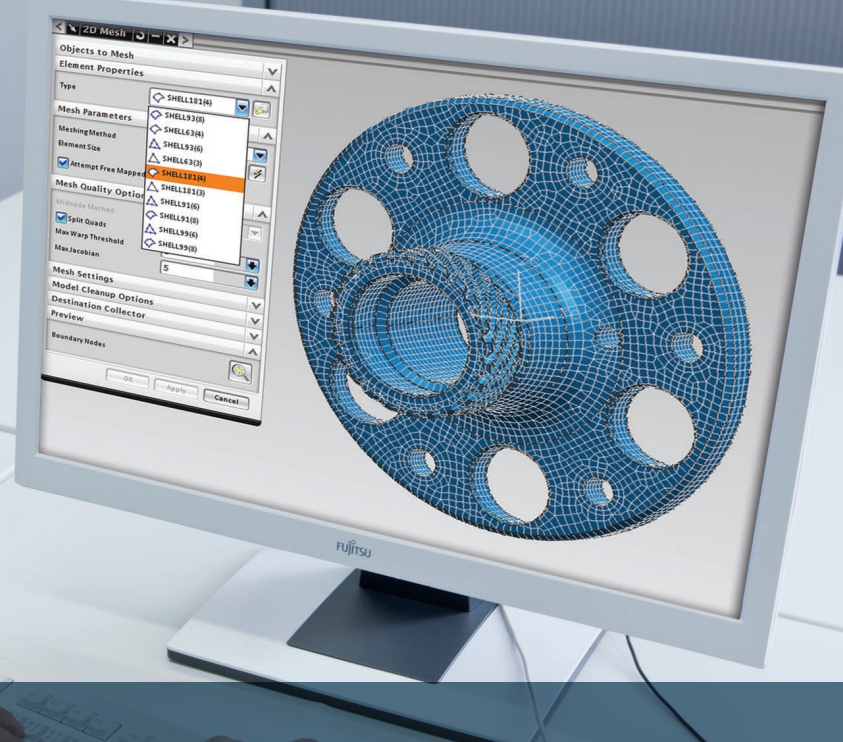




SIEMENS



GEOMETRIC  
SOLUTIONS



Siemens PLM Software

# NX CAE environment for Abaqus solver

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

## Benefits

- Enables engineers using NX CAE to generate finite element models for the Abaqus solver
- Simplifies the Abaqus modeling process by enabling engineers to create analysis models based on geometry or legacy Abaqus 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 Abaqus environment by using familiar Abaqus terminology and extensive support of Abaqus-specific elements and entities

## Summary

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

The power of NX CAE pre- and post-processing is an ideal partner for creating Abaqus 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 Abaqus environment to NX CAE enables you to build Abaqus run-ready

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

## Import Abaqus models

- Abaqus finite element models can be imported into NX.
- Model information from Abaqus input files is supported.

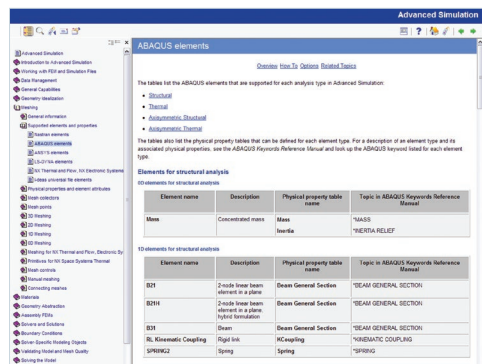
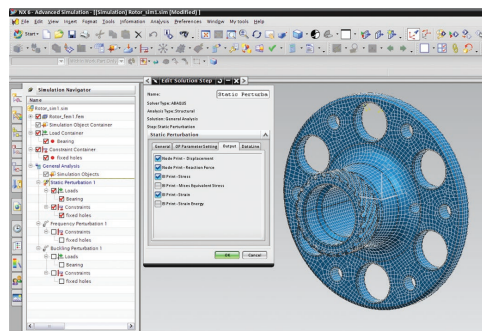
## Create and export Abaqus models from NX

- Create and export complete run-ready Abaqus decks including model data (finite element entities) and history data (step definition, solution control, boundary condition set and contact definition).

# NX CAE environment for Abaqus solver

Supports the following analysis types:

- Structural: statics, buckling and modal
- Steady-state heat transfer
- Visco analyses
- Transient modal dynamic analyses
- Implicit dynamic stress and displacement analyses
- Direct cyclic analyses
- Transient heat analysis in axisymmetric thermal analyses



## Elements and other entities

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

- Shell and solid elements
- Axisymmetric solid elements
- Spring, beam elements
- Initia, rotary inertia, point mass, rigid body elements
- Interaction elements including gap, rigid surface

- Abaqus surface-to-surface contact
- Shell and solid laminates

## 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
- Multipoint constraints and kinematics coupling
- Nodal temperature restraint (from NX boundary or from Abaqus FIL-file or other solvers' temperature results)
- Bolt pre-loaded length constraint

A complete list of Abaqus 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/.

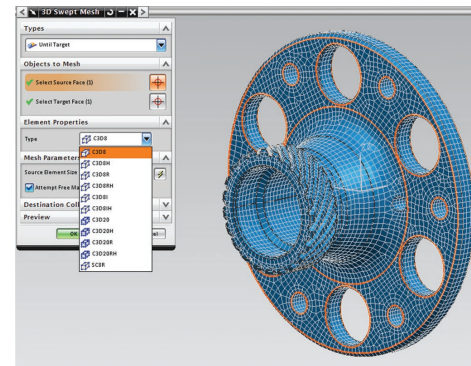
## Compatibility

The Abaqus environment is compatible with the following Abaqus releases:

- Abaqus v6.13 or earlier

## Supported hardware/OS

The Abaqus environment is an add-on module within the NX Advanced Simulation suite. It requires a license of NX CAE as a pre-requisite. 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](http://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.  
 10667-Y7 12/14 B



844-GEO-SUPT  
 support@geoplms.com  
 geoplms.com