NX Advanced FEM environment for Nastran solver

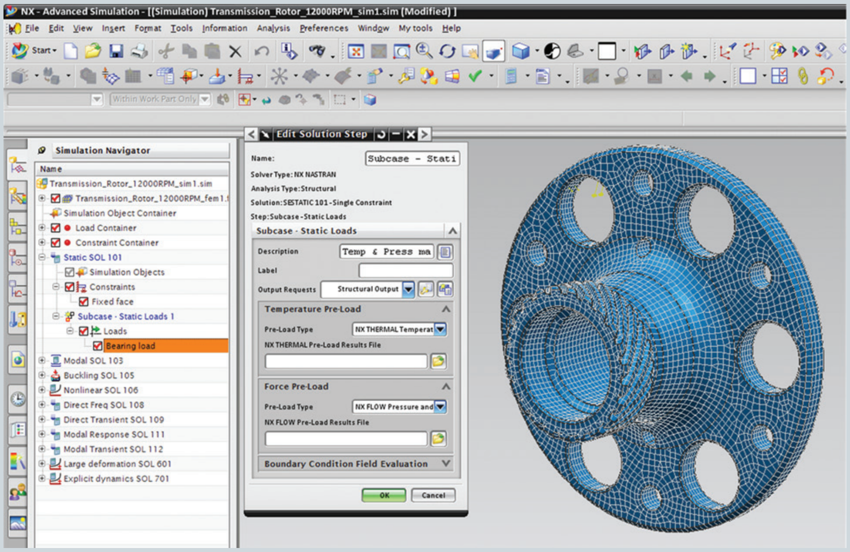
NX CAE

Benefits

- Enables engineers using NX Advanced FEM to generate finite element models for NX Nastran or MSC Nastran solvers
- Simplifies the Nastran modeling process by enabling engineers to create analysis models based on geometry or legacy Nastran data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from NX Advanced FEM
- Immerses engineers in the Nastran environment through familiar Nastran terminology and extensive support of Nastran-specific elements and entities

Summary

The Nastran® environment for NX™ Advanced FEM software enables engineers to build finite element models, define solution parameters and view the solution results for either the NX Nastran or MSC Nastran solvers. The environment immerses engineers with familiar Nastran language for element definitions, loads and boundary conditions, solution parameters and other common Nastran nomenclature. In addition to model definition capabilities, the Nastran environment provides bi-directional import/export capabilities that enable you to import current or legacy Nastran bulk-data files and results as well as export run-ready Nastran data files.



The power of NX Advanced FEM preprocessing and postprocessing is an ideal partner for Nastran models and solutions. NX Advanced FEM simplifies the modeling process by integrating high-end analyst modeling tools with world-class geometry capabilities that assist you with developing analysis models faster than with traditional CAE preprocessors. Adding the Nastran environment to NX Advanced FEM enables you to build Nastran run-ready

NX

www.siemens.com/nx

SIEMENS

NX Advanced FEM environment for Nastran solver

bulk data decks, so little or no intermediate processing is ever needed. In addition to building Nastran models, the Nastran environment imports solution results directly from Nastran binary results files into NX for postprocessing. The environment delivers import/export capabilities so you can import Nastran data decks into NX for modification and then export run-ready decks for solution.

Import/Export Nastran models

- Import/export complete Nastran finite element models including bulk data as well as executive and case controls
- Import model information from either bulk data decks or binary output2 files

Create Nastran models in NX

- Create complete run-ready Nastran decks including executive and case controls, bulk data
- The Nastran environment supports solutions 101, 103, 105, 106, 107, 108, 109, 110, 111, 112, 129, 153, 200, 601/106, 601/129 and 701

Elements and other entities

A wide variety of elements and other model entities are supported.

- Lumped mass, spring, rigid elements
- Axisymmetric solid elements
- Rod, beam and bar elements
- Plane stress and plane strain elements
- Solid elements
- Permanent single-point constraints

A complete list of Nastran import/export entity support is provided in the NX online help documentation under the following header: Advanced Simulation/Solving the Model/Importing and Exporting Model Data/.

