SIEMENS



Siemens PLM Software

NX CAE environment for Ansys solver

Pre- and postprocessing analysis models in NX CAE for the Ansys solver

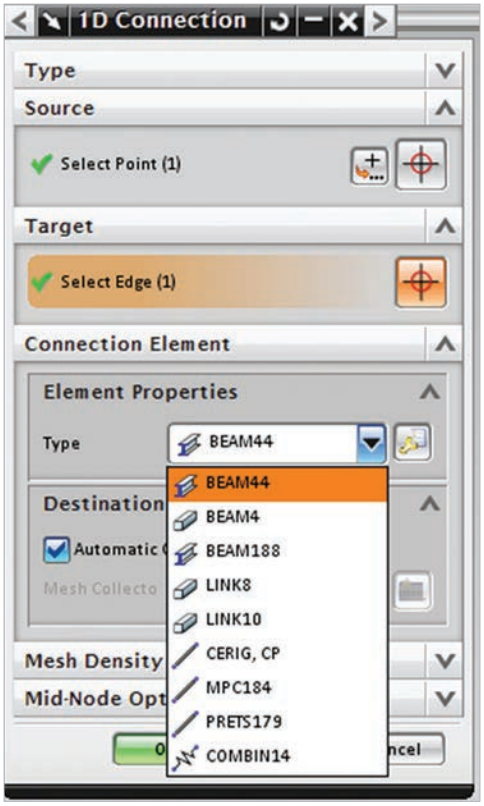
Benefits

- Enables engineers using NX CAE to generate finite element models for the Ansys solver
- Simplifies the Ansys modeling process by enabling engineers to create analysis models based on geometry or legacy Ansys input data files
- Reduces or eliminates intermediate manual processing of data files by generating run-ready decks directly from NX CAE
- Immerses engineers in the Ansys environment by using familiar Ansys terminology and extensively supporting Ansys-specific elements and entities

Summary

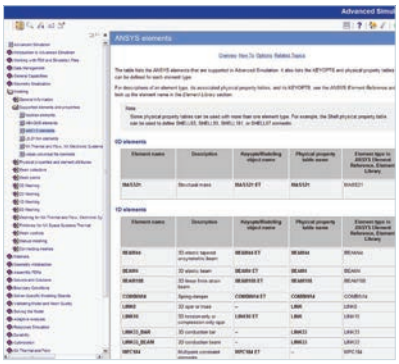
The Ansys environment for NX™ CAE enables engineers to build finite element models, define solution parameters and view the solution results for the Ansys solver. The environment immerses engineers with familiar Ansys language for element definitions, loads and boundary conditions, solution parameters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

The power of NX CAE pre- and post-processing is an ideal partner for creating Ansys models and solutions. NX CAE 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 pre-processors. Adding the Ansys environment to NX CAE enables you to build Ansys run-ready input



www.siemens.com/nx

NX CAE environment for Ansys solver



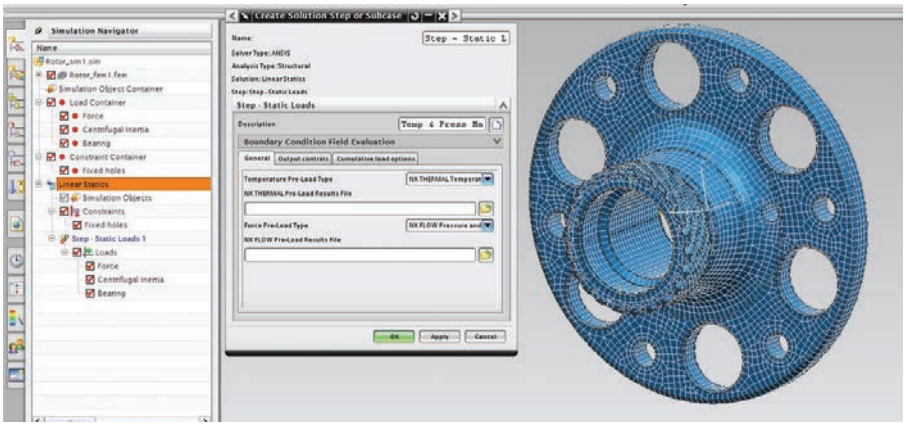
data files, so little or no intermediate processing is ever needed. In addition to building Ansys models, the NX Ansys environment imports solution results directly from Ansys results files into NX CAE for post-processing. The environment delivers import/export capabilities so you can import Ansys data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

Import Ansys models

- Import complete Ansys finite element models including bulk data as well as solution controls and step controls. PREP7 and CDWRITE formats are supported
- Import beam element cross-section shapes for graphical display
- Import support for nodal thickness values on 2D elements
- Importing material orientation vectors as spatial fields

Create and export Ansys models from NX
The following types of analyses are supported:

- Structural linear static, modal, buckling, and nonlinear static
- Axisymmetric structural linear and nonlinear statics
- Nonlinear buckling
- Transient dynamic



- Thermal and axisymmetric thermal
- Thermal transient
- Thermal-structural multiphysics

