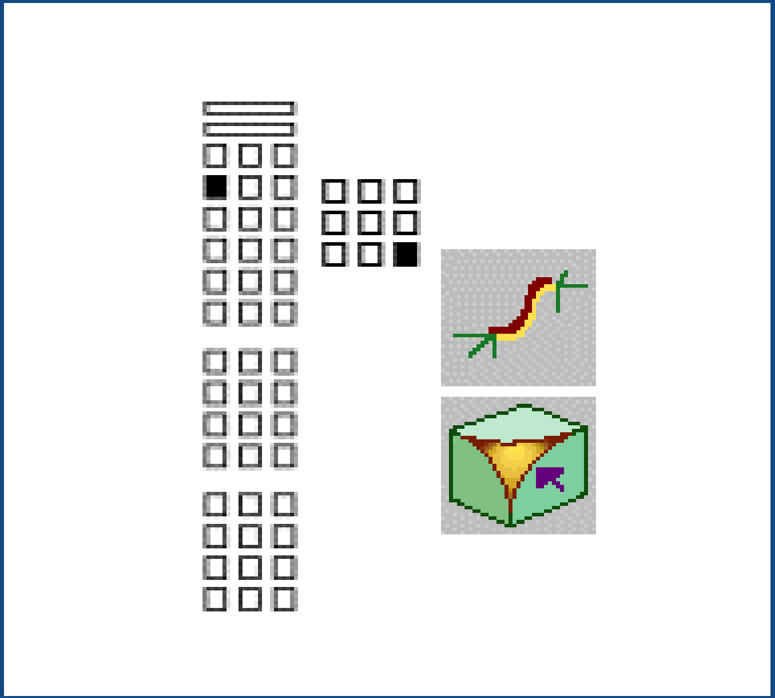
What's New I-DEAS 10 NX Simulation

General Enhancements

- Wireframe Tools Subpanel
- Laminates Thickness
- Control Model Locking by Results Sets

Additional Capabilities Added to the Wireframe Tools Subpanel

- The *Tangent Curve* and *Blend Fillet* icons have been added to the *Wireframe Tools* subpanel within the Meshing task. This allows you to directly access these useful wireframe modification commands without changing to the Modeling task.



Laminates

- In this release, you can now specify a bottom fiber distance for laminates. A laminate's bottom fiber distance is defined as the distance from the reference plane to the laminate's bottom fiber. In the FE model, the bottom fiber distance is the distance between a node and the bottom surface of a shell element.

- When you create a new laminate, the software automatically sets the bottom fiber distance to a value of -T/2, where T is the thickness of the laminate. You can change this default value when you create or modify the laminate.

New Param File Entry to Control Model Locking by Results Sets

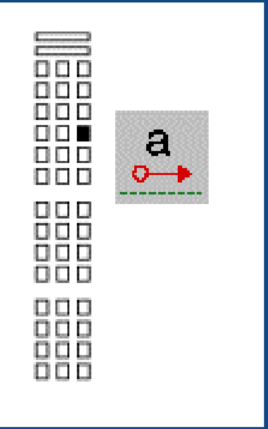
- In previous releases, the software simply displayed an informational message if your model was locked by results sets. Once you'd dismissed the message, you had to go to Model Solution, Post Processing, or the Visualizer to manually delete those results sets. In this release, you can use a new param file entry to control whether the software should also give you the option of automatically deleting those results sets:
- *fem.ModelLockDeleteResults: #* where # is as follows:
  - 0 : Never display the option to allow the automatic deletion of result sets.
  - 1 :Prompt for the automatic deletion of results sets when only pre-processing results sets exist. Pre-processing results sets include element thickness results, element quality check results, and, beginning in this release, boundary condition results sets. This is the default setting for this parameter.
  - 2 :Prompt for the automatic deletion of results sets when either pre-processing or analysis results exist.

What's New Overview – Pre-Post Processing

- **Boundary Conditions Updates**
  - Accelerations – UI, graphics, units
  - Boundary Conditions Set Management
  - Model Check Sum – total applied load
  - Contact Updates
  - Data Surface on Sections
- **FEM Point Connectors**
- **FE Model Checks**
- **Large Model Abstraction and Meshing Enhancement**
- **Frozen Mesh Preview**
- **Mapped Mesh Around Holes**
- **Interference Check/Fix**
- **Section meshing Updates**
- **FE Model Documentation**
- **Visualizer**

User Interface Improvements for Defining Acceleration Loads

- You can now use the new *Accelerations* icon and form to define gravity, translational accelerations, as well as the rotation term for total acceleration.
- Acceleration loads are now represented by temporary graphics while you're working on the *Accelerations* form
- In addition to these improvements, you can also use the following new param file options to control the default colors for the acceleration graphic loads:
  - Fem.gravity_color
  - Fem.translational_acceleration_color
  - Fem.angular_velocity_color
  - Fem_angular_acceleration_color

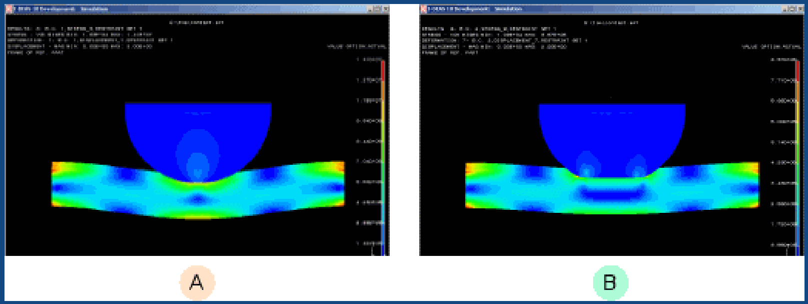


Boundary Conditions Set Management User Interface Improvements

- This release includes a newly redesigned *Boundary Condition Set Management* form that allows you to easily manage all boundary conditions sets in your model file. For example, with this form, you can now:
  - create new boundary condition sets
  - list all existing boundary condition sets according to their analysis type
  - rename or delete selected boundary condition sets
  - list the contents of one or more selected boundary condition sets as well as the names of any solution sets that reference them
  - copy an existing boundary condition set to create a new set

Boundary Condition Updates

- Ability to Define Data Surfaces on Sections-on-Meshes
  - You can now define data surfaces on sections-on-meshes. In previous releases, data surfaces weren't supported for models that contained sections-on-meshes. Contact User Interface Modifications
- In this release, the user interface for defining a contact search distance for contact sets and contact pairs has been modified. Now, you can use the new *Search Dist. Between* option on the *Contact Set* and *Contact Pairs* forms to have the software create contact elements within a specified range.
  - For example, in (A), we defined a search distance between 0 and 1. In (B), we defined a search distance between 1 and 2. Notice the difference the results.



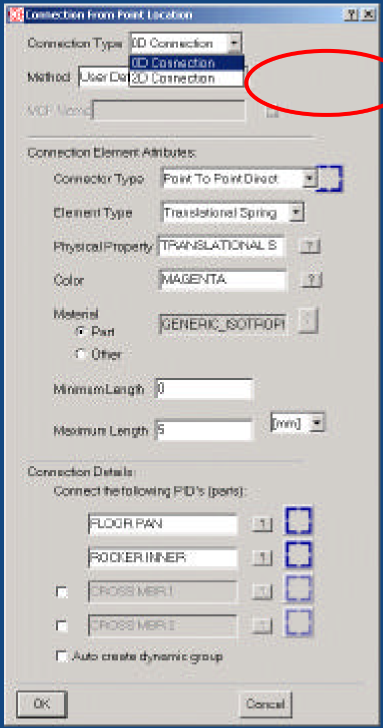
- Ability to Define Geometry-based Contact Regions and Pairs on Sections
  - You can now define both geometry-based contact regions and geometry-based contact pairs on sections-on-surfaces as well as sections-on-meshes. In previous releases, these types of boundary conditions weren't supported for models that contained sections.

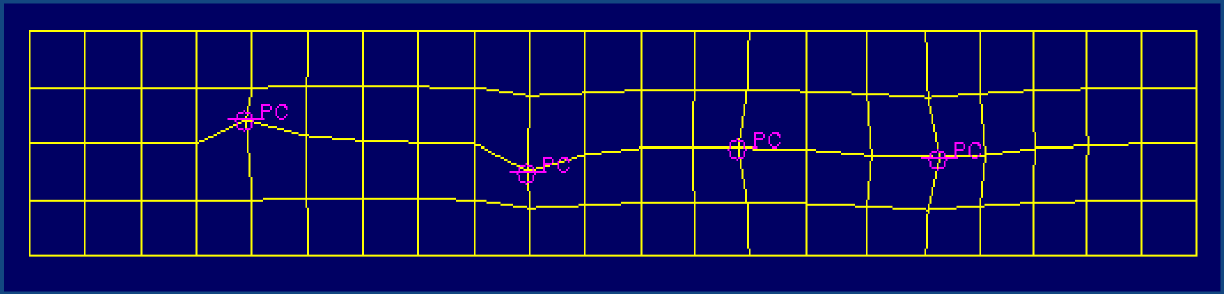
New Point-to-point Direct Methodology Available

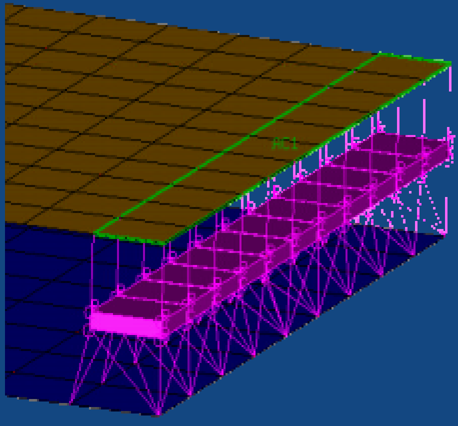
- In I-DEAS 9, you could create FEM point connectors using either the linear brick method or the point-to-point method. The point-to-point method modeled spot welds using 1D elements, such as rigid bars and gaps. These 1D elements were then attached into the mesh on the connected components using rigid and/or constraint elements. To clarify its use, this point-to-point method has been renamed *Point-to-Point Indirect*.
- This release introduces a new, *Point-to-Point Direct* method, which allows you to directly connect the 1D element into the component meshes. If necessary, when you generate the FEM point connectors, the software forces a local remesh in the component meshes around the endpoints of the 1D elements. You can use new options on the *Point Connector Tolerance* form to control aspects of the local remesh, such as how many layers of elements are affected.
- In previous releases, the only way to model a spot weld as a 1D element connected directly to the component meshes was to carefully align the nodes on the components within the assembly FEM and then manually connect them using either geometry-based other elements or elements created from point proximity.

