

Available Packaging

Core Simulation functionality can be licensed in one of two ways—Master FEM or Simulation Modeling Set.

Master FEM

Master FEM is a standalone CAE application that requires no other licenses to operate. Master FEM fulfills all prerequisites for Simulation applications, but not for Design applications.

Simulation Modeling Set

Simulation Modeling Set is an add-on to the Core Master Modeler, I-DEAS® Product Design Package, or I-DEAS Artisan™ Package license.

I-DEAS Finite Element Modeling™

I-DEAS Finite Element Modeling (FEM) software provides you with comprehensive capabilities for building finite element models and reviewing analysis results. It uses I-DEAS Master Modeler™ and I-DEAS Master Assembly™ geometry directly and includes the fundamental modeling functions of automatic mesh generation, application of loads and boundary conditions, and model checking. Material property data can be selected from the I-DEAS Material Data System™ software. Post-processing functions allow the recovery of analysis results and provide extensive graphical and numerical tools for gaining an understanding of results. Interfaces to over 20 major analysis programs are available. Specialized capabilities for design and analysis of laminate composite structures and the modeling of beam structures are optionally available. The rules-based I-DEAS Simulation Advisor™ software is also available to guide you through the analysis process.

Geometry Construction/Access

I-DEAS FEM directly uses wireframe, surface, and solid geometry capabilities of I-DEAS Master Modeler and I-DEAS Master Assembly for construction of finite element models. Comprehensive interfaces are also available to give you access to data from other CAD systems.

Geometry for Analysis

Often design geometry must be modified to build an effective finite element model. Details may need to be suppressed or eliminated, additional geometry to control mesh density may be required, or surfaces not present in the design geometry may be needed for meshing. A comprehensive set of tools is provided for geometry editing.

- Geometric features defined in I-DEAS Master Modeler may be suppressed or eliminated interactively.
- Geometric entities (points, curves) can be added, modified, or deleted in more than 20 ways.
- For accurate results using shell finite elements, the mesh must often be generated on the mid-plane of the inner and outer surfaces usually present in the design model. An automated capability for extracting midsurfaces is provided in I-DEAS Master Modeler. Thickness data is automatically extracted for application as shell finite element physical properties. In addition, a set of tools is provided to quickly construct surface geometry from the original solid model:
 - Surface offset
 - Surface interpolate or projection (surface between surfaces)
 - Surface extend
 - Surface divide
 - Surface join and intersection
 - Shell operator

•Analysis data is associated with original part geometry through the I-DEAS Team Data Manager™ software. If the original part is updated, the analysis user is notified of changes so that appropriate updates can be made.

- Unique partitioning capabilities let you quickly segment geometry for mesh control.

Geometry Access

- Wireframe, surface, and solid geometry from other CAD systems can be accessed through standards-based (IGES, VDA, SET) or direct interfaces.
- Specific tools for clean-up of wireframe and surface geometry from traditional CAD systems are provided in I-DEAS Master Modeler and I-DEAS Master Assembly.

- Automatic removal of duplicate points and curves minimizes geometry editing.
- Gaps in curves are removed automatically.
- Small surface patch regions are merged for meshing.
- Inconsistent topologies are merged and cleaned automatically.
- Refit of curve and surface order provides for compatibility with I-DEAS software.
- Ability to define “sections” on top of original geometry facilitates surface and volume meshing.

Mesh Generation

Building the mesh of elements that describes the stiffness properties of a structure is typically the most time-consuming and labor-intensive part of finite element analysis. In addition, the selection of element type and proper mesh density to assure analysis efficiency and accurate results is necessary and part of the “art” of finite element analysis. I-DEAS FEM offers state-of-the-art capabilities for automatic mesh generation.

Free Meshing

- Generates a user-controlled distributed mesh on surface and solid geometry.
- Directly meshes I-DEAS Master Modeler geometry in a single step.
- Uses surfaces defined by an unlimited number of curves and solids bounded by an unlimited number of faces.
- Automatically meshes surfaces including holes, and volumes including holes and interior voids.
- Generates unlimited number of elements on a single surface or in a single volume.
- Generates surface meshes with linear or parabolic quadrilaterals or triangles.
- Generates solid meshes with linear, parabolic, or p-element tetrahedra.
- Fits P-elements accurately to underlying geometry with up to 5th-order representation.
- Lets you define allowable distortion for tetrahedral elements before meshing begins.

- Recognizes anchor nodes used to control the positioning of generated nodes.
- Recognizes reference geometry, points and series, for mesh definition.
- Allows geometry-based definition and generation of lumped mass, rigid bars, and spring elements.
- Allows automatic transition of mesh density.
- Lets you specify global density with a single entry.
- Automatically defines local density based on geometric curvature, proximity of geometric features, or user-defined values on points, curves or surfaces.
- Develops surface mesh transitions with all quadrilaterals.
- Provides automatic optimization-based smoothing to seek minimum element distortion.
- Associates mesh generation settings with geometric features so that updates can occur with design geometry changes.

Section Meshing

Section meshing gives you tools for abstracting only the topology relevant to the FE analysis from the overall design geometry. In addition, section meshing allows you to mesh unstitched geometry imported from external CAD software.

- At a basic level, section meshing lets you define and generate a mesh on a collection of surfaces rather than on each individual surface. Rather than directly modifying the design geometry, you can create sections on top of the model's current topology.
- At a more advanced level, section meshing lets you suppress features that aren't important to the FE analysis.

Adaptive Meshing

I-DEAS FEM provides unique capabilities for automatic modification of mesh density, element order, and element shape, to reduce mesh-induced analysis errors. An analyst can begin with a coarse mesh and allow adaptive meshing to refine the model only in areas where more elements are needed for accuracy.