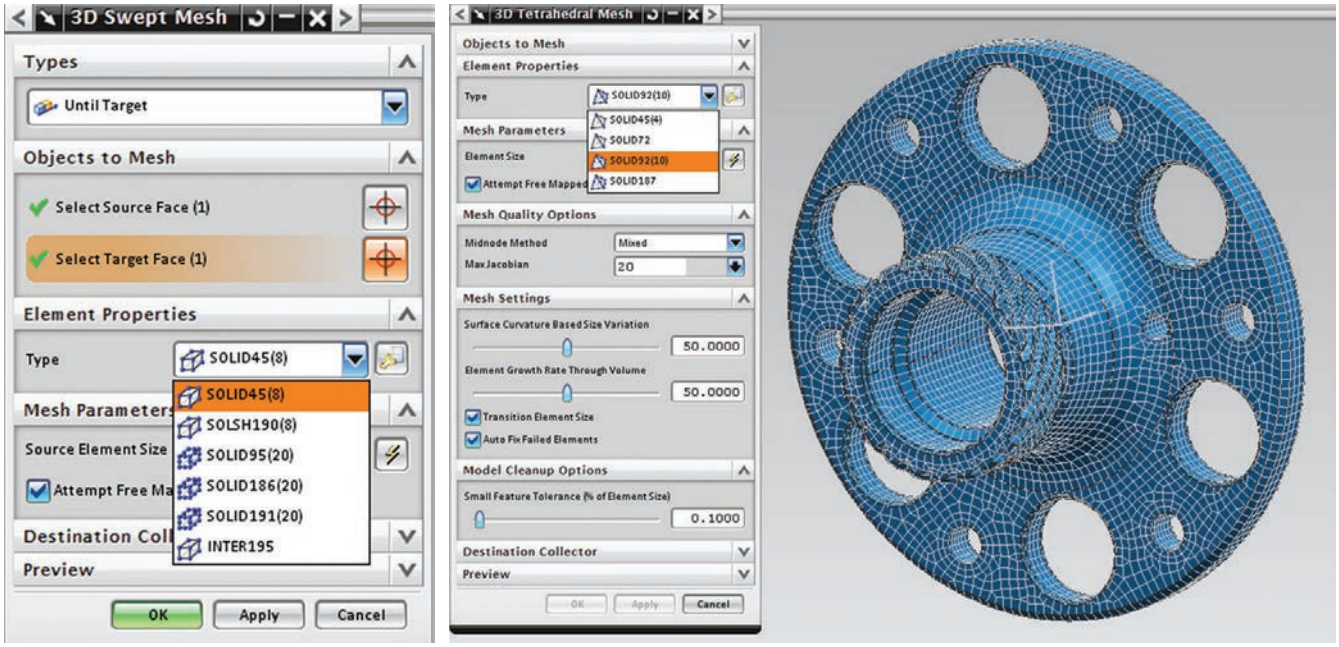
Elements and other entities
A wide variety of elements and other model entities are supported.

Structural element types:

- Rod (BEAM4, LINK8, LINK10, LINK180)
- Beam (BEAM44, BEAM188)
- Pipe (PIPE228)
- Axisymmetric solids, plane stress and plane strain (PLANE42, PLANE82, PLANE182, PLANE183, INTER192)
- Thin shell (SHELL63, / 93/ 91/ 99/ 181/ 281)
- Solid elements (SOLID45 /92 /95 /185 /186 /187 /191 /226 /227 SOLSH190, INTER195)
- Surface-to-surface contact elements (CONTA174/ TARGE170)
- Rigid, constraint, spring, damper, gap and mass elements (CP, CE, CERIG, MPC184, COMBIN14, COMBIN39, COMBIN40, CONTAC12, CONTAC52, CONTA178, MASS21)
- Pretension elements (PRETS179)
- Shell and solid laminates

Thermal element types:

- Rod (BEAM4) and beam (BEAM44) elements
- 3D conduction bar (LINK33_BAR)
- 3D conduction beam (LINK33_BEAM)
- 1D convective element (LINK34)
- Thin shell (SHELL57/ 93, SHELL131/132)
- Axisymmetric solids (PLANE55 /77)
- Solid elements (SOLID70 /87 /90 /226 /227)
- Mass elements (MASS21)



- Thermal-fluid pipe (FLUID116)
- Surface elements (SURF151, /152)

All KEYOPTS and Real Constants are supported in the graphical user interface.

A complete list of Ansys 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 force
- Nodal temperature and heat source
- Varying nodal pressure loads
- Acceleration loads
- Elemental face and edge pressure
- Elemental face and edge convection
- Elemental heat flux
- Coupled DOF
- Multi-point constraints
- Beam distributed load
- Nodal restraint
- Nodal temperature restraint (NX boundary or from Ansys temperature results file or other solvers' temperature results)
- Contact in modal and buckling solutions
- Thermal coupling
- Thermal contact

Compatibility
The Ansys environment is compatible with the following Ansys releases:

- Ansys v15 or earlier

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

Contact
Siemens PLM Software
Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3308

www.siemens.com/plm

© 2014 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Fibersim, Geolus, GO PLM, I-deas, JT, NX, Parasolid, Solid Edge, Syncrofit, Teamcenter and Tecnomatix are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other logos, trademarks, registered trademarks or service marks belong to their respective holders.
10666-Y5 12/14 B