FEM Point Connectors

- Based on the expressed need by Ford and Jaguar to expand the existing FEM Connection utilities
- Point-to-point direct connectors
  - Support a true ACM1 entity.
  - Connection type: 0D
- Area connectors
  - Support adhesives using ACM1 methodology
  - Connection type: 2D

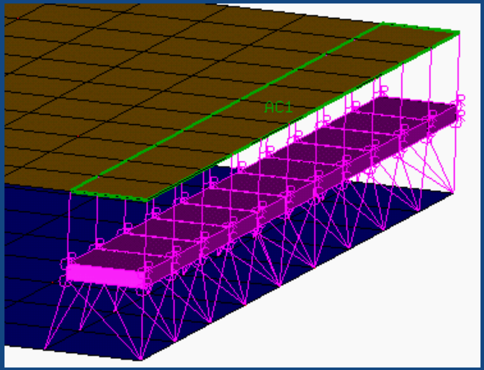


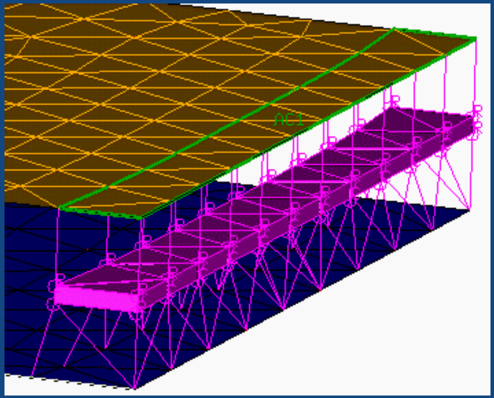




New FEM Area Connector Entity for Modeling Adhesive Connections

- In this release, you can now define FEM area connectors to model adhesive connections between the mating flanges of components. With FEM area connectors, the software uses a series of either linear brick or wedge elements to define the adhesive. The centroid of the brick or wedge element is located at the mid-point between the two flanges. The bricks or wedges are then connected to the meshes on the flanges using I-DEAS constraint elements (Nastran RBE3 elements) and/or rigid elements (Nastran RBE2 elements).
- In this example, FEM area connectors have been used to create an adhesive connection between two linear quadrilateral meshes that do not match. The nodes on the meshes are joined by linear brick, rigid bar, and constraint elements generated from the FEM area connectors.
- In this example, FEM area connectors have been used to create an adhesive connection between a linear triangular and a linear quadrilateral mesh. The nodes on the meshes are joined by linear wedge, rigid bar, and constraint elements generated from the FEM area connectors.
- FEM area connectors are defined and generated very similarly to FEM point connectors. To define a FEM area connector, pick *Connection from Point Location* on the *Define Connector Elements* subpanel. Then, select 2D as the *Connection Type*. You can then use the options on the *Connection from Area Location* form to define the area connectors. For example, you can use the *Thickness* option to define the thickness of the adhesive connection



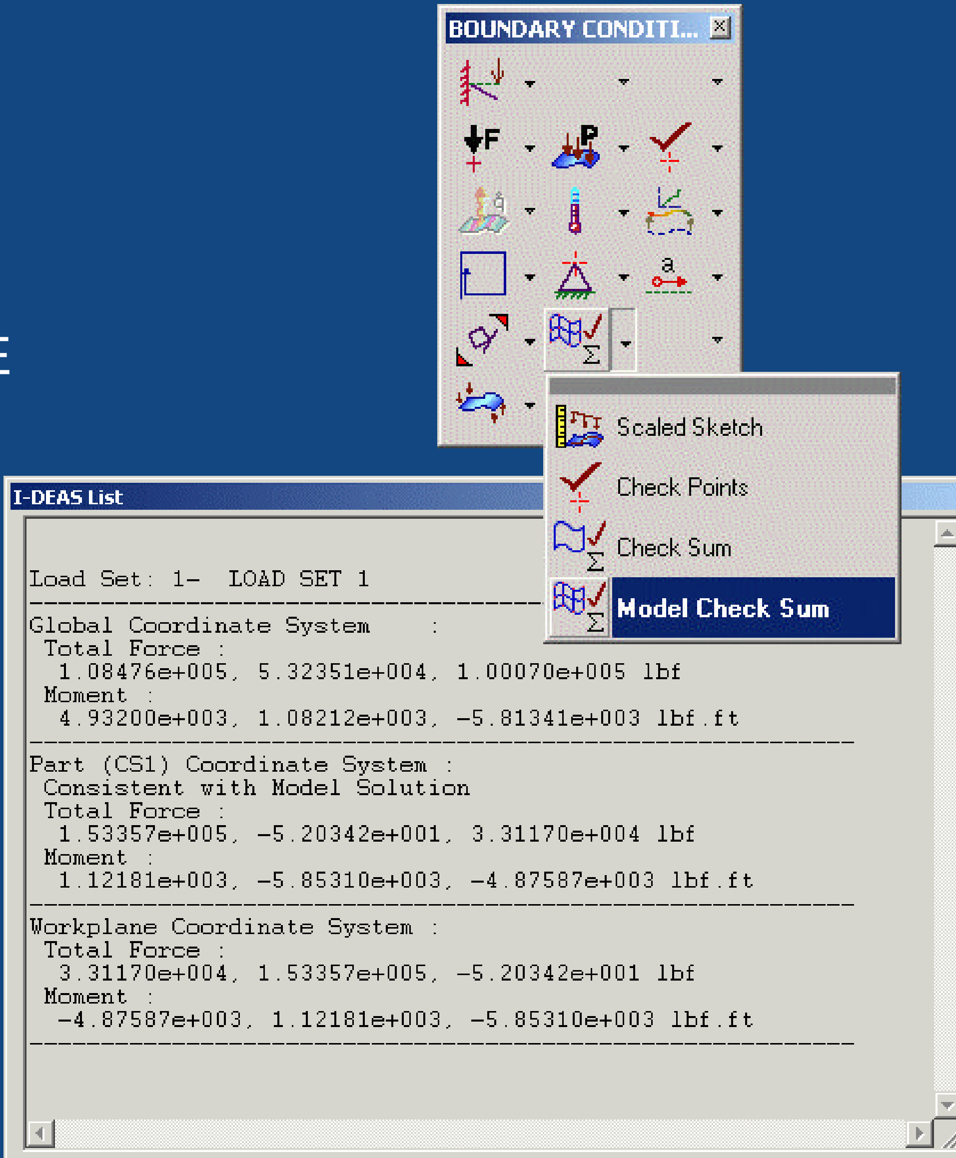


New Model Check Sum Command

- The new *Model Check Sum* command lets you calculate the total applied forces and moments for selected load sets. This allows you to verify that the loads have been correctly applied to the model.
-
- You can use *Model Check Sum* to simultaneously sum both geometry- and fe-based load sets in any coordinate system. In previous releases, you could only check geometry-based loads. Additionally, you had to sum each type of load set by hand to obtain the total load vector.

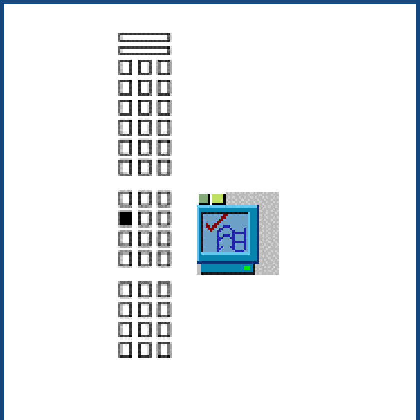
FE Model Check Sum

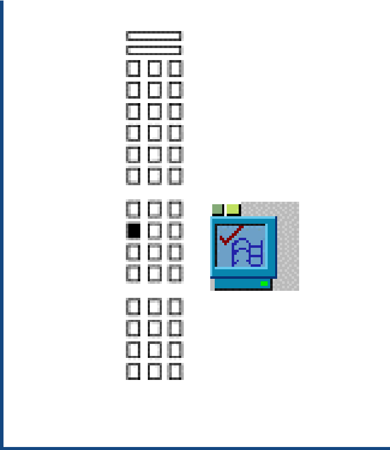
- Pre-processing check of total applied mechanical loads
- Includes geometry based and FE based loads
  - Geometry must be meshed
- Based on Model Solution Total Applied Loads calculation
- Output reported to List in global, part, workplane, and local CS (optional)



New Model Checking Display Capabilities

- Model checking is a new persistent display capability that allows you to visually validate various aspects of your model, such as element quality and the application of certain types of boundary conditions. For most display types, when you turn model checking on, the software colors the elements in your model based on the type of display and criteria you select. When you turn model checking off, the software displays elements in their original colors.