- Adaptation is based on any scalar analysis result quantity used as error measure.
 - Independent of analysis program or type of analysis.
 - Strain energy density or strain energy error norm commonly used for structural analysis.
 - Strain energy error provides a global error measure.
 - General gradient calculation.
- Element distortion may be considered as part of the adaptation process.
- Selective node movement, element subdivision, and remeshing provided for shell models.
 - Element subdivision with triangles or quadrilaterals.
- Node movement and selective remeshing provided for solid elements.
- P-element adaptivity provided for linear structural analysis using tetrahedron elements.
- Selective p-order update provided for efficient solution.
- Ability to freeze movement of selected nodes.

- Automatic and adaptive meshing, using a combination of r, h, and p adaptivity, provides fast, accurate analysis with a minimum of user interaction.

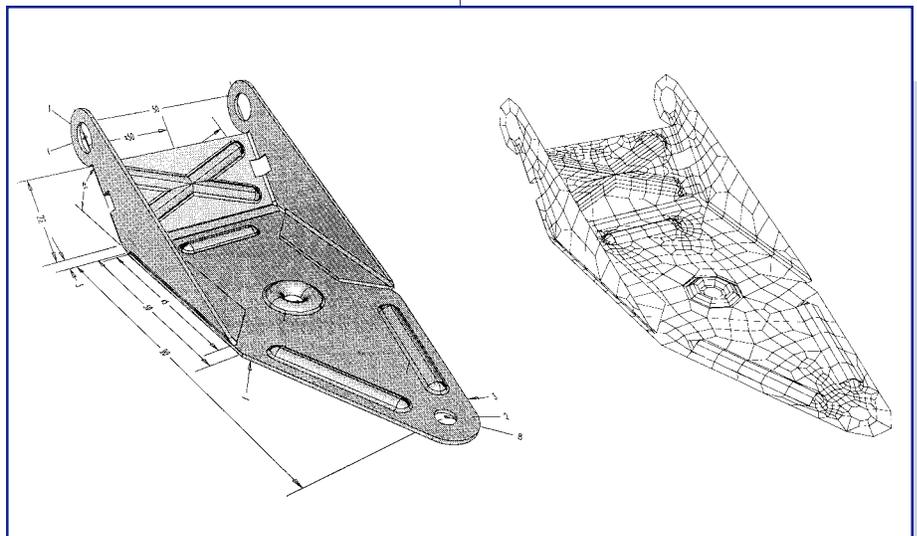
Model Completion

After mesh generation, a number of tasks are needed to prepare a model for analysis. I-DEAS FEM allows completion of the model and verification of its adequacy for your purposes using mathematical and interactive graphic tools.

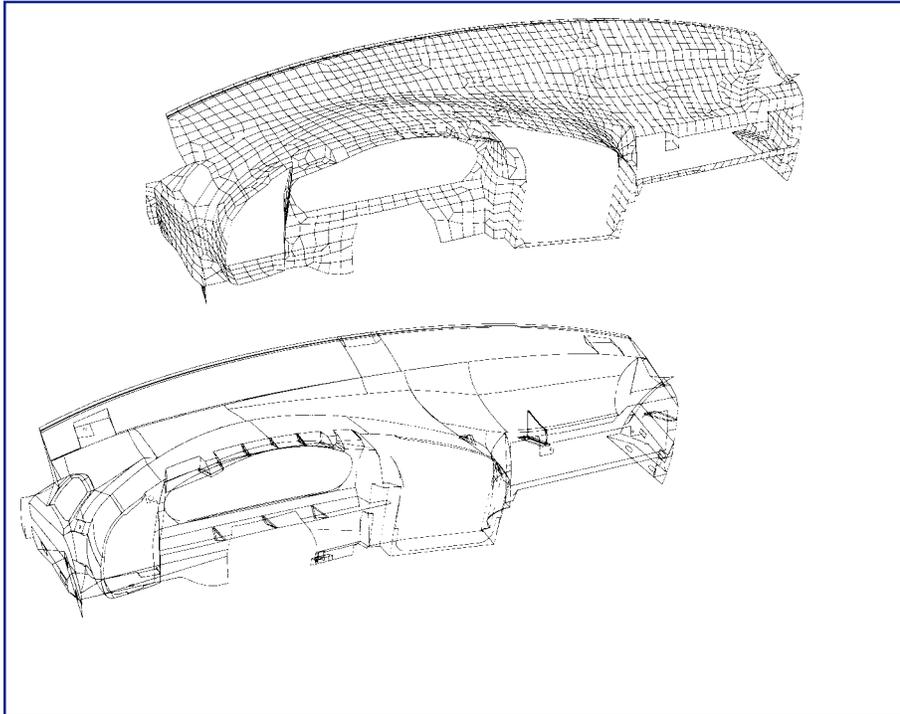
Element Library

A comprehensive library of finite elements lets you perform many types of analyses and modeling in the most efficient manner.

- More than 50 elements are provided including linear and parabolic forms of shells and solids, axisymmetric shells and solids, beams, rods, springs, dampers, masses, and gaps.
 - Scalars and other special elements have unique graphic symbols.
 - P-elements (solid tetrahedron) are supported for linear structural analysis.



Analysis models are created directly from and associated with the I-DEAS master model.



Wireframe and surface geometry from traditional CAD systems can be accessed directly in I-DEAS software and section-meshed for efficient use in analysis models.

Model Development Tools

I-DEAS FEM provides a set of capabilities that assist the user in preparing the finite element model for analysis.

