

Abaqus Data Translator for NX I-deas

Advanced bi-directional exchange of NX I-deas Simulation and Abaqus data

fact sheet

Siemens PLM Software

www.siemens.com/plm

► Summary

The Abaqus Data Translator for NX® I-deas® software provides bi-directional exchange of finite element models and simulation results between NX I-deas and Abaqus. This allows the user to combine the NX I-deas finite element pre- and post-processing software (NX MasterFEM) with Abaqus analysis. NX I-deas Simulation and the Abaqus Data Translator provide all the tools needed to build models, to define Abaqus solution parameters and to view the solution results. Both Abaqus Standard and Abaqus Explicit solvers are supported.

Benefits

NX I-deas geometry-based FE modeling tools simplify the modeling process

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

Support is provided for a wide variety of elements and other model entities

Features

Supporting a broad range of Abaqus entities and solutions

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

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

Importing solution results directly from solver output files for NX I-deas post-processing

The power of NX I-deas pre- and post-processing is an ideal partner with Abaqus solution capabilities. NX I-deas geometry-based finite element modeling tools simplify the modeling process. The NX I-deas translator builds Abaqus run-ready bulk data decks, so little or no intermediate processing is needed. Solution results are imported directly from Abaqus binary or ASCII results files.

Some Abaqus entities, which are not supported in NX I-deas, can be directly created in the translator graphical user interface, such as analytical rigid surface, contact properties and amplitude curves.

Importing Abaqus models and results

The Abaqus Data Translator for NX I-deas directly reads Abaqus input files and results files and creates an equivalent NX I-deas finite element model. It translates Abaqus files into NX I-deas Universal files or imports directly into NX I-deas. Complete Abaqus FE models including nodes, elements, physical and material properties, boundary conditions and loads can be imported to NX I-deas. Mapping of Abaqus element types to specific NX I-deas element types is supported.

Abaqus FE models imported into NX I-deas can then be post-processed or solved using NX Model Solution. Imported Abaqus models can also be exported from NX I-deas to Nastran, Ansys or back to Abaqus for solution. The Abaqus Data Translator provides bi-direction FE model exchange with Abaqus.

Exporting Abaqus models

- Complete run-ready Abaqus decks including model data (restart control, element mapping, ELSET, NSET mapping, initial conditions, contact and amplitude curves definitions) and history data (step definition, solution control, boundary condition set and contact definition)
- Support for the following analysis types:
 - Structural: statics, Eigenvalue buckling, dynamic, steady-state dynamic, frequency, modal dynamic, geostatic
 - Heat transfer: steady-state and transient
 - Coupled temperature/displacement
- Optional selection of the Abaqus basic coordinate system
- Optional selection of model subset using NX I-deas groups
- User-defined entries in model data and history data

Prerequisite

NX MasterFEM

NX I-deas Simulation Modeling Set

Elements and other entities

A wide variety of elements and other model entities are supported. In addition, NX I-deas/Abaqus element mapping allows you to define specific element types in NX I-deas to map to Abaqus element types.

- Plane stress and plane strain elements
- Shell and solid elements
- Axisymmetric shell and solid elements
- Infinite elements
- Membrane and axisymmetric membrane elements
- Spring, truss, beam and pipe elements
- Inertia, rotary inertia, point mass, rigid body elements
- Interaction elements including gap, rigid surface
- Full support of Abaqus Contact. You can create new contact or use contact pairs defined in NX I-deas
- Linear/nonlinear material including stress-strain data and temperature dependent properties

A complete list of Abaqus entity support is provided in the NX I-deas online help documentation.

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 coupled degrees of freedom

Analysis results

The following results are recovered into NX I-deas for post-processing and display:

- Nodal based results including displacement, velocity, acceleration, reaction force, contact stresses and clearances, temperature
- Element based results including stress, strain, elastic/inelastic/plastic strain, creep strain, logarithmic/nominal strain, strain energy density, element nodal force, beam and shell section force, heat flux

**Contact**

Siemens PLM Software

Americas 800 498 5351

Europe 44 (0) 1276 702000

Asia-Pacific 852 2230 3333

www.siemens.com/plm**SIEMENS**