Simulation Solution Set

I-DEAS® Model Solution™ Linear

I-DEAS Model Solution Linear software is a general finite element analysis program for linear structural, thermal, and flow analysis. It is an integrated part of I-DEAS, which means no additional input files need to be created just for the analysis. All solution control parameters and output selections are forms-based, making it easy to set up a solution. This reduces the overall time to do analysis and helps assure accuracy by maintaining data integrity between model building, solving, and results interpretation. User interaction with I-DEAS Model Solution Linear is through three easily understood forms.
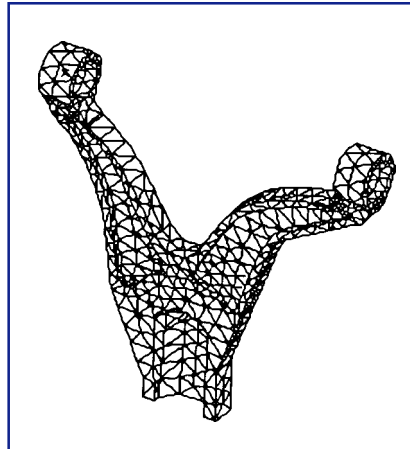
Once the desired setup is made, a single button executes the solution. I-DEAS Model Solution Linear uses proprietary, state-of-the-art numerical methods to minimize compute time. To make FEA usable for general structures, I-DEAS Model Solution Linear has a comprehensive set of options for:
- Elements
- Modeling
- Analysis
- Results

Elements

I-DEAS Model Solution Linear offers the following comprehensive set of finite elements to model a wide variety of structures, all with evolving state-of-the-art mathematical formulations to minimize computer time and maximize accuracy. In addition to the industry standard beam, shell, and solid elements, I-DEAS Model Solution Linear supports a p-version tetrahedron element.

With the p-element, adaptive analysis using I-DEAS Model Solution Linear will automatically increase the order of elements in the model during the solution process based on a specified level of accuracy. The level of accuracy is determined using the convergence of strain energy approach, a state-of-the-art error estimator. Adaptive analysis using conventional and p-elements allows many types of analysis to be automated with confidence in results.

I-DEAS Model Solution Linear software has a comprehensive set of finite elements including a p-element tetrahedron shown here. Automated adaptive analysis is provided.

Modeling

I-DEAS Model Solution Linear software offers the following modeling options used with the comprehensive set of elements and analysis types:
- Coordinate systems for nodes can be Cartesian, cylindrical, or spherical. Local coordinate systems can also be used for individual nodes or groups of nodes. Different coordinate systems can be used to define node locations, apply loads and restraints, and evaluate results.
- Material properties can vary for individual elements or groups of elements. These properties can describe standard isotropic, orthotropic, or general anisotropic materials. Using I-DEAS Material Data System™, you can also select materials based on a variety of mechanical property criteria. Laminate composite material properties can be created in I-DEAS Laminate Composites software, and the resulting model solved in I-DEAS Model Solution Linear. Material properties that vary with temperature can also be solved.

- Physical properties are required to complete the modeling of beam and shell elements. Cross-sectional properties for the beams can be automatically generated with I-DEAS Beam Modeling. Thickness of the shell elements can be uniform or variable at each corner node.
- Loading conditions are available to support many types of analysis:
  - Specified displacements.
  - Forces and moments at nodes.
  - Forces and moments on beams can be a constant or a linear distribution, and can be applied as axial, shear, and bending loads.
  - Face pressures on shell surfaces or solid faces.
  - Edge pressures on shell edges can be a constant or a linear distribution, and can be applied as membrane, in-plane shear, out-of-plane shear, and bending loads.
  - Temperatures at nodes, at elements, on surfaces (top and bottom), as ambient, and as reference.
  - Heat fluxes and convections applied to faces or edges.
  - Concentrated and distributed heat sources.
  - Mass effect loads can be applied as gravity, linear acceleration, or rotational velocity.
- Boundary conditions complete the list of modeling options by providing the following:
  - Restraints are applied to nodes as zero or nonzero displacements in any degrees-of-freedom in individual displacement coordinate systems.
  - Kinematic degrees-of-freedom allow for an analysis solution of an unrestrained (free) structure.
  - Coupled degrees-of-freedom constrain two nodes as the same motion. This is useful in modeling cyclic symmetric sectors and spot welds.
  - Multi-point contsraint equations provide a way of relating the motion of several nodes. Equations can include various nodal degrees-of-freedom with scale factors.
  - Master degrees-of-freedom for Guyan-reduction dynamic analysis.

• Contact surfaces (with surfaces offsets and friction values) define the surfaces (and the associated finite element mesh) over which contact will be computed.