- Dynamic Navigator™ functions used for interaction.
 - Dynamic pre-highlighting of entities.
 - Filter to control active entities highlighted and selected.
 - Pre- or post-command selection.
 - Bounding box and logical (e.g., entity relationship) selection.
- Grouping capability allows logical subsets of a model to be defined and manipulated for display, property assignment, and results calculation.
 - Groups can be defined by interactive screen selection of any and all entities.

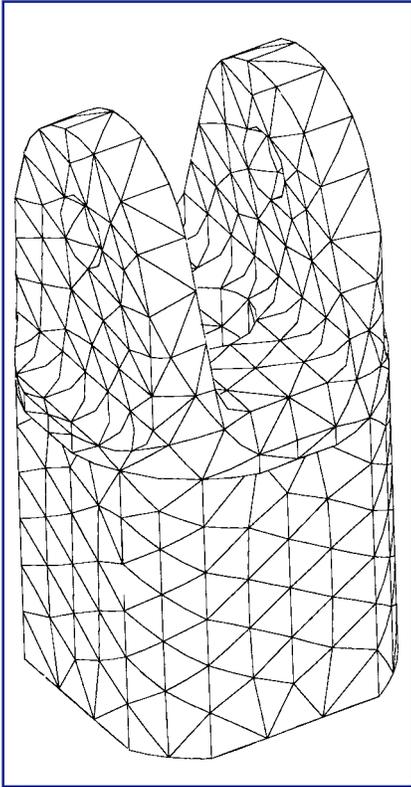
- Groups can also be defined by any entity attribute such as model geometry, color, model checking results, entity relationship (e.g., nodes associated with elements, physical, and material property identification).
 - Set operations can be used to join, intersect, or show the difference between groups.
- Coordinate systems can be used to define desired motions or look at specific results such as radial and tangential stresses.
 - Cartesian, cylindrical, and spherical systems available.
 - Global and local definitions.
 - Material coordinate systems defined for orthotropic and anisotropic materials.

- Material orientation tools for associating element coordinate systems to geometry are provided and are especially useful for composite materials.
- Various display operations control how and which parts of a model are viewed.
 - Entity display switches.
 - Blanking.

Loads and Boundary Conditions

I-DEAS FEM provides extensive capabilities to define loading and boundary conditions to correctly simulate the environment that a part will be subjected to in operation.

- Loads can be defined on and associated with geometry.
 - Point-, edge-, curve-, surface-, or section-based.
 - Independent of mesh but applied to nodes and elements for analysis at user request.
 - Functional variation using mathematical expressions.
 - Surface fit through defined points.
- Restraints defined on and associated with geometry.
 - Point-, edge-, curve-, surface-, or section-based.
 - Standard restraints such as pin, slider, and ball directly available for application.
- Automatic definition of geometry-based contact is provided.
- Constraints and restraints including nodal displacement and master DOF.
- Structural loads.
 - Nodal forces and temperatures.
 - Element face, normal and traction, and edge pressures.
 - Acceleration (gravity, translation, rotation).
 - Ambient and reference temperatures.
- Heat transfer loads.
 - Nodal and distributed heat sources.
 - Face and edge fluxes, convection, and radiation.
- Multiple load and restraint sets defined and stored for use in analysis.
- All loads and restraints displayed with unique graphical symbols.
- Associativity of geometry-based loads and restraints is maintained through design geometry changes.

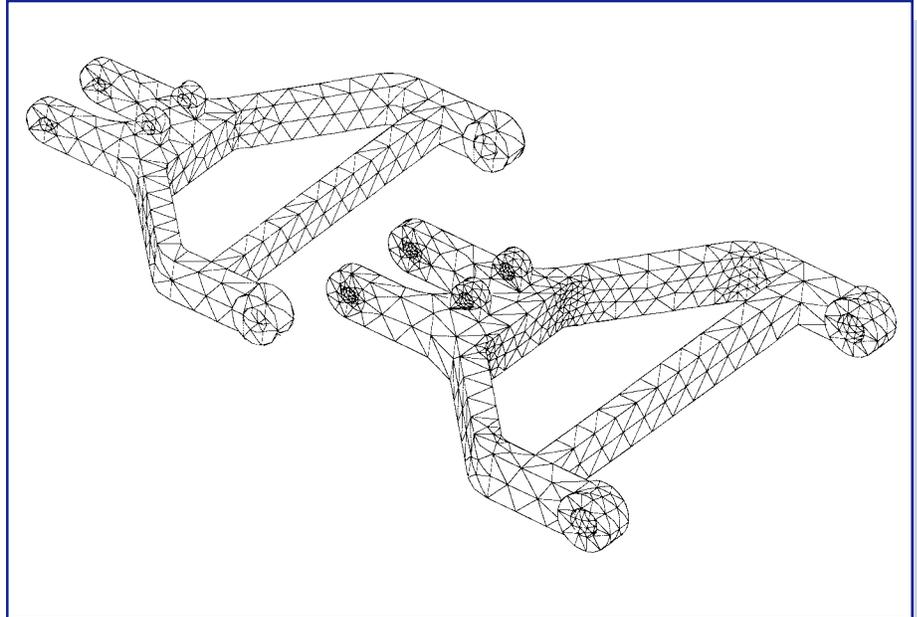


Automatic element generation within a general volume.

Physical and Material Properties

Physical properties such as shell element thickness and beam cross-section properties can be entered interactively and stored in the I-DEAS database.

- Properties extracted from and associated with geometry can be applied to elements at your request.
- Property mask available for entry of data specific to particular analysis programs.
- Standard catalog of beam sections available.



Automatic and adaptive meshing, using a combination of r , h , and p adaptivity, provides fast, accurate analysis with a minimum of user interaction.