Element Quality Checks Form Enhancements

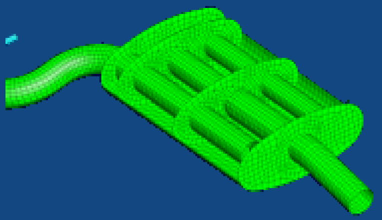
- Ability to Store Element Quality Check Settings in Files
  - New options on the *Element Quality Checks* form allow you to store both element quality values and settings to both your user param file or an external file.
  - *Write Settings To File* lets you store your current settings in an external ASCII file.
  - *Read Settings From File* lets you read in settings that you've saved in an external ASCII file.
  - *Save to Param File* lets you save your current settings in your I-DEAS param file. The software automatically loads these saved settings during your next I-DEAS session.
  - *Load from Param File* lets you override any current settings on the *Element Quality Checks* form using values you've saved in your param file at any time during your session.
- Ability to Check Height of Linear Triangular and Quadrilateral Elements
  - You can now use *Element Quality Checks* to evaluate the size of linear triangular
  - You can now use *Element Quality Checks* to evaluate the size of linear triangular and quadrilateral elements based on their height. When you select *Element Size* on the *Element Quality Checks* form, you can now select the new *Use height for linear tri and quad* option. With that option turned on, the software evaluates the size of linear triangular and quadrilateral elements based on their height, rather than on their edge length. The ability to do check elements based on their height is important, for example, in evaluating the quality of meshes for safety analyses.

Model Check Displays

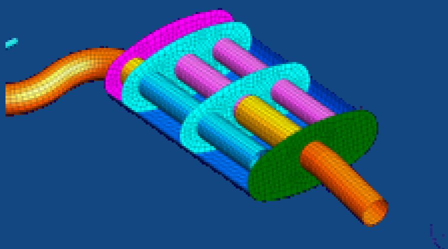
Persistent display

Physical Property Tables

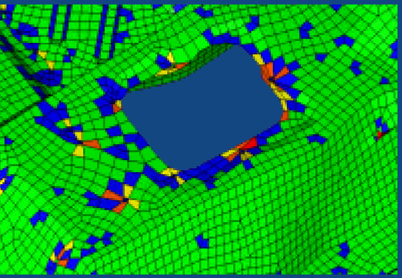
Model Check OFF



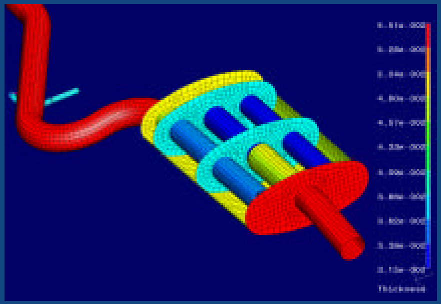
Model Check ON



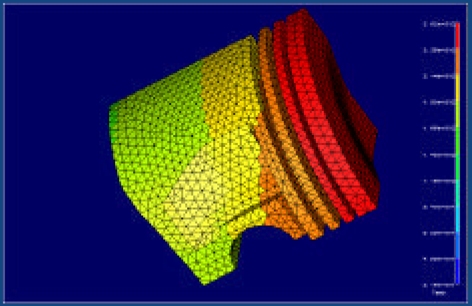
Element Quality Check



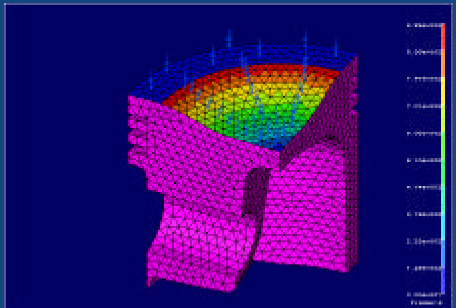
Element Thickness Check



Nodal Temperature Check

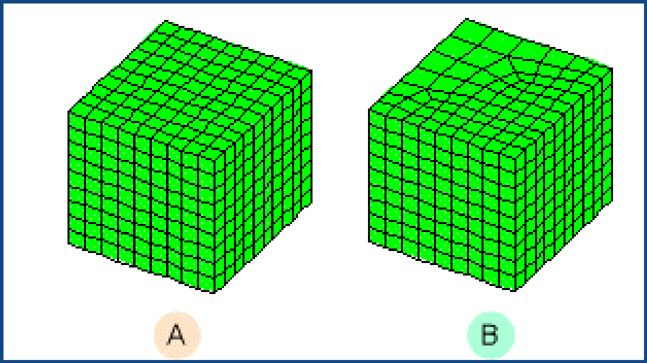


Pressure Check



Ability to Use the Modify Mesh Preview Form to Modify Existing Meshes

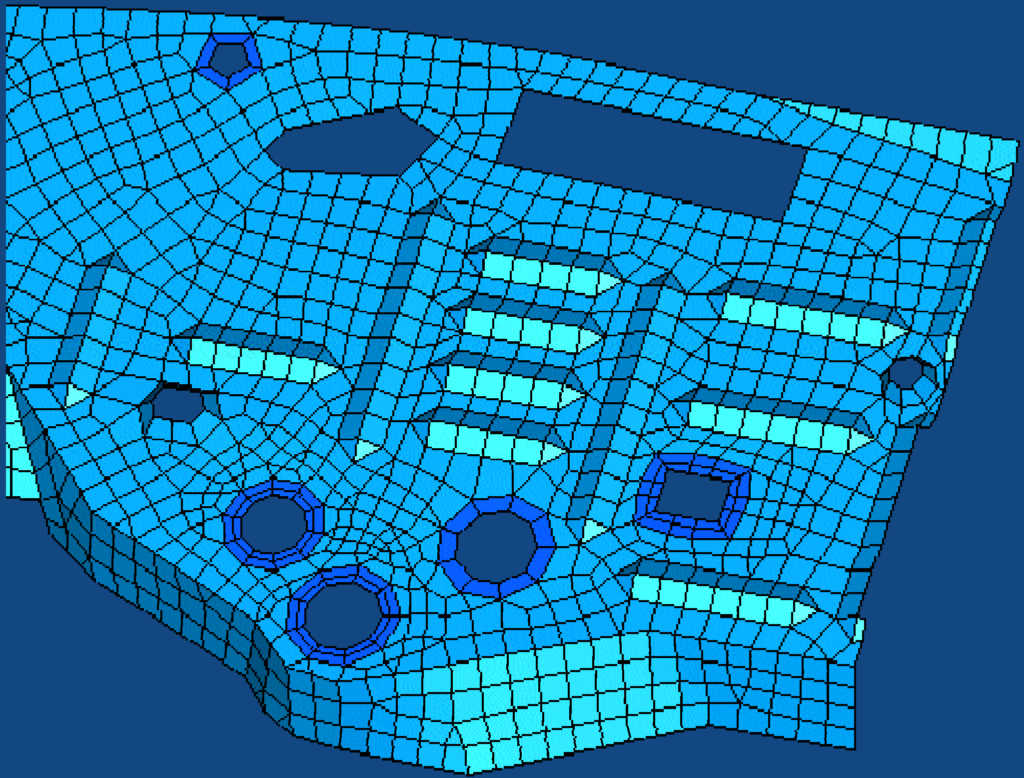
- You can now use the commands on the *Modify Mesh Preview* form to modify surfaces or sections that you've already meshed. In previous releases, you had to first delete the mesh and then remesh the selected surfaces or sections. In many cases, this meant that you then had to carefully select, delete, and modify the meshes on the adjacent entities as well.
- This new capability allows you to incrementally improve existing, frozen meshes without losing your previous work. Importantly, when you modify a mesh on a selected entity, the software automatically ensures continuity between the modified mesh and the meshes on any neighboring entities. For example, if you change the element length on a selected surface, the software creates elements to transition between the modified length and the elements on the surface's boundaries that are shared with neighboring, unmodified surfaces.
- In (A), all surfaces have an element length of 2mm. In (B), we used the *Modify Mesh Preview* form to modify the element length on the top surface to 4mm. Notice how the software transitions the mesh between the new element size we defined and the existing meshes on the neighboring surfaces.



Mapped Meshes Around Holes

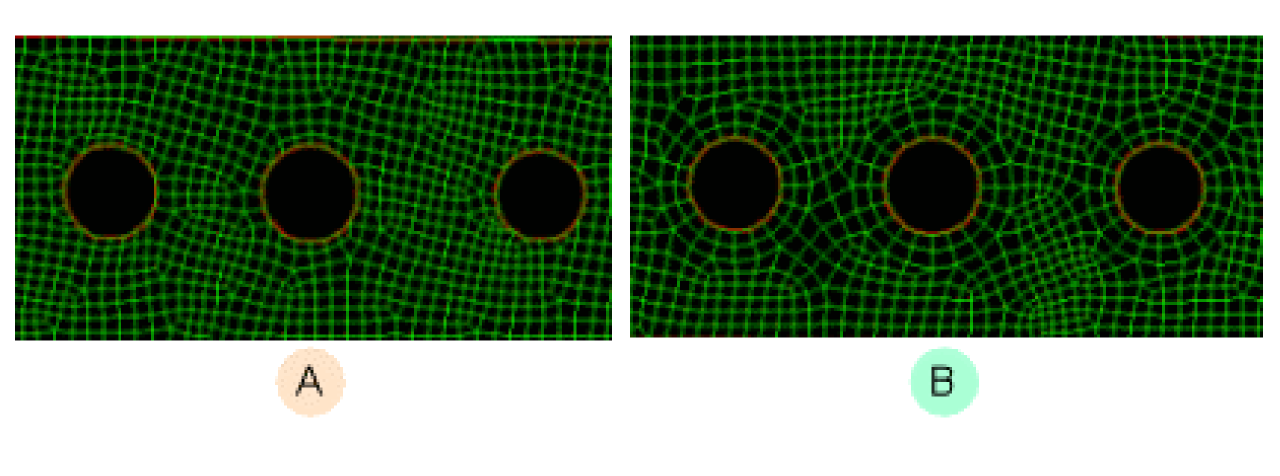
Provide extra control in the regions of highest interest

Combine free meshes with structured (mapped) meshes



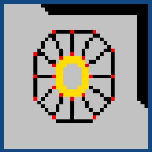
Ability to Define Mapped Meshes Around Inner Loops

- In this release, you can now create locally mapped (structured) meshes around inner loops and interior boundaries. This allows you to define mapped meshes around holes, which could improve the accuracy of local stress calculations. This also allows you to create a locally structured mesh around key interior boundaries of interest within the context of an overall free mesh.
- Notice the difference between the meshes around the holes in the following figure. In (A), the meshes around the interior holes are free. In (B), mapped meshes have been defined locally around the interior holes.

