I-DEAS® Data Translator, ABAQUS to/from I-DEAS

for exchanging FE models and results between I-DEAS® and ABAQUS

ABAQUS Finite Element Modeling and Post-Processing

This I-DEAS® Data Translator allows you to use I-DEAS Finite Element Modeling™ (FEM) software to pre- and post-process ABAQUS analysis. I-DEAS Simulation Modeling and the ABAQUS Data Translator provide all of the tools needed to build models, boundary conditions and define solution parameters for ABAQUS. The translator provides bi-directional exchange of FE models and simulation results with ABAQUS solvers. Both ABAQUS Standard and ABAQUS Explicit solvers are supported. FE models constructed in I-DEAS can be directly written to an ABAQUS input file. ABAQUS results can be directly imported back into I-DEAS for post-processing results. ABAQUS input files (FE Models) can also be translated into I-DEAS Universal files and imported to I-DEAS.

Practical Usage

The power of I-DEAS pre- and post-processing makes an ideal partner with ABAQUS solution capabilities. I-DEAS geometry-based FE modeling tools simplify the FE modeling process. I-DEAS Data Translator builds ABAQUS solver input files so no intermediate processing is needed. Solution results are imported directly from solver output files.

An ABAQUS file exported from I-DEAS contains the complete model information and control cards required for an ABAQUS solve. The model can be solved locally or copied to a remote computer for execution. The ABAQUS solver results output file can then be imported into I-DEAS for post-processing.

The capabilities include:
•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 I-DEAS post-processing.

Importing ABAQUS FE Models

The translator directly reads ABAQUS input files and creates an equivalent I-DEAS finite element model. It translates ABAQUS files into I-DEAS Universal files. Complete ABAQUS FE models including nodes, elements, physical and material properties, boundary conditions and loads can be imported to I-DEAS. Mapping of ABAQUS element types to specific I-DEAS element types is supported.

ABAQUS FE models imported to I-DEAS can then be post-processed or solved using I-DEAS Model Solution™. Imported ABAQUS models can also be exported from I-DEAS to NASTRAN, ANSYS or back to ABAQUS for solution. The I-DEAS ABAQUS Data Translator provides bi-directional FE model exchange with ABAQUS.

Technical Specifications

Analysis Types

I-DEAS can directly create models for the following types of analysis:

Structural
•Statics.
•Eigenvalue buckling.
•Dynamic analysis using direct integration.
•Direct-solution steady-state dynamic.
•Natural frequency extraction.
•Transient modal dynamic.
•Mode-based steady-state dynamic.
•Explicit dynamic.

Heat Transfer
•Steady-state and transient heat transfer.
•Coupled temperature/displacement.

Elements/Entities

A wide variety of elements and other model entities for structural and thermal analysis are supported. In addition, I-DEAS/ABAQUS element mapping allows you to define specific element types and I-DEAS to map to ABAQUS element types.

Structural
•Plane stress, plane strain, and generalized plane strain elements.
•Solid elements.
•Axisymmetric solid elements.
•Infinite elements.
•Membrane and axisymmetric membrane elements.
•Truss, beam and pipe elements.
•Shell and axisymmetric shell elements.
•Inertial and rigid elements including point mass, rotary inertia, and rigid body elements.
•Spring elements.
•Contact and interaction elements including gap, interface contact, and rigid surface elements.

Heat Transfer
•One-dimensional solid elements.
•Two-dimensional solid elements.
•Three-dimensional solid elements.
•Axisymmetric solid elements.
•Shell elements.
•Axisymmetric shell elements.

A complete list of ABAQUS element support is provided in the I-DEAS online help documentation.

Loads and Boundary Conditions

Loads and boundary conditions for structural and thermal analysis are supported.

- Nodal force
- Nodal and beam temperature
- Element face and edge pressure, flux, convection, and radiation
- Element distributed heat source
- Coupled DOF multi-point constraints
- Nodal restraint
- Gravity loads

Analysis Results

The following results are recovered into the I-DEAS FEM or MTS test correlation software for post-processing and display:

Nodal-based results

- Displacement
- Velocity
- Acceleration
- Reaction force
- Contact tractions
- Contact clearances
- Temperature

Element-based results

- Stress
- Total strain
- Total elastic strains
- Total inelastic strains
- Plastic strain
- Creep strain
- Logarithmic strain
- Nominal strain
- Strain energy density
- Element nodal force
- Beam and shell section force
- Heat flux

Compatibility

I-DEAS is compatible with ABAQUS 5.0 to 5.8 releases. The translator is supported on UNIX and Windows NT hardware platforms. Contact SDRC for up-to-date compatibility information.

Prerequisite

Core Simulation

For More Information

For more information, contact your local SDRC representative or call 1-800-848-7372.