Material Repository

I-DEAS Material Data System (MDS) software offers you a very efficient and flexible means of specifying, modifying, and displaying the mechanical, thermal, and rheological properties required for mechanical product design, simulation, and manufacturing applications. I-DEAS MDS is a robust repository for all necessary material characterizations that provide the ability to search for, display, archive, update, and delete material properties and related information.

Properties that do not show a significant dependence on other variables can be specified as constants. Properties that have variable characteristics can be represented by mathematical functions (e.g., constants of a specific stress-strain equation), or by tabular data for various combinations of dependent variables, such as stress, strain rate, and temperature.

User Interaction

I-DEAS MDS is an interactive forms-based capability. Relational operations to list, index, search, archive, track, add, delete, and modify any and all material data are provided. You can access I-DEAS MDS in two ways:

- The interactive shell provides a simple set of forms for entering, modifying, graphing, and comparing material data. This shell also facilitates data management by modifying, deleting, and copying data in and among material data catalogs.
- The material data universal file provides access to external material data catalogs.

The I-DEAS Material Data System offers a unified means of storing, retrieving, modifying, and viewing all material properties in a wide variety of forms. Data displays include:

- Tabular lists
- XY graphs
- Displays of functions or equations

All data reside in material data catalogs, separate from user model files. The I-DEAS Master Modeler software extracts the material properties from one of the I-DEAS material data catalogs to use in design, simulation, or manufacturing applications.

Data Types and Models

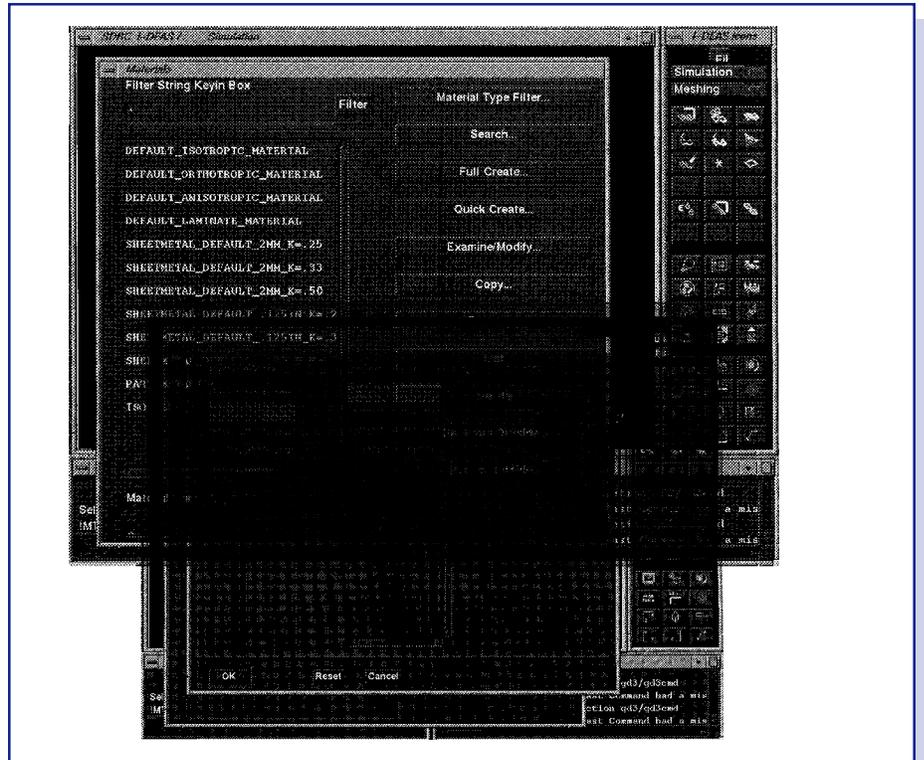
You can enter data in constant, multi-dimensional table, or equation forms. I-DEAS MDS provides flexibility to enter any equational variation or function (with up to five independent variables) capable of analytically representing the material model. For example, standard supported viscosity models include, but are not limited to:

- Power law
- Modified Cross law
- Carreau
- Second order polynomials

I-DEAS MDS also supports multiple domain equations. For example, pressure-volume-temperature data might be fitted to two Tait or modified Tait equations, each applying in a different domain. Isotropic, orthotropic, and anisotropic materials are supported. While 164 material property quantities are predefined, you can add additional properties as required for specialized applications or searches.

Applications

By assigning a material to the part, a wide variety of analyses can be performed. Virtually any material property can be characterized and stored in I-DEAS MDS for use with I-DEAS software or any third-party application.



Interaction with the I-DEAS Material Data System is done through a forms-based graphical interface.

I-DEAS MDS supports a comprehensive set of material properties for structural analysis applications. Elastic, plastic, creep, visco/hyper-elastic, and plastic materials can be completely defined. All of these properties can be defined with temperature dependent stress/strain relationships. The I-DEAS Material Data System software can also support other non-structural analysis property data, such as those for air density, as a function of temperature and altitude.

I-DEAS Model Solution™ Linear and I-DEAS Model Solution Nonlinear software automatically use material properties supplied by the I-DEAS Material Data System. Material properties are automatically written out to any SDRC-supported analysis application, such as NASTRAN, ANSYS, or ABAQUS. A broad cross section of typical and specific mold, coolant, and resin materials needed for plastic process simulation is available. The database of plastic materials is grouped into 18 type classifications (e.g., ABS, PC, PP, etc.) which are further subdivided for various grades and filler contents.

