

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

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

ANSYS 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 ANSYS analysis. I-DEAS Simulation modeling and the ANSYS Data Translator provide all of the tools needed to build models, boundary conditions and define solution parameters for ANSYS. The translator provides bi-directional exchange of FE models and simulation results with ANSYS solvers. FE models constructed in I-DEAS can be directly written to an ANSYS input file. ANSYS results can be directly imported into I-DEAS for post-processing results.

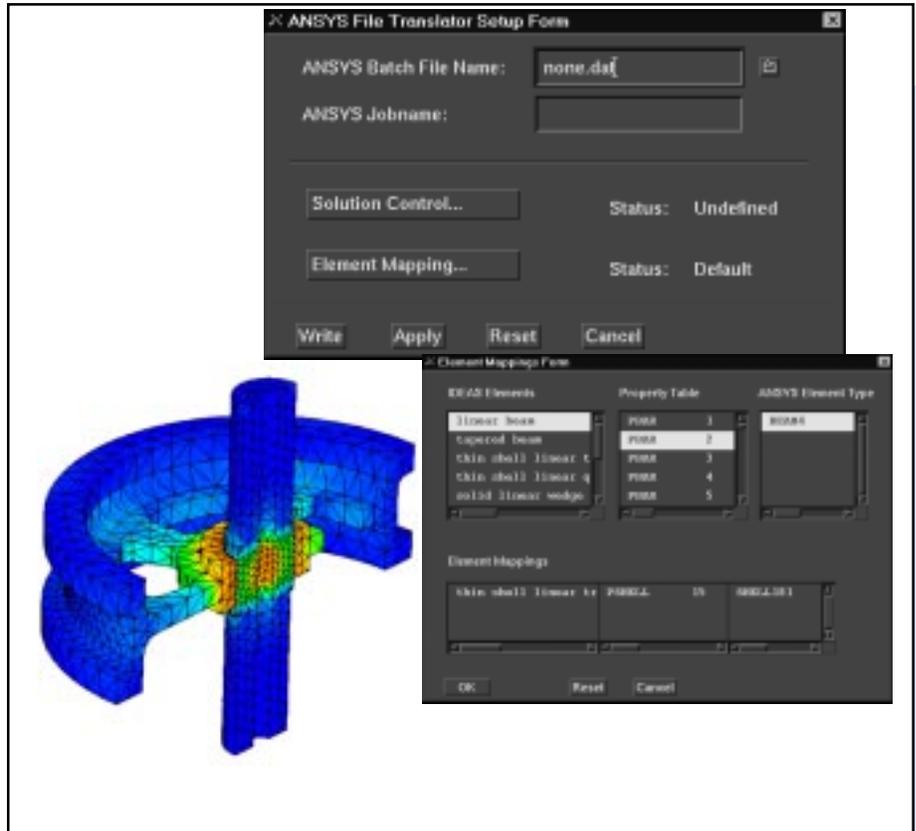
Practical Usage

The power of I-DEAS pre- and post-processing is an ideal partner with ANSYS solution capabilities. I-DEAS geometry-based FE modeling tools simplify the FE modeling process. The I-DEAS Data Translator builds ANSYS PREP 7 solver input files so no intermediate processing is needed. Solution results are imported directly from solver output files.

An ANSYS PREP 7 file exported from I-DEAS contains the complete model information and control cards required for an ANSYS solve. The model can be solved locally or copied to a remote computer for execution. The ANSYS 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 ANSYS input files.
- Exporting ready-to-run ANSYS models for structural, thermal, and magnetic analyses.
- Importing solution results directly from solver output files for I-DEAS post-processing.



I-DEAS Finite Element Modeling and the ANSYS to/from I-DEAS Data Translator combine to make I-DEAS an integrated pre- and post-processor for ANSYS.

Technical Specifications

Analysis Types

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

- Structural, static, modal, and transient dynamics
- Steady-state and transient heat transfer
- Eigenvalue buckling
- Magnetics

Elements/Entities

A wide variety of elements and other model entities for structural, thermal, and magnetic analysis are supported. In addition, I-DEAS to ANSYS element mapping allows you to define specific element types in I-DEAS to map to ANSYS element types. I-DEAS groups can be exported as ANSYS node and element components.

Structural

- Rod, beam and pipe elements
- Axisymmetric solids
- Axisymmetric thin shell elements
- Thin shell, plane stress, plane strain, and membrane elements
- Solid elements
- Rigid, constraint, spring, damper, gap, and lumped mass elements

Thermal

- Rod and beam elements
- Thin shell, plane stress, plane strain and membrane elements
- Axisymmetric solids
- Solid elements
- Lumped mass elements

Magnetic

- Thin shell, plane stress, plane strain and membrane elements
- Axisymmetric solids
- Solid elements

A complete list of ANSYS 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 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
- Kinematic DOF
- Nodal restraint
- Nodal temperature restraint

Analysis Results

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

- Nodal displacement
- Reaction forces
- Nodal temperature
- Magnetic potential
- Stress
- Magnetic field intensity
- Strain
- Thermal flux
- Magnetic forces
- Thermal gradient
- Magnetic flux
- Heat flow rates
- Magnetic flux density
- Element forces

Compatibility

I-DEAS 8 is compatible with the ANSYS 5.6 release. 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.