

NX and the Ansys environment

fact sheet

Siemens PLM Software

www.siemens.com/plm

► Summary

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

Benefits

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

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

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

The Ansys environment utilizes Ansys terminology and supports a wide variety of elements and other Ansys-specific entities

Features

A broad range of Ansys entities and solutions are supported

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

Export ready-to-run Ansys models for structural and thermal analyses

Import solution results directly from the solver output file for NX post-processing

ANSYS elements

The table lists the ANSYS elements that are supported in Advanced Simulation. It also lists the KEYOPTS and physical property tables that can be defined for each element type.

For descriptions of an element type, its associated physical property tables, and its KEYOPTS, see the ANSYS Element Reference and look up the element name in the Element Library section.

Note:
Some physical property tables can be used with more than one element type. For example, the Shell physical property table can be used to define SHELL63, SHELL93, SHELL181, or SHELL57 elements.

0D elements

Element name	Description	Keyopts/Modeling object name	Physical property table name	Element type in ANSYS Element Reference, Element Library
MASS21	Structural mass	MASS21 ET	MASS21	MASS21

1D elements

Element name	Description	Keyopts/Modeling object name	Physical property table name	Element type in ANSYS Element Reference, Element Library
BEAM44	3D elastic tapered unsymmetric beam	BEAM44 ET	BEAM44	BEAM44
BEAM4	3D elastic beam	BEAM4 ET	BEAM4	BEAM4
BEAM188	3D linear finite strain beam	BEAM188 ET	BEAM188	BEAM188
COMBIN14	Spring-damper	COMBIN14 ET	COMBIN14	COMBIN14
LINK8	3D spar or truss	-	LINK	LINK8
LINK10	3D tension-only or compression-only spar	LINK10 ET	LINK	LINK10
LINK33_BAR	3D conduction bar	-	LINK33	LINK33
LINK33_BEAM	3D conduction beam	-	LINK33	LINK33

Environment capabilities

- A broad range of Ansys entities and solutions are supported
- Creating complete finite element models including boundary conditions, applied loads and solution control for Ansys input files
- Exporting run-ready Ansys models for structural and thermal analyses
- Importing solution results directly from solver results files for NX post-processing

Importing Ansys models

- Complete Ansys finite element models including bulk data as well as solution controls and step controls. PREP7 and CDWRITE formats are supported.
- Estimation of beam element cross-section shapes for graphical display

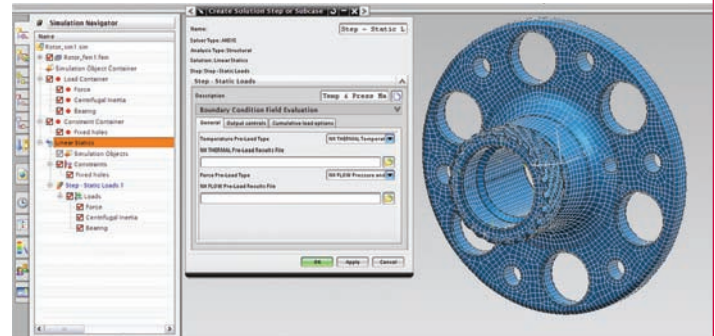
Using NX to create Ansys models

The following types of analyses are supported:

- Structural static, modal
- Steady-state heat transfer
- Eigenvalue buckling

The power of NX Advanced FEM pre- and post-processing is an ideal complement to the capabilities of the Ansys solver.

NX Advanced FEM geometry-based finite element modeling tools simplify the modeling process. The Ansys environment builds an Ansys run-ready input file, so little or no intermediate processing is needed. Solution results are imported directly from Ansys results files. The Ansys translator is enhanced with every release. It is also possible to manually modify your input file prior to solution if required.