Data supplied with I-DEAS MDS includes required properties for I-DEAS molding simulations:

- Density as a constant or function of temperature.
- Thermal conductivity as a constant or function of temperature.
- Specific heat as a constant or function of temperature.
- Viscosity as a function of strain rate and temperature.
- Softening (VICAT) temperature.
- Density as a function of pressure and temperature (or standard P-V-T equivalent) for packing analysis.
- Young's modulus as a constant or function of temperature.
- Poisson's ratio as a constant or function of temperature.
- Relaxation shear modulus versus time.
- Glass transition temperature.

Flexibility is offered through extensive text description fields and the ability to store and maintain separate versions of any material, property, or field. Any version can be made current before running a simulation. SDRC has published standards and guidelines for plastic-specific materials testing. In addition, non-standard software is available to curve fit discrete data points into any applicable non-linear form which may then be read from formatted files into I-DEAS MDS.

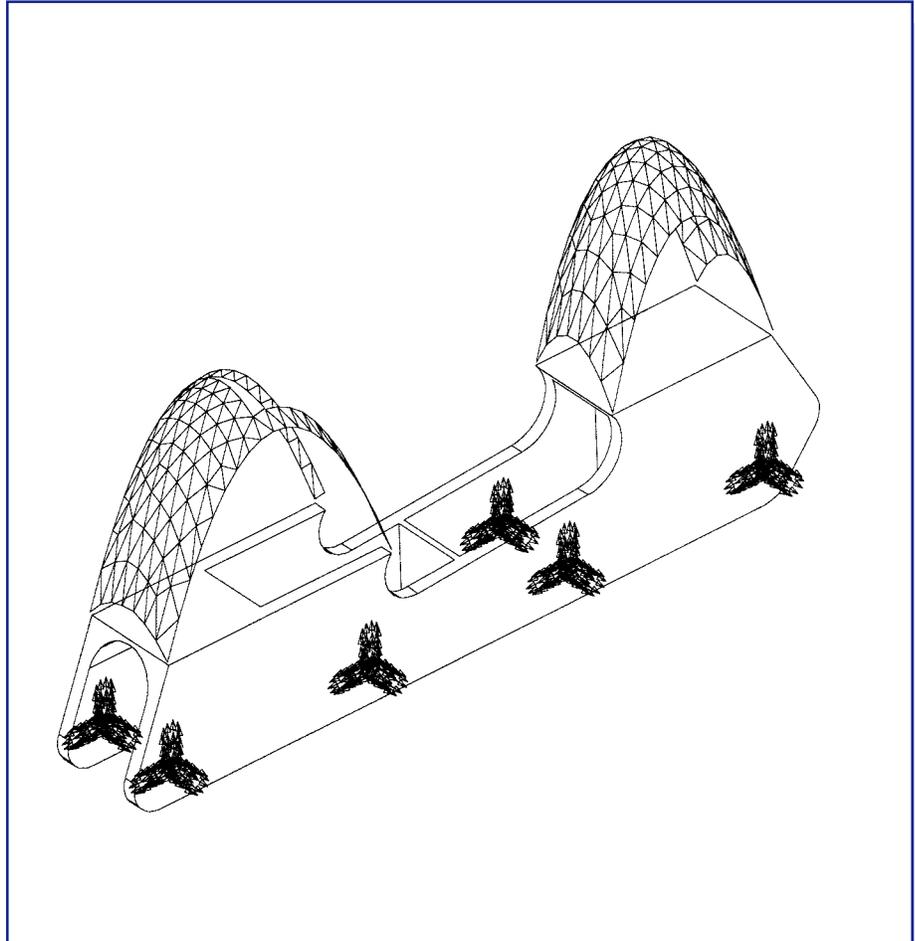
I-DEAS MDS Catalogs

Material catalogs are available for specific applications. See individual specifications or contact SDRC for the latest information on catalog availability.

Model Checking

Analyzing a model with errors can be time-consuming and expensive, and errors are often not detected even after analysis. I-DEAS FEM provides a comprehensive set of graphical and mathematical tools to help verify that a model is complete and correct before you submit it for solution.

- Coincident node and element checks eliminate duplications.



All finite element entities (loads, boundary conditions, mesh definitions, etc.) are associated with geometry so models can be updated automatically with design changes.

- Free-edge and free-face checks avoid unwanted cracks in a model.
- Shrink element display verifies that elements are located properly.
- Coordinate system and element connectivity displays assure correct orientation of beams and shell normals.
 - Element connectivity can be made consistent automatically by specifying seed elements for control.
- Mass and inertia property calculations check physical and material properties.

- Element shape checks verify that elements do not violate limits and can produce accurate results.
 - Distortion
 - Warping
 - Stretch
 - Interior angle
 - Mid-side node location
 - Skew
 - Taper
 - Jacobian
 - Element size
 - Included angle

Analysis

Once a model has been completed and checked, it is ready for analysis. I-DEAS FEM provides direct integration with I-DEAS Model Solution and I-DEAS Optimization™ software.

Thermal/Flow Analysis

I-DEAS provides access to thermal analysis using the finite difference method through I-DEAS TMG and I-DEAS Electronic System Cooling™ software. See the I-DEAS TMG and I-DEAS Electronic System Cooling articles for more information.

