I-DEAS® Model Solution™ Nonlinear

for analyzing nonlinear behavior

Efficient and Accurate Nonlinear Finite Element Analysis

Linear finite element analysis is useful whenever the stress-strain relationship is linear, and displacements and rotations are small. These assumptions are valid for a wide variety of problems. However, many structural problems are, to varying degrees, nonlinear. Whenever loads, material properties, contact conditions, or structural stiffness change with displacements, the problem is nonlinear. I-DEAS® Model Solution™ Nonlinear software provides the ability to analyze these types of nonlinear behavior.

I-DEAS Finite Element Modeling™ software provides all the capabilities necessary to create finite element models for I-DEAS Model Solution Nonlinear. Virtually all elements in the I-DEAS Model Solution Linear software library are supported for nonlinear analysis. Conversion of existing linear models for nonlinear analysis only requires definition of the nonlinear boundary conditions, selection of solution options, and input of nonlinear material properties. I-DEAS Extended Finite Element Modeling software has the ability to define time varying loading and boundary conditions directly. The I-DEAS Material Data System™ supports linear elastic, elastic-plastic, and creep models available in I-DEAS Model Solution Nonlinear. Combined, these capabilities provide you with a complete modeling and analysis solution for nonlinear problems.

I-DEAS Model Solution Nonlinear is an integrated part of I-DEAS software, which means no additional input files need to be created for the analysis. This greatly simplifies the process of nonlinear analysis. The overall time to do analysis is reduced, and accuracy is assured by maintaining data integrity among model building, solving, and results interpretation.

General Capabilities

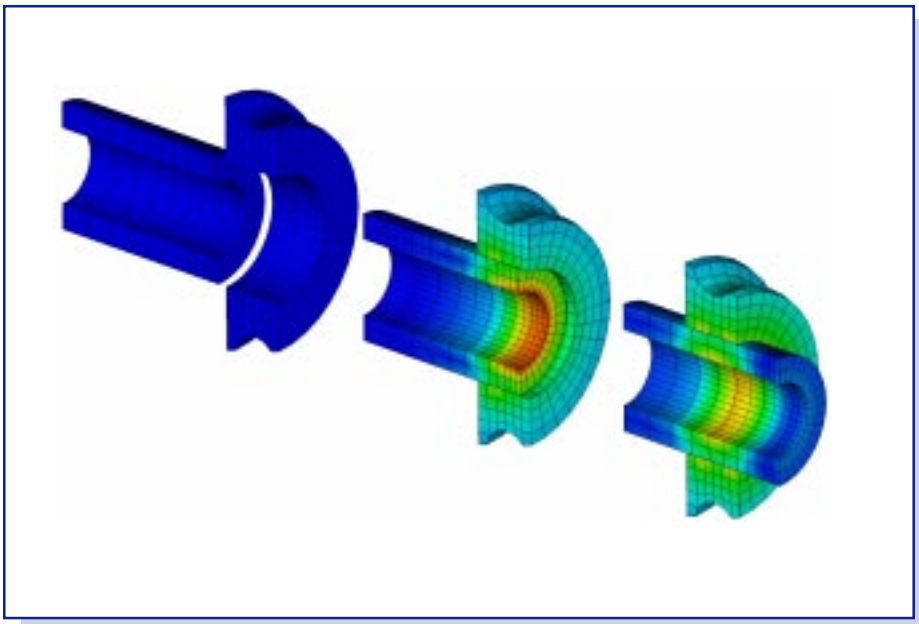
I-DEAS Model Solution Nonlinear is a general finite element analysis program for nonlinear static structural analysis. It is capable of performing the following types of analysis:

- Geometric nonlinear analysis.
- Material nonlinear analysis.
- Surface-to-surface contact analysis.
- Combined geometric nonlinear, material nonlinear, and contact analysis.
- Automatic load stepping control assures that the most complex solutions are completed automatically.
- Post buckling analysis uses the arc length methods.

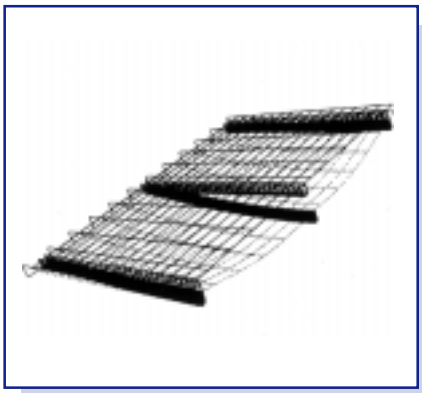
Geometric Nonlinearity

I-DEAS Model Solution Nonlinear uses an updated Lagrangian formulation. With this formulation, the finite element equilibrium equations are written with respect to the current (deformed) configuration of the structure.

The solution of the equilibrium equations is iterative, and improved estimates of the solution are successively generated until the user specified convergence tolerance is satisfied. This formulation accounts for displacement dependent loads and boundary conditions, large rotations and displacements, and stress stiffening.



I-DEAS Model Solution Nonlinear software has capabilities to solve geometric and material nonlinear problems.



Results such as this deformed geometry, are automatically available at your selected time steps for evaluation in I-DEAS FEM software.

Material Nonlinearity

I-DEAS Model Solution Nonlinear software contains yield surface plasticity models and time dependent creep models.

Plasticity

The plasticity model uses a Von Mises yield function with associated flow rule. Isotropic hardening, kinematic hardening, and combined hardening are available options. Kinematic and combined hardening use either Prager or Ziegler-Prager hardening options. The required stress-strain data are entered through the I-DEAS Material Data System software.

Creep

Six different creep models with associated flow rule are available in I-DEAS Model Solution Nonlinear. Creep constants are entered through I-DEAS Material Data System. Adaptive time-stepping is used to produce accurate results economically. New software is available for curve fitting creep data to the I-DEAS creep models.

Curve Fitting of Creep Data

To use creep data in I-DEAS Model Solution, it is necessary to curve fit the data and then enter the coefficients into the material database system. To assist users in performing this task, there is a curve fitting package that is delivered with I-DEAS software and runs on UNIX platforms. This “creep fitter” provides functions to enter the data in various formats and units; curve fit the data using a linear and a non-linear solver to five of the standard creep models utilized in I-DEAS Model Solution; and finally to output the desired curve fits to a universal file.

Solution Monitor

The Nonlinear Convergence Monitor graphically displays the convergence of the solution, estimating the time to completion, and enables you to safely terminate the solve at any time.

Results

The following results, at your specified time steps, are automatically available for viewing in I-DEAS Finite Element Modeling software and can be optionally written to a log file:

- Displacements
- Element Forces
- Reaction Forces
- Creep Strains
- Strains
- Elastic Strain Energies
- Plastic Strains
- Stresses
- Stresses on Shell Surfaces
- Stress or Strain at Integrated Points
- Shell Stress Resultants
- Constraint forces (due to contact or constraint equations)

Extensive error reporting is provided to warn you of possible modeling problems and limitations in the analysis. Guidance on where the problems exist and how to correct them is provided.

Quality Assurance

SDRC tests I-DEAS Model Solution Nonlinear against internationally recognized standards including NAFEMS (National Agency for Finite Element Methods & Standards) benchmarks. These test results are available on request; many are included in the Verification manual.

Prerequisites

Core Simulation

For More Information

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