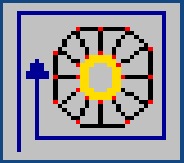


Define Mapped Meshes Around Inner Loops

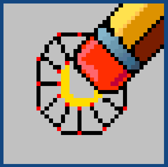
- *Create Mapped Loop* Creates a mapped mesh around a selected inner loop.



- *Modify Mapped Loop* Modifies an existing mapped mesh around a selected inner loop.

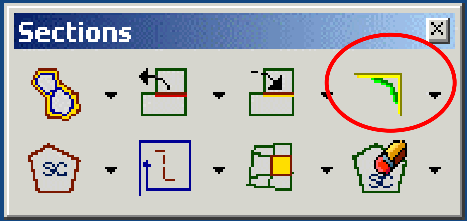


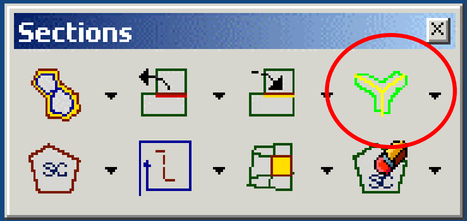
- *Delete Mapped Loop* Deletes an existing mapped mesh around a selected inner loop.



CAD Fillet/Blend Suppression

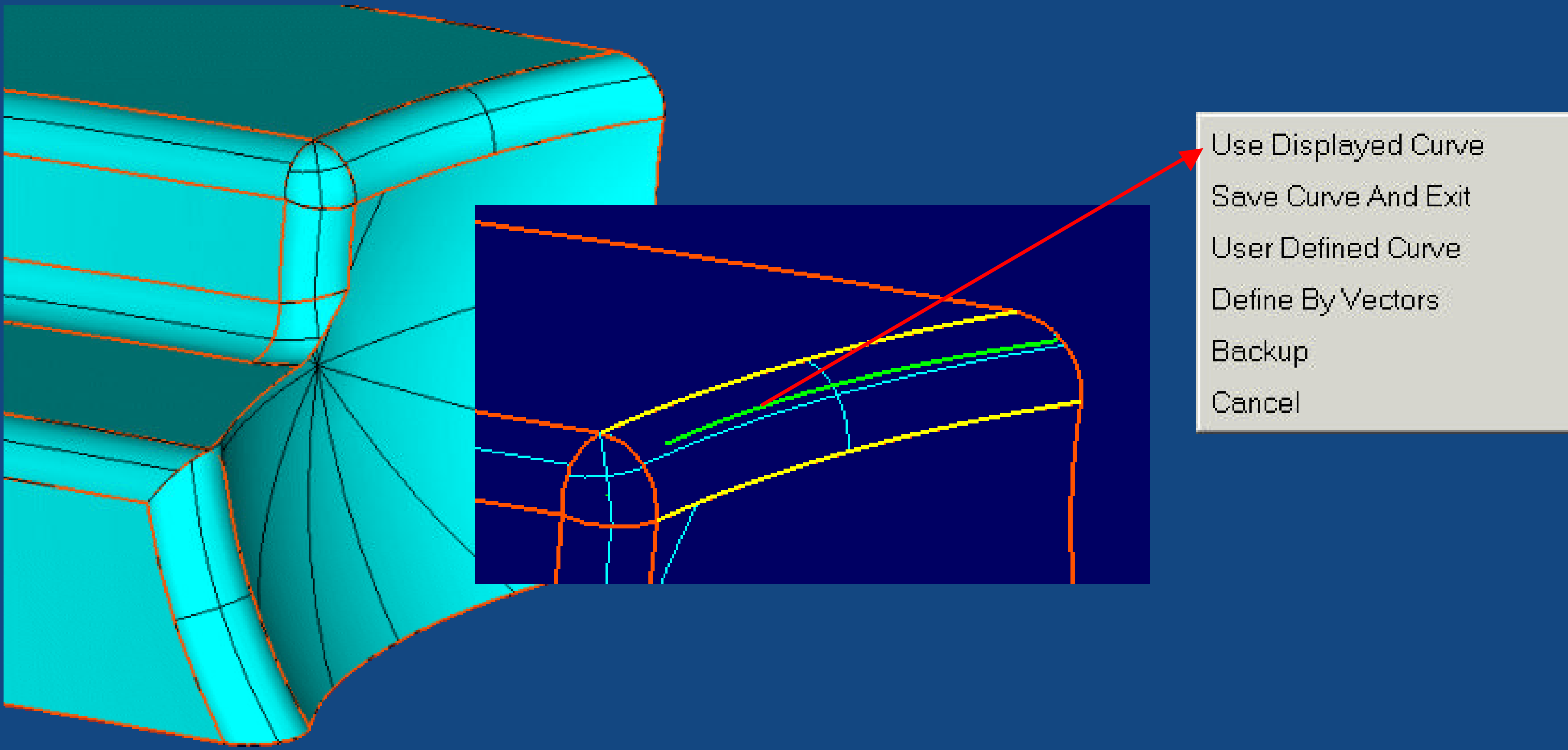
- Remove fillet/blend detail in CAD via sectioning tools
  - Independent of part history tree
  - Abstract blend
    - From rails, create corner
  - Make corner
    - From intersection of corner curves, create corner junction
      - Where multiple blends merge





