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

for exchanging data between I-DEAS® and NASTRAN

NASTRAN Finite Element Modeling and Post-Processing

This I-DEAS® Data Translator allows you to use I-DEAS Finite Element Modeling™ software to pre- and post-process NASTRAN analysis. I-DEAS Simulation modeling and the NASTRAN Data Translator provide all of the tools needed to build models, boundary conditions and define solution parameters for NASTRAN. The translator provides bi-directional exchange of FE models and simulation results with NASTRAN solvers. FE models constructed in I-DEAS can be directly written to a NASTRAN input file. NASTRAN results can be directly imported back into I-DEAS for post-processing results. MSC, CSA, UAI and COSMIC NASTRAN are supported.

Practical Usage

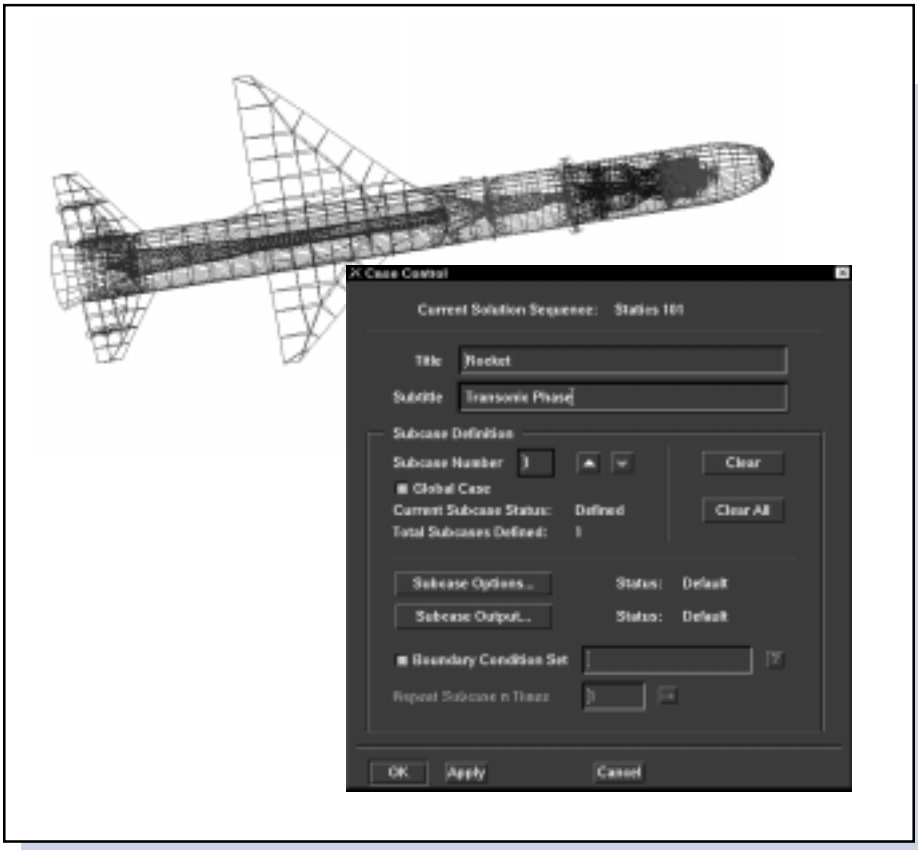
The power of I-DEAS pre- and post-processing is an ideal partner with NASTRAN solution capabilities. I-DEAS geometry-based FE modeling tools simplify the FE modeling process. In addition to models generated using I-DEAS, you can import and combine previously built NASTRAN models. Models can be imported via NASTRAN BULK data or NASTRAN Output2 files. Solution results are imported via NASTRAN Output2 files.

A NASTRAN deck exported from I-DEAS contains the complete model information and control cards required for a NASTRAN solve. The NASTRAN deck can be solved locally or copied to a remote computer for execution. The NASTRAN solver results Output2 binary file can then be imported into I-DEAS for post-processing. Both the FE model and the results can be imported from the Output2 file. Binary files from UNIX, Windows NT, and CRAY are supported.

Import/Export Capabilities

NASTRAN import capabilities include:
•Complete NASTRAN finite element models including boundary conditions, applied loads, and analysis results.
•Importing model data from either bulk data file(s) or binary Output2 file.
•Importing solution results from binary Output2 file.
•Estimating cross-section shapes for bars, beams, rods, tubes, and bend elements, enabling shape display in I-DEAS.
•Assigning element colors based on physical property ID or super-element/ substructure membership.
•Translating the NASTRAN data to an I-DEAS/Universal file for import into I-DEAS which allows models to be transferred to/from other computers for remote NASTRAN solves.
•Importing NASTRAN data directly to I-DEAS without generating intermediate files.

NASTRAN export capabilities include:
•Creating complete finite element models including boundary conditions, applied loads, and solution control for NASTRAN input files.
•Exporting ready-to-run NASTRAN decks for linear statics and normal modes analyses.



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

Technical Specifications

Analysis Types

I-DEAS can directly create models for the following types of analysis:
•Linear statics
•Statics with inertia relief
•Normal mode dynamics
•Steady-state heat transfer

Elements/Entities

A wide variety of elements and other model entities for structural and thermal analysis are supported.
•Lump mass, spring, damper, gap, rigid, and constraint elements
•Axisymmetric shell elements
•Axisymmetric solid elements
•Rod, beam, and pipe elements
•Shell and membrane elements
•Plane stress and plane strain elements
•Solid elements
•Super-elements are supported through SESET definitions based on I-DEAS groups.

A complete list of NASTRAN 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 and element loads
•Gravity, rotational velocities, and acceleration loads
•Thermal heat loads
•Nodal restraints and temperatures

Analysis Results

The ASCII NASTRAN input file generated by I-DEAS software can be transferred to other computer systems for analysis in NASTRAN. The following results are recovered into I-DEAS FEM or MTS test correlation software for post-processing and display:
•Stress and ply stress
•Strain, ply strain, strain, and energy
•Composite failure index
•Displacement, mode shape, velocity, acceleration, and pressure
•Element force and grid point force
•Reaction force, temperature, and heat flux
•For dynamics, both real and complex results can be processed.

Compatibility

Full support for CSA, MSC, and UAI NASTRAN is provided with partial support for COSMIC NASTRAN. I-DEAS 8 is compatible with the following releases:
•CSA NASTRAN V99
•MSC NASTRAN V70.7
•UAI NASTRAN V20.0

The translator is supported on UNIX, Windows NT, and CRAY 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.