Solver Interfaces

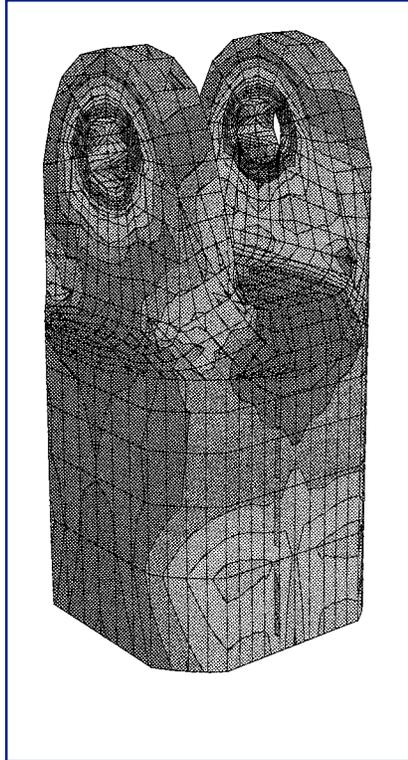
A wide variety of interfaces is available for more unusual types of analysis. SDRC provides direct interfaces, and more than 20 vendors have prepared interfaces to I-DEAS:

- Direct interfaces available to ABAQUS, ANSYS, NASTRAN, and MARC.
- Vendor-written and supplied interfaces to programs for nonlinear structural analysis, acoustic, thermal, electromagnetic, manufacturing process, and flow analysis are provided through the I-DEAS Solutions NetworkSM.
- User interfacing available through ASCII SDRC universal files, or through Open I-DEAS.

Results Review and Post-Processing

For analysis to impact design decisions, results must be presented in an understandable form. I-DEAS FEM provides extensive graphics and data manipulation capabilities that focus on critical data and present it for review and action.

- Supports results from complete library of beam, shell, and solid elements.
- Processes displacement, stress, strain, strain energy, reaction force, kinetic energy, temperature, and heat flux data.
- User-definable results units.



Extensive capabilities for post-processing provide an understanding of results, not just displays of data.

- Contour display for loads and results.
 - Line, color fill, and shaded image modes.
 - Up to 78 contour levels, with user control of levels and intervals.
 - Contours on deformed geometry.
 - Contours on cutting planes through models.
 - Contour animation on top of displacement or of time varying results.
- Deformed geometry display, including animation.

- 2D and 3D function graphing (XY and XYZ).
 - Across model geometry.
 - For time-varying results.
- Criterion display for design criteria evaluation.
 - Numerical display and color coding.
- Arrow display for all vector and tensor data.
 - Single and multiple component arrows.
- Free body diagram extraction and display for model groups.
 - From element force or stress data.
- Data map for text display of model and results data directly on models.
 - User control of data items, legend, fonts, and color coding.
- Overlay of all above displays using consistent color levels.
- Separate groups for calculation and display.
 - Multiple groups to control averaging of results across model and material boundaries.
- Accepts nodal stress tensor data and computes derived values such as Von Mises, principal, and bending values (XYZ components and shear values).
 - Combination and scaling of results for load case combination and “what if” studies.
 - User-defined results computations via mathematical functions for user-defined failure criteria evaluation.
 - Shell surfaces and bending/membrane stresses.
- Selective sorting and tabular reporting of results.
- Numerical reporting of individual screen selected results.
- General annotation capability for creating report-ready plots.
- Allows review of multiple results simultaneously, in the same viewport or in separate viewports.
- Allows you to easily print the display.

FE Visualizer

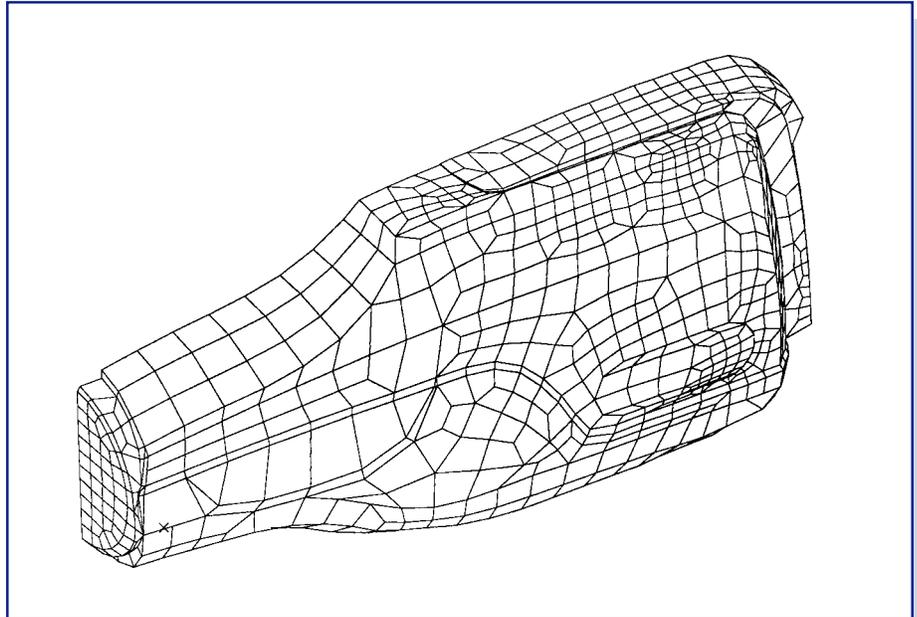
The I-DEAS Visualizer lets you review and make decisions about the results of a finite element analysis. The Visualizer generates displays very quickly, lets you view multiple results simultaneously, and allows you to easily print the display.

Using the Visualizer, you can:

- Create contour displays.
- Create element displays.
- Create arrow displays.
- Review deformed geometry.
- Animate displays.
- Create cutting plane displays.
- Control how data is displayed (for example, data components to use, coordinate system to use, averaged or unaveraged data).
- Create templates of display options.
- Display multiple results simultaneously.
- Display results in multiple viewports.
- Control the quality of the display.
- Review areas where specific result values occur in the display.
- Control colors on the display and the color bar.
- Create stepped or smooth shaded displays.
- View results on beams along with other elements.
- Capture displays to print them, or export them as CGM, PostScript PNG, or VRML files.
- Create SGI or CGM movie files.
- Control the appearance of text and headers in the display.
- Move, copy, or delete results.