Loads and boundary conditions

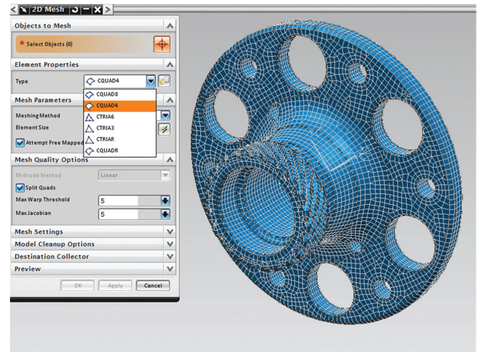
Loads and boundary conditions for structural and thermal analysis are supported.

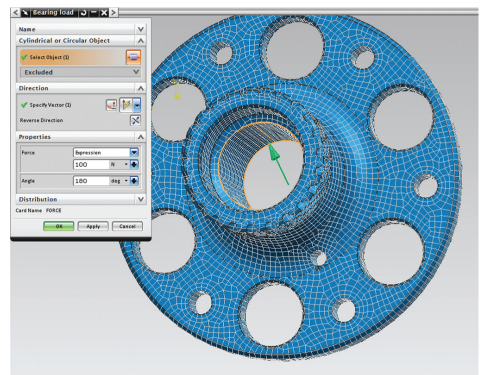
- Nodal, elemental and geometry-based structural loads
- Beam-concentrated and distributed loads
- Gravity, rotational velocities and acceleration loads
- Nodal, elemental and geometry-based thermal heat loads
- Nodal restraints and temperatures
- Traction loads
- Contact regions and sets
- Time and temperature variations
- Subcase manager to easily manage the loads used in each subcase

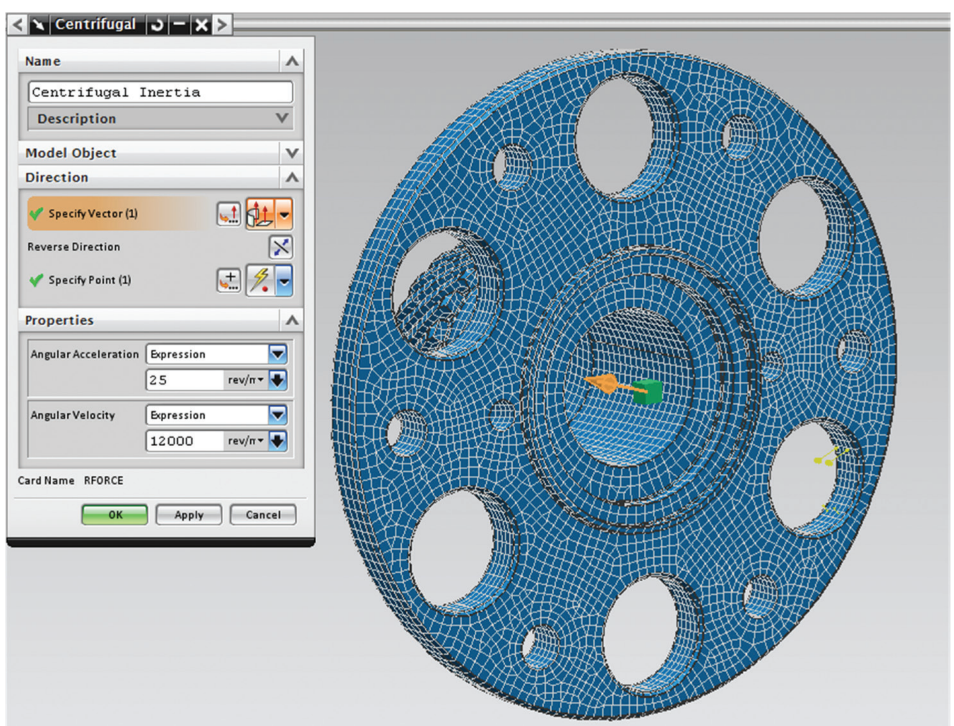
Automatic connection mesh support

NX Advanced FEM provides a number of solver-supported methods for connecting different meshes together.