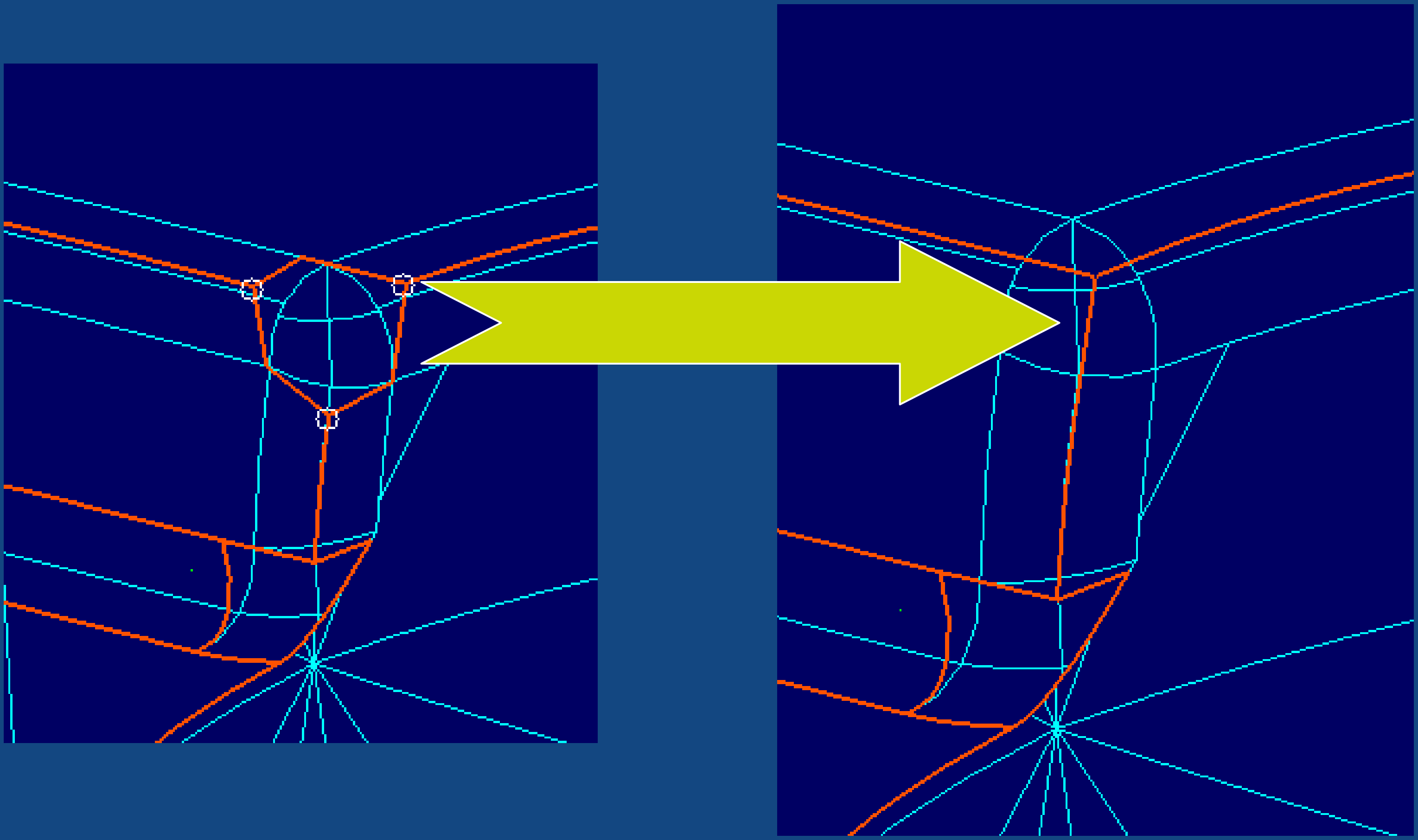
Abstract Blend

- Select first rail, second rail
  - Options to accept result, save curve

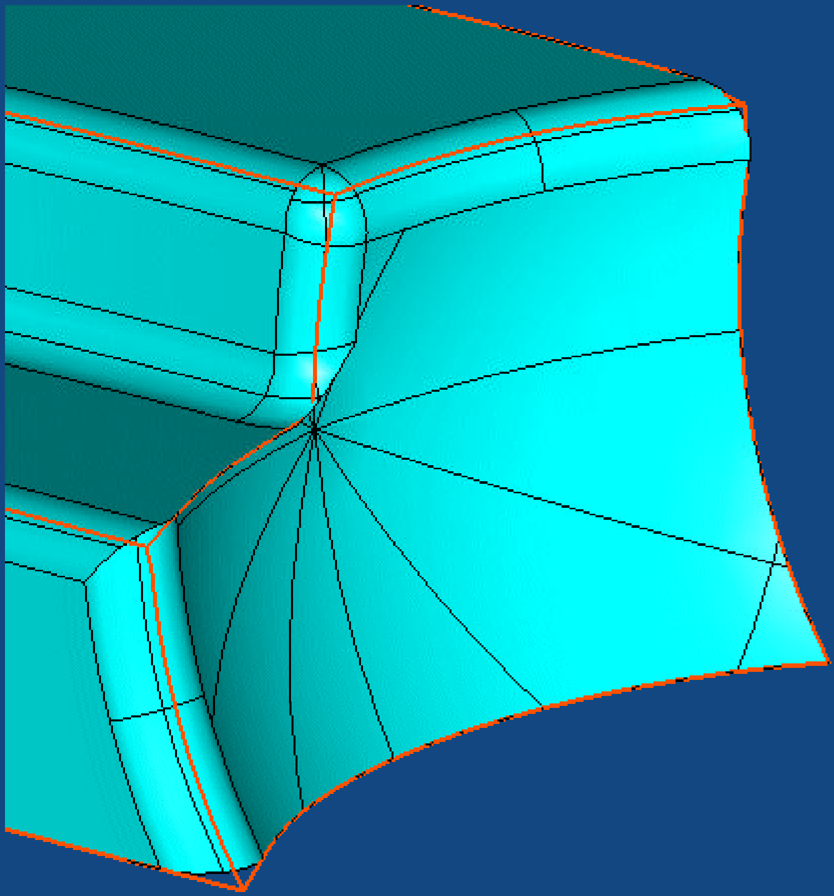


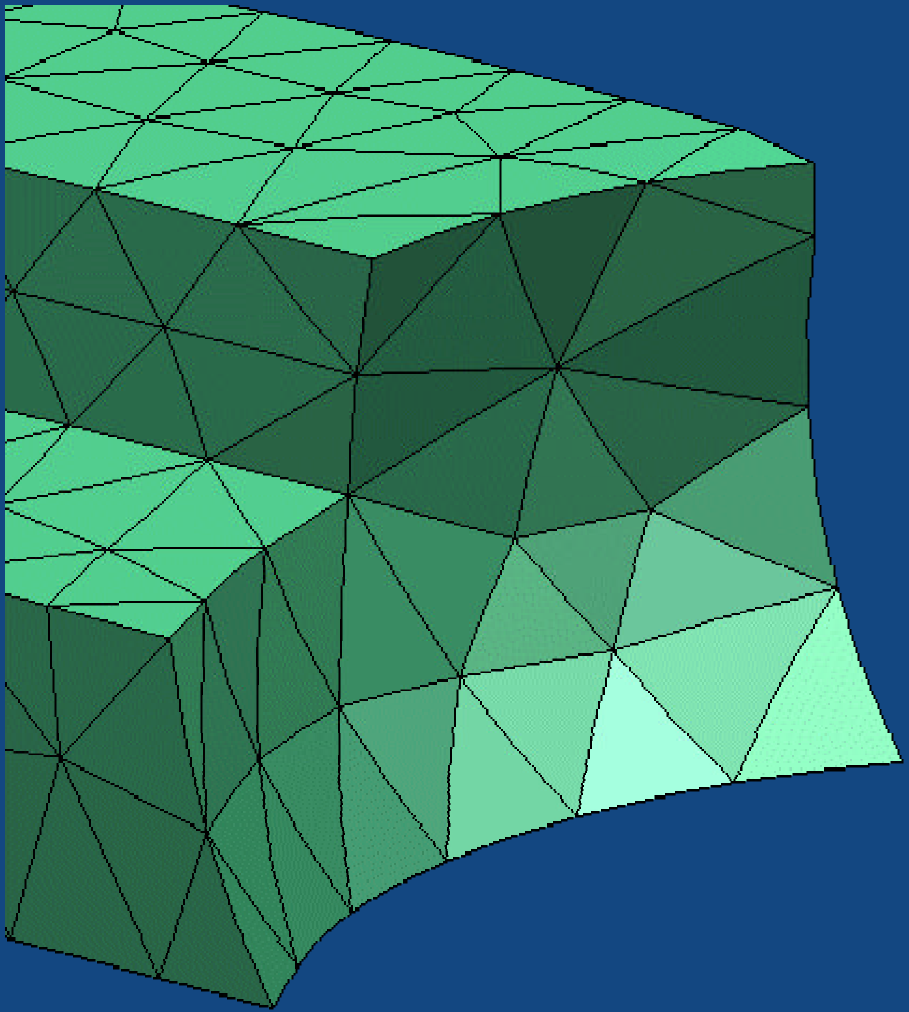
Make Corner

- Secondary operation to abstract blend
- Select corner connectors; software extends curves



Abstraction Results

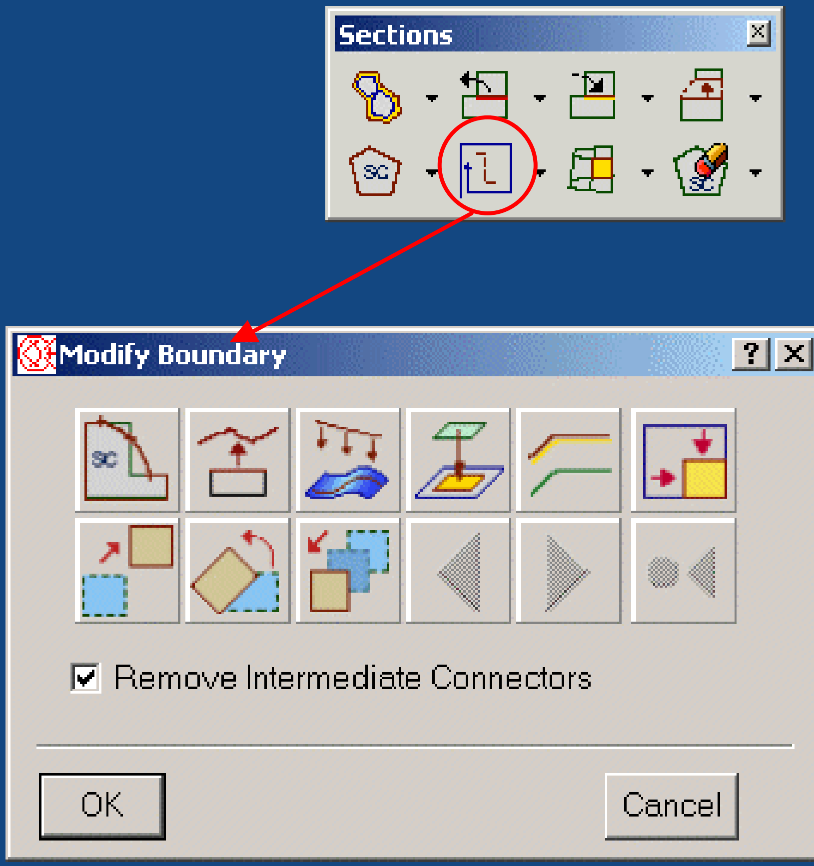




Fillet radius = 4 mm
Element length = 12.5 mm

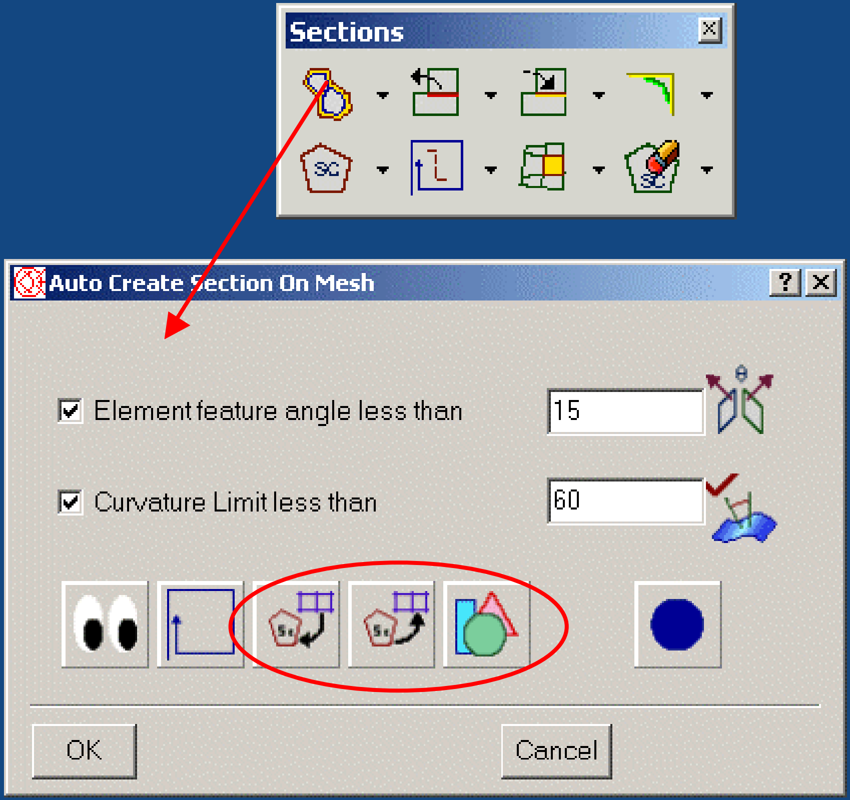
Modify Boundary Advanced Options

- Advanced tools for morphing section boundaries – concept modeling prior to detailed CAD
- Applies to SOS and SOM
  - Smooth boundary, replace curve, closest point project, vector project, offset, scale
  - Translate, rotate, dynamic orient
  - Undo, redo, return to first