I-DEAS Extended Finite Element Modeling

I-DEAS Extended Finite Element Modeling (FEM) software provides added capabilities for building finite element models and reviewing analysis results. It builds on the foundation capabilities of I-DEAS Finite Element Modeling—providing added depth in mesh generation and modeling for nonlinear analysis.



Free, mapped, and manual mesh generation techniques can be combined on a single model—assuring the most productive modeling.

Mesh Generation

I-DEAS Extended FEM adds a selection of mesh generation tools to effectively and efficiently model all structures, including:

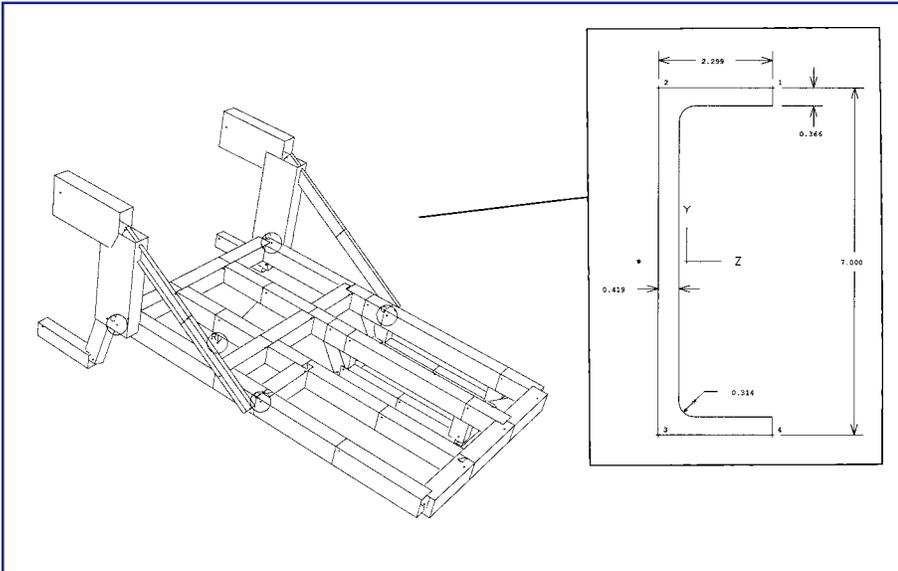
Mapped Mesh Generation

- Generates uniformly shaped elements on simple surface regions and in simple volumes.
 - Linear, parabolic, or cubic quadrilateral or triangular elements on 3- or 4-edged surface regions.
 - Linear, parabolic, cubic hexahedral (brick), or wedge elements in 5- or 6-faced volume regions.
- Each edge, section, or surface for mesh definition can be composed of multiple curves to simplify geometry construction and segmentation.
- Degenerate regions are automatically handled with triangles or wedges.

- User-defined biasing to allow mesh skewing.
- Transitions can be defined for surface meshes merely by specifying a different number of elements on the opposite sides of geometric regions.
- Automatic transition of a number of elements across surface regions is provided.
- Mapped mesh regions developed with partitioning are associative to geometry, so mapped meshes automatically update with geometry changes.
- Mesh density specification is automated through intelligent user prompting.

Bottom-up Mesh Creation and Editing Using Geometry

Meshes, including transitions, can be created directly between wireframe curves without defining surfaces. Surfaces are created as required and resulting models can be stitched into valid solids.



Cross sections "cut" from the part can be directly used in construction of a finite element model.

Direct Mesh Creation/Editing

While free or mapped mesh generation is useful for most tasks, bottom-up creation of nodes and elements can often be the easiest way to build or modify certain types of models. I-DEAS FEM provides tools for building finite element models directly, independent of geometry.

- Extrude and revolve capabilities allow creation of quadrilateral shell elements from an arbitrary set of beams or hexahedral elements from quadrilateral shells. Path options include general curve local coordinate systems and twist.
- Copy and reflect capabilities allow sets of nodes and elements to be replicated.
- A surface coating command generates beams on the edges of shell elements or shells on the faces of solids.
- Nodes and elements can be created and modified individually.
 - Splitting and combining elements.
 - Modifying element connectivity.
 - "Stitching" unconnected meshes.
 - Rotating and moving nodes and elements.

Editing operations including element split, combine, drag, move nodes, and replace nodes, are interactive and maintain geometry associativity when desired. Nodes can be directly created on curves or projected onto a plane.

Combined Methods

Automatic or manual mesh generation methods can be combined on a particular model. This lets you use the most effective approach for each situation.

Model Completion

I-DEAS Extended FEM provides advanced tools for detailed model building and support of nonlinear analysis.

Loads and Boundary Conditions

I-DEAS Extended FEM provides capabilities to define time-varying loading and boundary conditions to correctly simulate nonlinear loading conditions.

- Nonlinear loads and restraints defined as discrete tables of values or functions.
- XY graphs provide verification of loads and boundary conditions.

Relational Database

Access and manipulation of tabular data are provided in I-DEAS software through a relational database. Capabilities include:

- Menu-driven relational operations.
 - Set operations on tables.
 - Table sorting and comparison operations.
 - Column arithmetic.
 - 3D graphing (XYZ).
- Subroutine and ASCII file access.