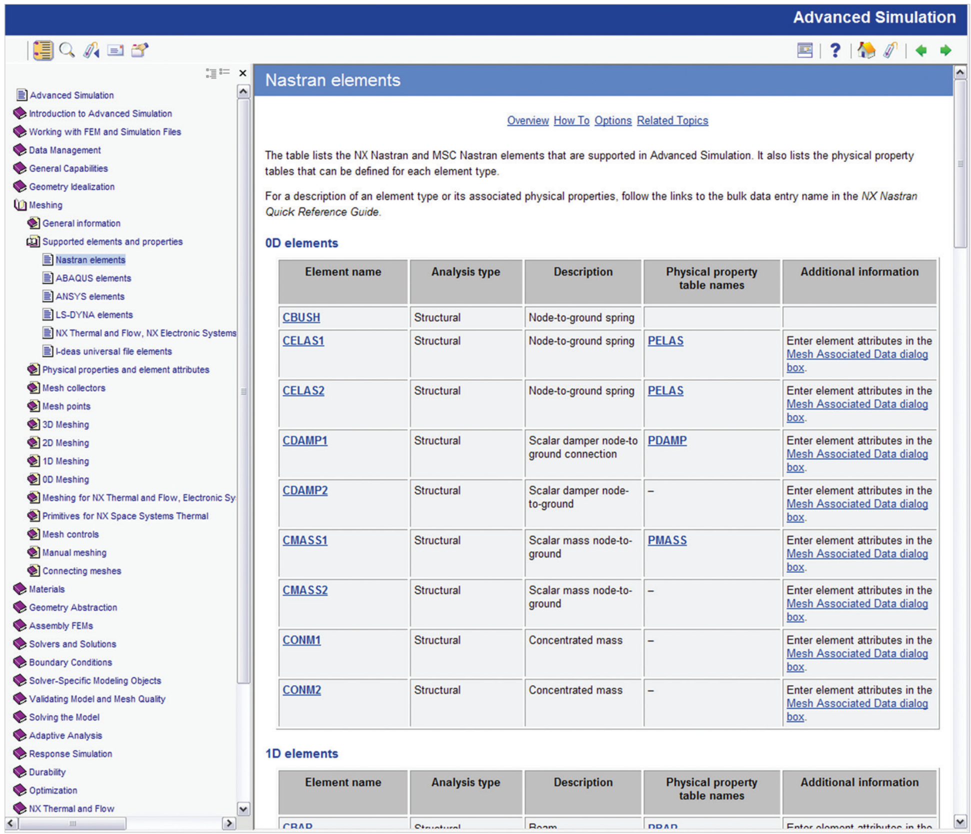
- The mesh mating condition connects individual 2D or 3D meshes together at a specified interface.
- The edge-face connection defines the connection between a set of edges and a set of faces. You can use this feature whenever there are meshes to be connected in T-junction configuration; for example, fins or stiffeners attached to surfaces.







- Weld mesh locates and automates the recognition of weld features (connections) and then automatically creates their FE model representation, including consideration for mid-surfaces. You can use weld mesh to create weld elements (1D mesh) from weld features.
- Contact mesh creates point-to-point contact between two edges or a portion of two edges defined by limiting points.
- Surface contact mesh creates and defines contact elements between two selected faces of a solid or between different components.



Compatibility

The Nastran environment is compatible with the following Nastran releases:

- NX Nastran v8.0 or earlier
- MSC Nastran v2011.1 or earlier

Supported hardware/OS

The Nastran environment is an add-on module within the NX Advanced Simulation suite. It requires a license of NX Advanced FEM as a prerequisite. It is available on all NX supported hardware/OS platforms (Windows and Linux) including selected 64-bit platforms.

Contact
Siemens Industry Software
Americas +1 800 498 5351
Europe +44 (0) 1276 702000
Asia-Pacific +852 2230 3333

www.siemens.com/nx