Compressible computational fluid dynamics (CFD)

Contact
Siemens PLM Software
Americas 800 498 5351
Europe 44 (0) 1276 702000
Asia-Pacific 852 2230 3333

www.siemens.com/nx

X8 7453 8/10 A