Elements and other entities

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

Structural element types:

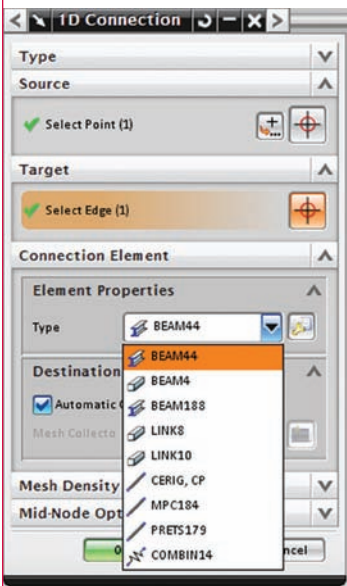
- Rod (BEAM4, LINK8, LINK10)
- Beam (BEAM44, BEAM188)
- Axisymmetric solids, plane stress and plane strain (PLANE42, PLANE82, INTER192)
- Thin shell (SHELL63, / 93/ 91/ 99/ 181)
- Solid elements (SOLID45 /92 /95 /186 /187 /191 SOLSH190, INTER195)
- Surface-to-surface contact elements (CONTA174/ TARGE170)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

Thermal element types:

- Rod (BEAM4) and beam (BEAM44) elements
- Thin shell (SHELL57/ 93)
- Axisymmetric solids (PLANE55 /77)
- Solid elements (SOLID70 /87 /90)
- Mass elements (MASS21)

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
- 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)

Compatibility

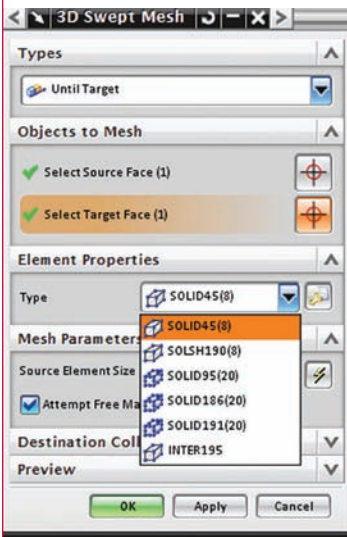
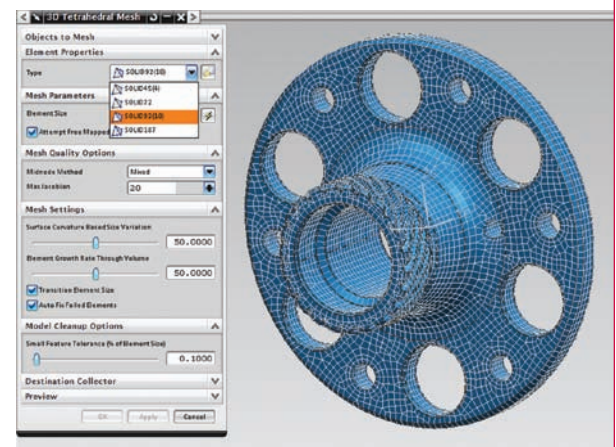
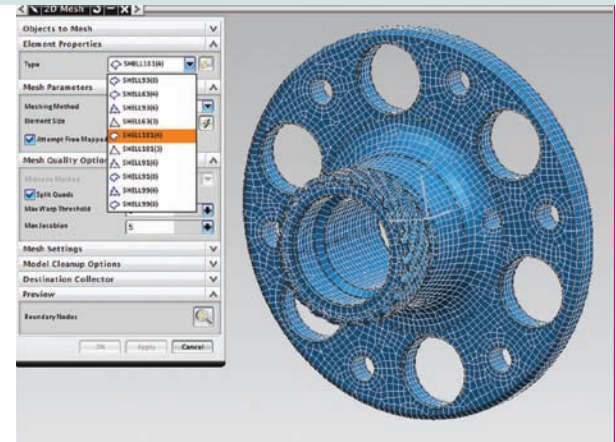
The Ansys environment is compatible with the following Ansys releases:

- Ansys v11.0 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 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

© 2008 Siemens Product Lifecycle Management Software Inc. All rights reserved. Siemens and the Siemens logo are registered trademarks of Siemens AG. Teamcenter, NX, Solid Edge, Tecnomatix, Parasolid, Femap, I-deas, Velocity Series and Geolus 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 used herein are the property of their respective holders. 8/08