Sections on Mesh Process Enhancements

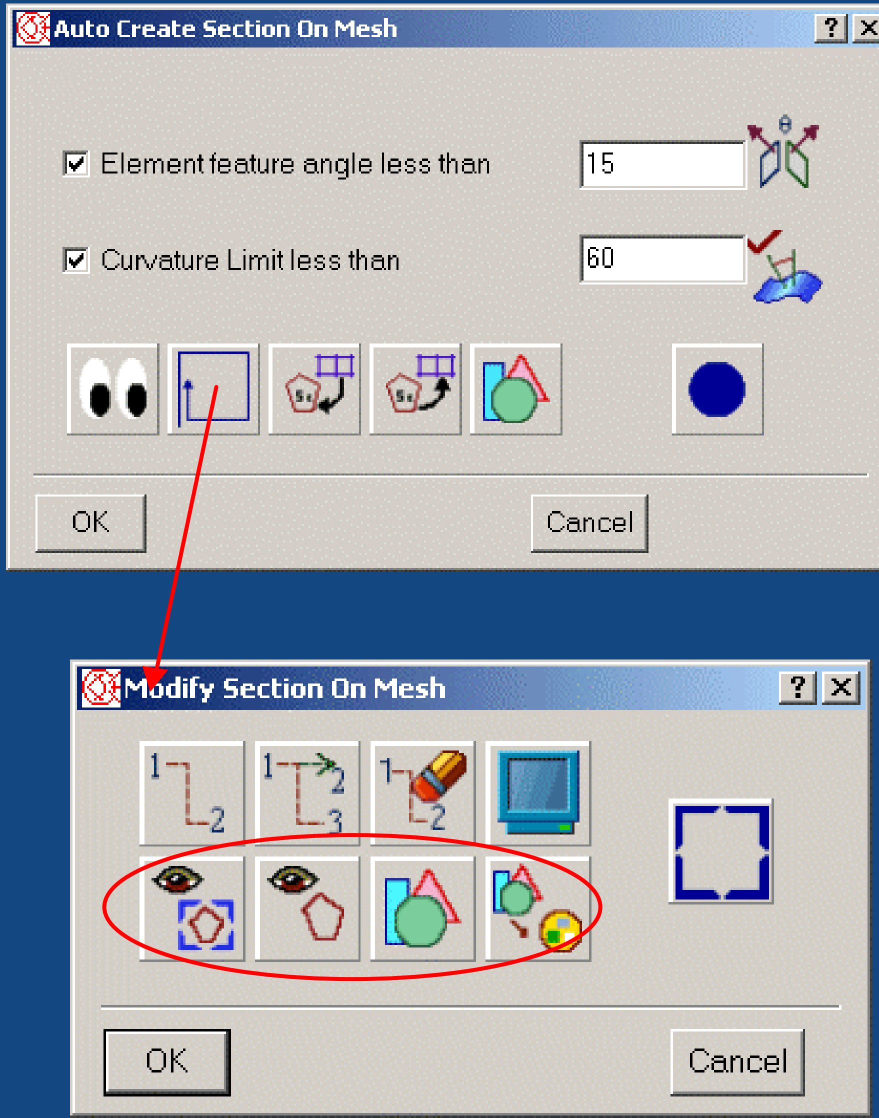
- Ability to add/remove elements to/from selection
- Ability to display all model
  - When in display selected mode and need more of model for completeness



Sections on Mesh Process Enhancements

Modify Section on Mesh

- Further control of display
- Display selected candidate sections and related mesh
- Display all candidate sections and related mesh
- Display all
- Create group from display



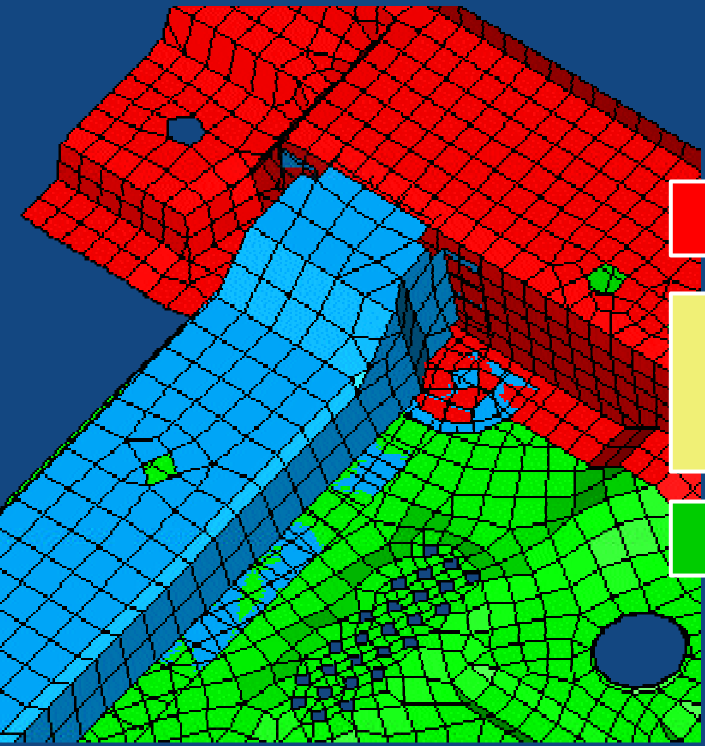
Interference Check Enhancements

- The *Interference Check* command has been significantly enhanced to allow you to repair, as well as identify, regions of element interference and penetration within your model. In I-DEAS 9, if *Interference Check* found any areas of interference within your model, you had to use a variety of manual meshing techniques to move the elements to resolve the interference. Now, when *Interference Check* detects interference within the elements you've selected, you can pick the *Result/Fix Form* option and use the new *Element Interference Result/Fix* form to view, manipulate, and try to repair the areas where interference occurs.
- The *Element Interference Result/Fix* form divides your model into separate "problem areas" that represent the regions where interference occurs. The software determines these regions by finding each interfering element and then adding additional elements in a radius around that element. Because resolving interference on a large model can be a difficult and iterative process, working with one problem area at a time can help you manage the scope of the problem. Each problem area is comprised of multiple components (physical property table IDs).
- To actually repair the interference between components, the software moves the nodes on the interfering elements as well as on elements surrounding them based on a percentage of the elements' thickness. Different options on the *Element Interference Result/Fix* form allow you to constrain either specific nodes or entire components to control this movement.
- The *Element Interference Result/Fix* form contains a number of different options that help you repair interference problems. The following sections describe the different options on that form

Interference Check/Fix

- Component color

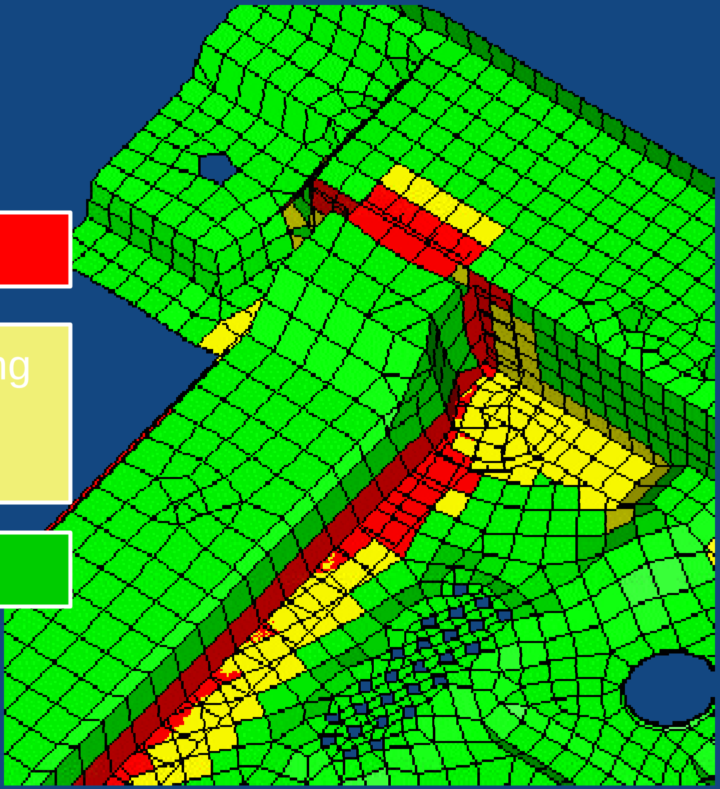
Interference color



RED : Intersecting

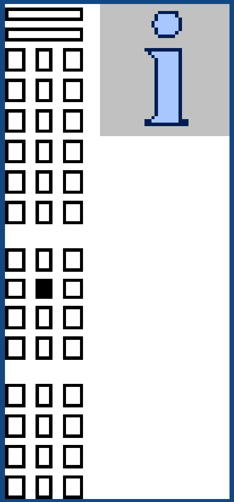
YELLOW : Interfering (Only in Element thickness)