Analysis

The following types of analysis are available. The default is a direct matrix solver based on an advanced sparse matrix strategy which minimizes disk space by taking maximum advantage of matrix sparsity. An alternative is an iterative solver which is especially effective on large problems (up to 100 times faster).

• Verification analysis checks for model completeness and calculates the total mass and inertia of the model and a vector resultant sum of any applied loads. Estimates of the solution time and storage space requirements are made for the full solution.

• Heat transfer analysis solves for steady-state temperatures due to convection, conduction, and heat generation. Material properties can be temperature dependent.

• Static analysis solves for linear, static stress, and deflection results when mechanical loads are present. Temperature results from a heat transfer analysis can be included with the mechanical loads. Static solutions can include node-to-node gap elements and contact surfaces.

   • Sparse Matrix Solver is a general purpose solver.

   • Iterative Solver is an exclusive static solver.

• Geometry-based contact allows automatic definition of and solution for contact problems, including friction effects, directly on master model geometry. Dissimilar meshes across contact regions are automatically accommodated. Contact between all combinations of shell and solid elements is supported. Contact analysis with P elements is provided. Accuracy is assured by direct use of geometry for contact determination.

• Dynamic analysis solves for natural frequencies and mode shapes of either unrestrained or restrained structures. Stress stiffening and spin softening effects can be included. Three options are available for efficient solutions:

   • Lanczos solver is the most efficient solution option for large numbers of modes. It is an accurate technique for solving many modes over a selected frequency range. No user-specified solution parameters (e.g., Guyan Masters) are necessary for this method. The Lanczos solver also has frequency shifting to find all natural frequencies or mode shapes within or near a user-specified range.

   • Simultaneous Vector Iteration (SVI) is also an efficient and accurate technique to solve for many modes over a selected frequency range. No Guyan reduction is done, and no master degrees-of-freedom are required. Selection of initial solution vector can be user-specified or program-selected. The SVI solver calculates the constraint mode representation with residual flexibility. It also has a frequency shifting capability that will find all natural frequencies or mode shapes within or near a user-specified range.

   • Jacobi solver with Guyan reduction is an efficient technique for calculating primary modes of large models. The accuracy is partially determined by your selection of master degrees-of-freedom. The Jacobi solver calculates the constraint mode representation. I-DEAS FEM automatically selects masters.

• Response dynamics solves include constraint modes for enforced motion analysis in I-DEAS Durability and I-DEAS Response Analysis.

• Dynamic forced response solutions are also available through the I-DEAS Response Analysis option. You may simulate steady vibration, transient events or evaluate response to statistical (PSD) descriptions of excitation.

• Potential flow analysis solves for pressures and velocities in a defined solution domain using the incompressible inviscid equations of fluid mechanics.

• Eigenvalue buckling analysis is a technique for predicting the loads at which the onset of structural instability will occur.

• Restarts for new loads are automatic.

• Adaptive analysis using automatic remeshing or p-adaptivity can be executed directly in I-DEAS Model Solution Linear. You specify the allowable convergence error before starting the solution, and the software automatically updates the model and iterates until the desired convergence is achieved.

Solution Monitors

Depending on the solution type, one or more solver monitors are displayed as your solve progresses. The solver monitors graphically display the progress of the solution and estimate the time required to complete the solve.

• Iterative Solver Convergence
• Sparse Matrix Solver
• Contact Analysis Convergence
• Nonlinear Convergence
• Eigenvalues Extraction
• Strain Energy Convergence

At any time while the solve progresses, you can cleanly terminate the solve.

Results

The following results of the selected analyses are automatically available for viewing in I-DEAS Finite Element Modeling™ software and can be written to a log file.

• Displacements
• Stresses
• Strains
• Strain energies
• Reactions
• Contact pressure
• Mode shapes
• Natural frequencies
• Modal mass properties
• Temperatures
• Heat fluxes
• Velocities

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

Quality Assurance
I-DEAS Model Solution Linear is in productive use at thousands of sites around the world. SDRC tests I-DEAS Model Solution Linear against internationally recognized standards (NAFEMS, Soc. Francais de Mecanicians). An extensive suite of test problems, with closed-form solutions, is used. These test results are available on request (Model Solution Verification Guide, P_50049).

Efficient
I-DEAS Model Solution Linear employs state-of-the-art, sparse matrix and iterative solution algorithms. Advanced memory management and multiple disk access capabilities are combined to take full advantage of computer hardware environments. A preconditioned Conjugate Gradient iterative solver is especially effective for very large problems. Improved analysis cycle times mean that more design iterations can be performed in a given time.