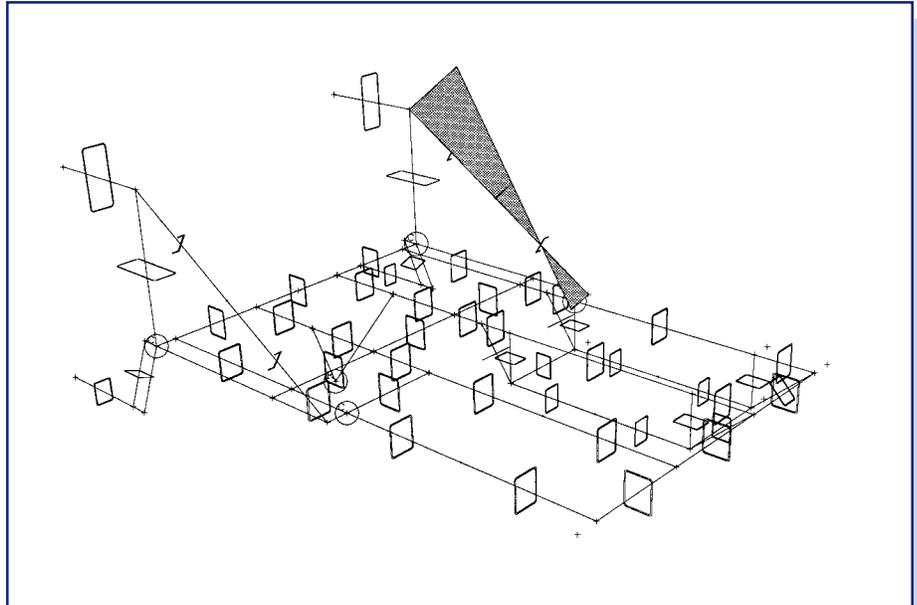
I-DEAS Beam Modeling

I-DEAS Beam Modeling software provides a comprehensive capability for building beam models and reviewing results - simplifying what can be a time-consuming, error-prone task. Many structures, or portions of structures, can be represented efficiently using beam finite elements. Structural elements that are long with respect to other dimensions, such as automobile or truck frames, and stiffeners on panel structures, can be modeled and analyzed most efficiently using beams. Beam elements require cross-sectional property data (moment of inertia, centroid location, torsion constant, etc.), as well as end connectivity geometry for inclusion in a model. Calculating beam cross-section properties for all but the simplest cross sections can be a complex, time-consuming task. In addition, specialized functions for loading are necessary. Reviewing results on beam finite elements also requires more than the standard contouring and display techniques that have become standard for shell and solid finite elements.

Cross-Section Modeling and Analysis

Cross-sectional data are fundamental properties required for use of beam finite elements. I-DEAS provides automated capabilities to define beam cross-sections, calculate required properties, and store them in libraries as part of the I-DEAS database:

- Beam cross-section properties can be defined as an entity in the I-DEAS database. Beam, rod, and pipe elements point to cross-section tables as well as material and physical properties.
- Beam cross-section properties can be defined in a variety of ways:
 - Direct entry into section tables from the keyboard.
 - Dimensional entry of standard shapes including rectangle, box, pipe, tube, channel, angle, zee, hat, tee, wide flange, and general polygon.
 - Dimensioned display of cross-section (including crosshatching) to confirm entry.
 - Modification by selecting a dimension including automatic update of properties.
- Cross-section shape catalogs from industry standards organizations:
 - AISC (American Institute of Steel Construction) 8th and 9th edition.
 - CISC (Canadian Institute of Steel Construction).
 - BSCI (British Steel Construction Institute).
 - JIS (Japan Industrial Standard).
 - Trade ARBED.
 - Geometric display of shapes provided.
 - Several search options are available to assist in locating the proper shape.
- User-defined section catalog.
 - Entry from an external file.
- Arbitrary definition of shapes from points, lines, and arcs.
 - Generated interactively or as a collection of standard sections.
 - Dimensioned and annotated sketch of shape produced.
 - Automatic calculation of section properties for arbitrary shapes through a finite element technique with no user intervention.



Specific beam post-processing capabilities assist in understanding results.

- Direct access to I-DEAS Master Modeler wireframe information is provided.
- Additional model checking capability provided for beams:
 - Ratio of length to depth.
 - Ratio of maximum depth to minimum depth for all beams connecting at a node.
 - Checks applicable to a total model or selected group.

General Beam Modeling

- Mesh generation using standard tools, and automatic remeshing for improved accuracy.
- Definition and management of beam offsets, orientation, and end releases.

Loading

- I-DEAS software provides the ability to define and display beam loading.
- Distributed and concentrated loads along element length.
 - Distributed loads can be constant or can vary.
 - Applied over total element length or over selected part.
 - Loads applied in either global or local element coordinate systems to all six degrees-of-freedom or selected degrees-of-freedom set.

Post-Processing

The one-dimensional nature of beam elements requires specialized tools for effective review of results. I-DEAS Beam Modeling software provides a selection of tools to meet these specific requirements, including:

- Contour displays along beam elements.
- Deformation display using high-order (cubic) interpolation.
- Beam shear and moment diagrams.
 - Display of individual component or combined forces, moments, or stresses.
 - Single or multiple beam elements.
 - Linear, curved, and tapered beams.
 - Any beam loading defined included in calculations.
- AISC code checks (8th and 9th editions).
 - From single or multiple element force data sets.
 - Ability to check all or individual elements.
 - Detailed tabular reporting.
 - Criterion plotting allows display of elements violating selected criteria.
 - Failed beams optionally written to group for display.
 - Individual beams checked at any location along beam.
 - Location of maximum stress identified.
 - Listing of unity ratios.
- Display of stress distribution on cross sections.
 - Any selected location along beam length.
 - Contour or arrow plot form.
 - Line display for standard thin-walled shapes.
 - Torsional and warping stresses are also included.
- Beam stress results:
 - Available at up to 12 user-defined points on cross section (depending on cross-section shape).
 - Can be sorted across multiple load cases.

Prerequisite

None

For More Information

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