GREEN : OK

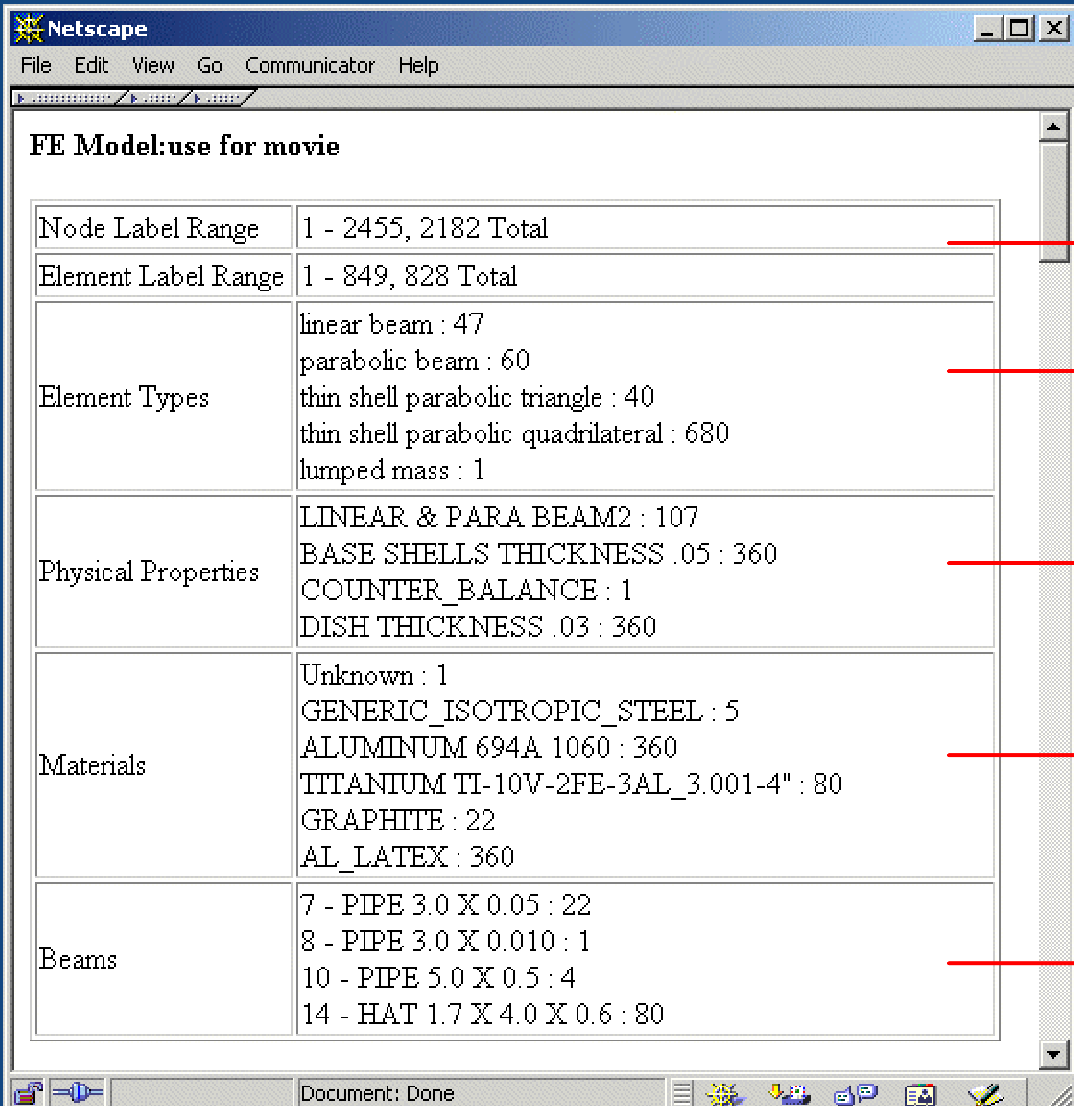


FE Model Documentation

- The *Model Documentation* icon is available throughout the Simulation application.
  - This capability allows you to capture FE data for the current workbench model and publish it as an HTML file.
  - Using *Model Documentation*, you can view information about the model's materials, physical properties, beams, boundary conditions, and results



FE Model Documentation



FE Model:use for movie

Node Label Range	1 - 2455, 2182 Total
Element Label Range	1 - 849, 828 Total
Element Types	linear beam : 47 parabolic beam : 60 thin shell parabolic triangle : 40 thin shell parabolic quadrilateral : 680 lumped mass : 1
Physical Properties	LINEAR & PARA BEAM2 : 107 BASE SHELLS THICKNESS .05 : 360 COUNTER_BALANCE : 1 DISH THICKNESS .03 : 360
Materials	Unknown : 1 GENERIC_ISOTROPIC_STEEL : 5 ALUMINUM 694A 1060 : 360 TITANIUM TI-10V-2FE-3AL_3.001-4" : 80 GRAPHITE : 22 AL_LATEX : 360
Beams	7 - PIPE 3.0 X 0.05 : 22 8 - PIPE 3.0 X 0.010 : 1 10 - PIPE 5.0 X 0.5 : 4 14 - HAT 1.7 X 4.0 X 0.6 : 80

Label ranges

Element types

Physical properties

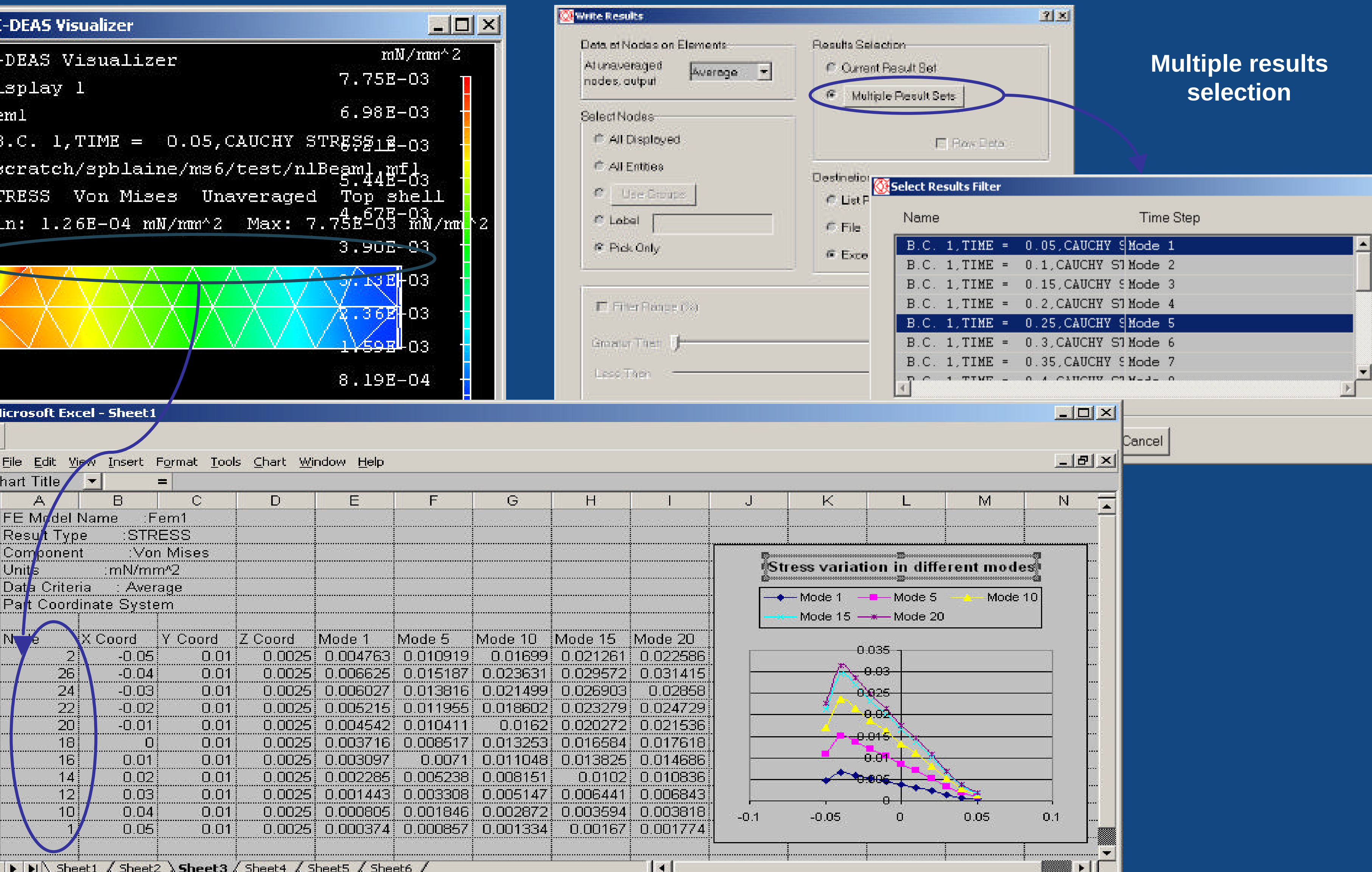
Materials

Beam cross sections

Visualizer

- Beginning with this release, you can write result sets into a spreadsheet data file or read result sets from a data file. These enhancements allow detailed results inspections and results manipulation.
- Writing Results
-
- The *Write Results* icon, available from the Visualizer subpanel, allows you to output results data directly to a spreadsheet. The Microsoft Windows version of I-DEAS writes directly to Microsoft Excel. The UNIX version of I-DEAS writes to a .dat file. This feature allows you to:
  - **further analyze results data**
  - **display the data graphically**
  - **manipulate the data to create new results**
- Reading Results
-
- You can use the new *Read Results* icon to specify parameters for reading results back into the software. This lets you manipulate tabular results and read them in as new results. All data that is brought in with this feature comes in as scalar data.