Added Power for Experienced Analysts
Direct access to I-DEAS Model Solution Linear matrix processing is available through I-DEAS programming capabilities. Custom solution sequences can be created to do special analysis or general numerical methods using a standard matrix math library. Direct matrix input and output completes the package for virtually unlimited customization.

Element	
Beam: linear	
Beam: curved	
Beam: parabolic	
Beam: tapered	
Rod	
Solid: linear tetrahedron	
Solid: linear wedge	
Solid: linear brick	
Solid: parabolic tetrahedron	
Solid: parabolic wedge	
Solid: parabolic brick	
Shell: linear triangular	
Shell: linear quadrilateral	
Shell: parabolic triangular	
Shell: parabolic quadrilateral	

Element	
Axisymmetric shell: linear	
Axisymmetric solid: linear triangular	
Axisymmetric solid: linear quadrilateral	
Axisymmetric solid: parabolic triangular	
Axisymmetric solid: parabolic quadrilateral	
Plane strain: linear triangular	
Plane strain: linear quadrilateral	
Plane strain: parabolic triangular	
Plane strain: parabolic quadrilateral	
Plane stress: linear triangular	
Plane stress: linear quadrilateral	
Plane stress: parabolic triangular	
Plane stress: parabolic quadrilateral	
2-D shell: linear triangular	
2-D shell: linear quadrilateral	
2-D shell: parabolic triangular	
2-D shell: parabolic quadrilateral	

Element	
Constraint	
Gap: node to-ground	
Gap: node-to-node	
Lumped mass	
Rigid	
Rigid bar	
Spring: node-to-ground rotational	
Spring: node-to-ground translational	
Spring: node-to-node rotational	
Spring: node-to-node translational	
Non-conducting elements	

I-DEAS Model Solution Linear software supports a wide variety of elements for complete modeling.

I-DEAS Optimization™
I-DEAS Optimization software helps you use analysis results to directly drive design improvements. Structural optimization can help weed out poor design concepts as well as identify and improve promising ones. This makes you more efficient because you can focus on the high-potential design concepts, and not waste time on those with little chance for success. It can also be used for fine-tuning designs by minimizing weight or cost, while ensuring they will meet all structural performance requirements.

Optimization Redesign
Optimization Redesign uses the finite element analysis (FEA) approach to simulate structural performance. The current design is represented in I-DEAS Finite Element Modeling, and can be linked to parts designed with I-DEAS Master Modeler™ software. Optimization Redesign only requires you to define the following information:

•Design parameters are the physical features to be modified to improve performance. Physical features may be an, part dimension (e.g., hole diameter), entries in a finite element physical property table (e.g., thickness), or dimensions on beam cross sections. Limits on allowable modifications can be specified.
•Design limits are physical quantities which measure performance such as stress, displacement, or natural frequency.
•Design goals define the objective of the optimization. You can minimize a design goal (e.g., mass, stress, displacement).

Once all these entities have been specified, Optimization Redesign will modify the design parameters to minimize the design goal, while staying within the design limits–resulting in an improved design.

Geometry-Based Shape Optimization
Optimization Redesign provides true geometric shape optimization through direct connection to I-DEAS Master Modeler. Design intent captured via the variational geometry network in I-DEAS Master Modeler can be used to specify and control allowable shape changes. Optimization Redesign will then determine the best combination of geometric design parameter modifications to meet the design goal. Complete associativity between the geometric and finite element models means suggested shape changes from Optimization Redesign can be automatically applied to parts in I-DEAS Master Modeler. The library capability of I-DEAS Master Modeler is used to manage updates through the design iteration process. Shape optimization generally involves a number of design iterations with modified geometry at each step. Remeshing of a part in I-DEAS Master Modeler during this iterative process is done automatically.

Size Optimization
Sizing optimization analysis employs an iterative strategy of changing plate thickness and beam cross-section dimensions to meet the design goal while staying within the design limits. Size and shape optimization may be performed simultaneously. The following design parameters are available for size optimization:
•Physical properties for certain elements may be used as design parameters. Elements may be grouped together, and a single design parameter can be specified for the physical property of that group. In certain cases (e.g., plate thickness), design parameters may be specified to have upper or lower limits to bound an allowable range. The following physical properties can be design parameters:
  •Thickness of shell elements.
  •Radius and area of rod elements.
  •Cross-section dimensions of beams.

For beam elements, you can specify a beam dimension change as a function of other beam dimensions in the cross section. For example, the height of a box section could be specified to be half the width of the section. Up to six beam dimensions can be controlled by a beam design parameter. Beam offset can also be included.

Adaptive Analysis
P-adaptive analysis can be used in the optimization loop. By turning on the adaptive option, you can assure that a converged analysis solution is automatically achieved at each design iteration.

