

NX and the Nastran environment

fact sheet

Siemens PLM Software

www.siemens.com/plm

► Summary

The Nastran environment provides the tools needed to build models, define Nastran solution parameters and view the solution results. The environment speaks the Nastran language in terms of element definitions, loads and boundary conditions nomenclature, etc. It allows the user to fully leverage NX® Advanced FEM software's finite element pre- and post-processing software capabilities to prepare for Nastran analysis. The Nastran environment also enables bi-directional NX import/export capabilities from/to Nastran run-ready input file data.

Benefits

Leverage the NX pre/post simulation toolset with the CAD-associativity and NX collaboration features to generate Nastran models

Use the NX geometry-based and/or FE-based modeling tools simplify the modeling process

Run-ready decks reduce, or eliminate, intermediate processing requirements

The Nastran environment speaks the Nastran terminology and supports a wide variety of elements and other Nastran-specific entities

Features

A broad range of Nastran entities and solutions are supported

Creating complete finite element models including boundary conditions, applied loads, and solution control for Nastran input files

Exporting ready-to-run Nastran models for structural and thermal analyses

Importing solution results directly from solver binary output op2 files for NX post-processing

Environment capabilities

- A broad range of Nastran entities and solutions are supported
- Advanced formatting options for tailoring the model and viewing the results

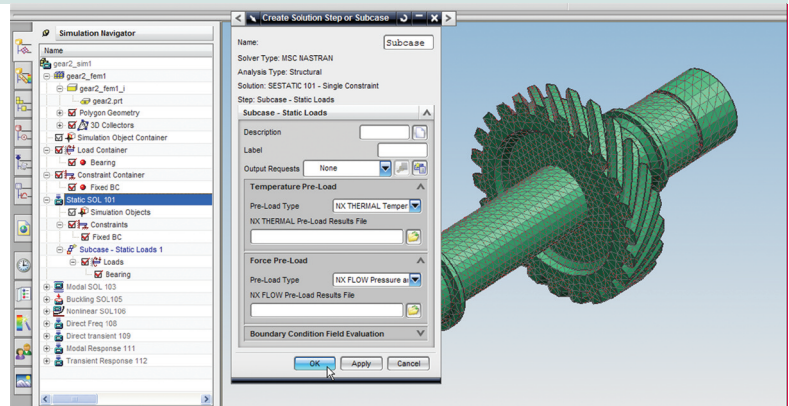
Importing Nastran models

- Complete Nastran finite element models including bulk data as well as executive and case controls
- Model information from either bulk data decks or binary output2 files
- Cross-platform reading of output2 files

Creating Nastran models

- Complete run-ready Nastran decks including executive and case controls, bulk data
- Supports solutions 101, 103, 105, 106, 108, 109, 111, 112, 153, 601/106 and 601/129

The power of NX Advanced FEM pre- and post-processing is an ideal partner with Nastran solution capabilities. NX Advanced FEM geometry-based finite element modeling tools simplify the modeling process. The Nastran environment builds Nastran run-ready bulk data decks, so little or no intermediate processing is needed. Solution results are accessed directly from Nastran binary results files. It is also possible to manually modify your input file prior to solution.



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 /.

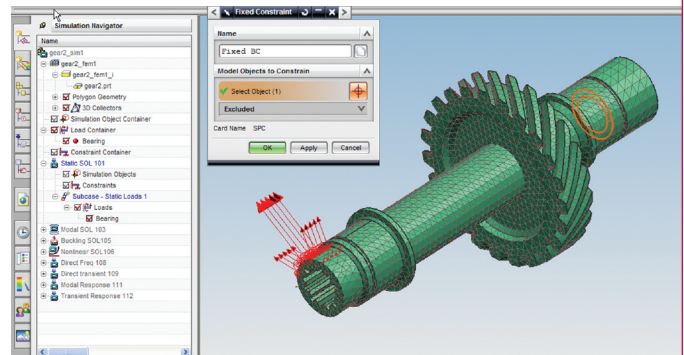
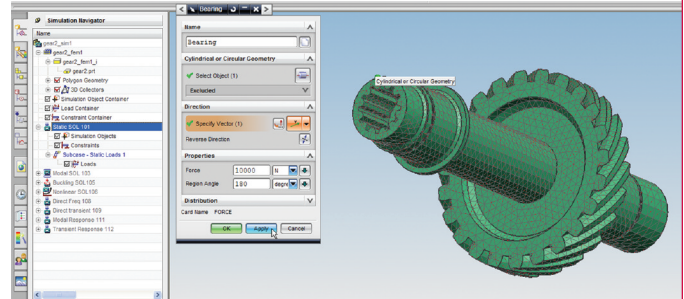
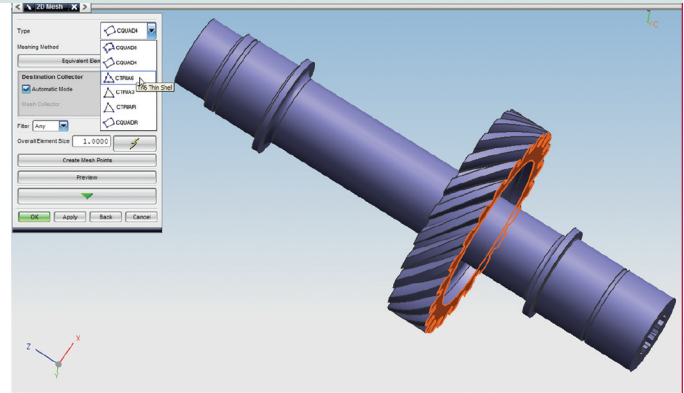
Loads and boundary conditions

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

Automatic connection mesh support

Use mesh mating condition to connect individual 2D or 3D meshes together at a specified interface. NX ensures that connectivity is maintained at that interface at runtime. Use edge-face connection to define the connection between a set of edges and a set of faces. Use this feature whenever there are meshes to be connected in T-junction configuration, for example, fins or stiffeners attached to surfaces.



Weld mesh lets you locate/automate the recognition of weld features (connections) and then automate the creation of their FE model representation, including consideration for midsurfaces. Use weld mesh to create weld elements (ID mesh) from weld features (connections).

Use contact mesh to create point-to-point contact between two edges or a portion of two edges defined by limiting points.

Surface contact mesh lets you create and define contact elements between two selected faces of a solid or between different components. The options available in the surface contact dialog depend on the solver environment selected as the currently active solution.

Compatibility

The Nastran environment is compatible with the following Nastran releases:

- NX Nastran V5.0 or earlier
- MSC Nastran V2005.0 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 pre-requisite. It is available on all NX supported hardware/OS platforms (Windows, LINUX and Unix) including selected 64-bit platforms.



Contact

Siemens PLM Software

Americas 800 498 5351

Europe +44 (0) 1276 702000

Asia-Pacific 852 2230 3333

www.siemens.com/plm