Design Limits
Design limits help capture the functional requirements of the design. Multiple design limits can be considered simultaneously including multiple analysis types. I-DEAS Optimization software can apply design limits to the following performance criteria:

•Material strength is specified as a maximum Von Mises stress. Multiple material strengths can be used in the same analysis for different loading conditions or different regions of the structure. Some loading conditions (e.g., high cycle fatigue loads), may have a lower allowable stress than other loads (e.g., ultimate load). A criterion can be specified for membrane and bending in shells.
•Allowable displacement may be specified as positive or negative displacement limit at discrete locations. This can be used to establish stiffness performance for the structure or a design fit requirement. Multiple allowable displacements can be used in the same analysis for different loading conditions or different regions of the structure.
•Dynamic vibration performance can be modified by changing natural frequency values. Ranges can include "greater than" or "outside of" limits. Many applications want to move the natural frequencies of the structure away from excitation frequencies like engine speeds or bump loads. Multiple ranges can be included in the same analysis.

Design Goal
I-DEAS Optimization allows you to select the best design goal for your specific needs:
•Mass
•Stress
•Displacement

Parameter Studies
Engineers often want to develop design curves of results to study the effects of design changes. I-DEAS Optimization gives the user the ability to interactively define sets of design variations, automatically proceed through the required analysis steps, then plot the results in an easily understood form.

Sensitivity Analysis

Sensitivity analysis provides design and engineering insight on the degree to which a change in each design parameter influences a change in the structural performance. The design parameters that have the greatest effect (highest sensitivity), lead designers and engineers to change those aspects of the design that will most cost-effectively achieve the desired performance.

Sensitivity information is available for the design parameters noted. Results of the sensitivity analysis are one sensitivity value per design variable per design limit. This value represents approximately how much the design limit value will change given a change in the design parameter.

Sensitivity Reanalysis

Sensitivity analysis calculates and stores these sensitivities. You can examine them, then modify design parameters and reanalyze the model for stresses and displacements based on the modified parameter values.

Sensitivity analysis stores sensitivities in I-DEAS results sets, which you can use in other applications or for sensitivity reanalysis in the Durability task. You do not have to specify limits or a design goal. The system automatically generates implicit design limits based on the current model.

Sensitivity analysis supports geometry or FE-based linear statics solutions.

Physical Properties
•Thickness of shell elements
•Stiffness of spring elements
•Mass of non-structural masses

Beam Sections
•All beam cross-section dimensions
•Area
•Moment of inertia - Y
•Moment of inertia - Z
•Shear area ratio - Y
•Shear area ratio - Z
•Torsional constant
•Eccentricity - Y
•Eccentricity - Z

Material Properties

These include isotropic, orthotropic, or anisotropic materials:
•Young's modulus
•Poison's ratio
•Mass density
•Shear modulus
•Coefficient of thermal expansion

Geometric Shape

These variables are defined by the I-DEAS Master Modeler variational geometry network or nodes which represent a specific geometric feature.

Optimization Solver

The optimization solver is integrated with I-DEAS Model Solution Linear software. All of the different types of loads, including thermal loads, available in I-DEAS Model Solution Linear for structural analysis, can be used in I-DEAS Optimization. Multiple loading environments can be analyzed at the same time.

I-DEAS software uses evolving state-of-the-art numerical solution techniques for fast, accurate solutions of the inherently nonlinear mathematical optimization problem. I-DEAS Optimization generally provides useful redesign information in three to five iterations. Two numerical algorithms are available for the most efficient solution:
•Fully stressing is a useful optimization technique when material strength is the only performance criteria. Sensitivities are not evaluated.
•Mathematical programming optimizes the design goal while satisfying specific design limits. This method linearizes the constraints in the problem and then solves the optimization problem using the sensitivities at the current design point. I-DEAS Optimization software also incorporates many techniques to minimize solution times and improve accuracy and performance. Active Set Strategy is implemented into the algorithm to ensure that unnecessary constraints are ignored during the solution process. A proprietary solution algorithm is used to increase stability by increasing the design envelope in which solutions can be found.

Optimization Results

Standard finite element results are available for any design iteration or loading condition. All of the post-processing capabilities of I-DEAS Finite Element Modeling software can be used to display and evaluate these results. I-DEAS Optimization includes additional post-processing capabilities for general sensitivities and for displaying:
•History of design parameters
•History of design goal
•History of design limits
•Reshaped design at each iteration

Optimization results may be displayed or written to a log file. Optimization results can be interrogated after any iteration. I-DEAS Optimization provides the capability to then automatically restart the optimization process from that point.

Prerequisites
Core Simulation

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