Visualizer Spreadsheet Interface

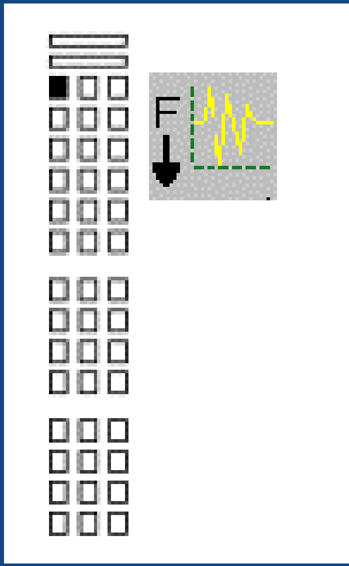


What's New Overview – BC and Solvers

- Direct Frequency Response
- Model Solution Memory Management
- 64 Bit Solver
- Binary Results File
- Auto Select sparse/iterative enhancements
- Contact enhancement
- Assembly Solver
- External Solver Translators

Ability to Define Frequency-dependent Loading

- Beginning with this release, Model Solution now supports a new direct frequency response analysis type. A direct frequency response analysis allows you to calculate the frequency response of a structure subjected to harmonic forces over a range of excitation frequencies. To support this new analysis type, the existing *Forced Response* analysis type icon has been renamed to *Frequency Response* to clarify its intended use with direct frequency response.
- When you select the *Frequency Response* analysis type icon, you can now define frequency-dependent loading for your model. These new capabilities allow you to:
  - define nodal forces and elemental pressure loads with phase angles and frequency histories
  - define prescribed nodal displacements, velocities, and accelerations with phase angles and frequency histories
- Frequency histories are similar to time histories, except that frequency histories have a real and an imaginary component. You can define them as a function using f as the frequency in hertz or w as the frequency in radians/second. You can enter these functions manually on the new *Frequency Variation* form, or you can import them from either ADF or spreadsheet files.



New Frequency Response Analysis Type

- Model Solution now includes a new frequency response analysis type. This frequency response analysis allows you to directly calculate the frequency response of a structure subjected to harmonic forces over a range of excitation frequencies. For example, this new analysis type allows you to perform frequency response analyses that include:
  - frequency-dependent dampers and springs
  - discrete damping characteristics
- This direct frequency response capability complements the frequency response capability available in the Response Analysis task. Whereas the frequency response capability in Response Analysis performs all calculations in the modal degrees of freedom and then transforms the results back to physical space, the new direct frequency response capability performs all calculations in the full physical space. Although this method is more computationally intensive, it offers several advantages. For example:
- you don't need to worry about whether you've included a sufficient number of modes in your analysis
- damping is handled accurately, rather than through a modal damping factor, as in Response Analysis.

Solver Memory Management Enhancements

- In this release, the *Memory Settings for Solve* form has been streamlined to greatly simplify the process of configuring the sparse matrix solver's memory management capabilities. The form now contains only two options: *Main Memory* and *Workspace*.
- *Main Memory* allows you to specify the amount of memory to be used for the sparse matrix solver's buffer and other areas that need large amounts of memory. If you select *Auto* for this option, the software initially uses 70% of the RAM on the machine that performs the solve as the *Main Memory* setting. If this amount of memory is insufficient for the sparse matrix solver, the software automatically reallocates additional memory and restarts the sparse matrix solver setup process. If this additional amount of memory is still insufficient, the software prints a message to the List region and terminates the solve.
- Note: The software allocates a maximum of 1500MB for 32 bit executables.
- *Workspace* allows you to specify the amount of memory to be used for smaller memory allocations. This option is identical to the *Fortran Workspace* option from previous releases. If you select *Auto* for this option, the software automatically calculates and allocates memory at the beginning of the solve. If this automatic memory allocation is not sufficient for your needs, you can manually specify a *Workspace* value.
- In addition to these user interface improvements, the memory settings used by the solver are now reported at the beginning of the list (.lis) file for each analysis.

64-bit Memory Address Space Support for UNIX Platforms

- Both the *Local Batch* and *Remote Batch* solve options in Model Solution now support a 64-bit memory address space on all supported UNIX platforms except for IBM and Sun. In previous releases, Model Solution only supported a 32-bit address space, which restricted it to using only 2Gb of real memory. Support for 64-bit address space allows you to run Model Solution on machines that are configured with more than 2Gb of memory.

Binary Results File

- When you use Model Solution's batch solve capabilities, the software now generates a binary results file (.bun). In I-DEAS 9, the solver generated ASCII universal (.out) files of the results, which tended to be large, and which took large amounts of time for writing and reading. To import a .bun file back into I-DEAS once you've solved your model, pick *File*, *Import* and select the new *I-DEAS Solver Results File* option.

Sparse/Interactive Solver

- New Ability to Automatically Select the Appropriate Solution Algorithm
  - A new *Auto Select* solution algorithm option has been added to the *Solution Options* form. Use this option to have the software automatically determine whether the sparse matrix or iterative solver is more appropriate for your model. When you pick *Auto Select*, if your model contains no shell elements and 90% or more of the elements are solids, the software uses the iterative solver.
- Guyan Reduction Enhancements
  - On the *Solution Options* form, several changes have been made for saving the reduced matrices calculated by the Guyan reduction method.
  - In previous releases, you could save those matrices to your model file, an external binary file, or a universal file. In this release, you can now choose to save the results to either a hypermatrix file or a universal file.
  - The *Save Recovery Matrix* option available in previous releases has been renamed to *Save Back-Expansion Matrix* to clarify its use. Additionally, if your model contains a group named NODE_OUTPUT, the software only writes data for the nodes in this group to the back-expansion matrix. However, the software will always write data for all master degrees of freedom to the back-expansion matrix.

Contact Control Enhancements

- In this release, a new *Allowed number of element status changes* option has been added to the *Contact Control* form that allows you to set less stringent contact convergence criteria for selected cases. In previous releases, when you performed a contact analysis, the software considered a solution to be converged only when the status of all contact elements had stopped changing. In this release, *Allowed number of element status changes* lets you specify how many contact elements can be changing and still have the software consider the solution converged. For example, if you set this option to 2, the software considers the solution converged as soon as two or fewer contact elements are changing status.

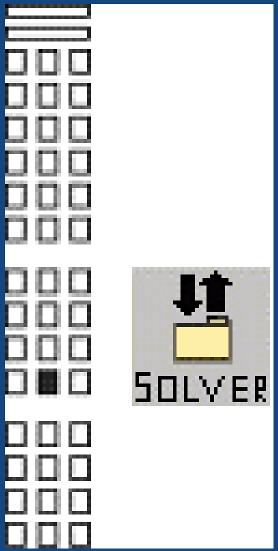
- If you select *Save Back-Expansion Matrix*, you can use the new *Translational Dof Only* option to save only the translational degrees of freedom to the back-expansion matrix. See Also

Assembly Solution

- This release includes several enhancements to the Assembly Solution capabilities.
- To use Assembly Solution, enter "/xx as" in the Prompt region of Model Solution. This activates the *Assembly Solution* toggle on the *Create FEM from Assembly* form.
- Support Added for Frequency Response Analysis
  - You can use the new Frequency Response analysis type to analyze Assembly Solution models. For more information on Frequency Response analyses, see the Model Solution topic.
- Craig-Bampton Reduction Override Enhancements
  - The *Craig-Bampton Reduction Override* form has been enhanced to allow you to deactivate selected modes and control damping. In previous releases, these options were only available with modal reduction overrides.

New Solver Operators Subpanel

- The new *External Solvers* icon available in all tasks within the Simulation application launches the new *Solver Operators* subpanel.
  - Icons on this subpanel allow you to perform various solver import and export functions for both Model Solution and external solvers, such as NASTRAN and ABAQUS. Additionally, icons on the *Solver Operators* subpanel also allow you to launch the new LS-DYNA, RADIOSS, and PAM-CRASH Toolkits.
  - In previous releases, these functions were available only through the *File*, *Export* or *File*, *Import* menus.



ABAQUS Solver Interface

- Abaqus Standard type versus Abaqus Explicit type
  - The exporter in I10 makes a real distinction between Abaqus Standard type and Abaqus Explicit type.including defining contact pairs in modal data for standard type and in step data for explicit type.
- Multiple Contact Pairs
  - The interface to define contact pairs has been enhanced so the users could create multiple contact pairs when they create contact pairs from a I-DEAS contact set. And they also can modify and delete multiple contact pairs. The preview feature allows the user to check the master and slave groups visually if this pair is created as Abaqus Exporter contact pair.
- User Specified Text
  - *ABAQUS user-specified-text* now enables you to build your ABAQUS input file in an easy and flexible way. You have 3 options to set it up: Text, Link-File, Merge-File that will allow you to save text messages with your solutions.
- Support gasket element
  - If the abaqus suptype is set to gasket on the PPT form, a *GASKET SECTION card will be output to the deck file with the MATERIAL parameter. The setting on the PPT subform for gasket subtype will be output into a dataline in input file. Note the user should use the user-specified text form to complete material data in input file

ABAQUS Solver Interface

- ABAQUS Amplitude
  - Using *ABAQUS Amplitude Curve Definition* enables you to define amplitude curves. You can create/export amplitude curves by assigning name, attributes and curve points. The new *ABAQUS Amplitude Curve Sets Manager* contains a list window for defined amplitude curve sets, showing the set name, and summarized relevant attributes. These amplitude curve sets can be modified, copied and deleted. They can also be referenced in defining step data.
- Data file control
  - Use this feature to control the output of whether you want element or nodal output and what types of output for elements or nodes in the **.dat* file.
- Contact Interference
  - If allowable interference is set to 0, there is no CONTACT INTERFERENCE card exported. Otherwise, you can enter your own setting on the form to decide which parameters you want to export.

ABAQUS Solver Interface

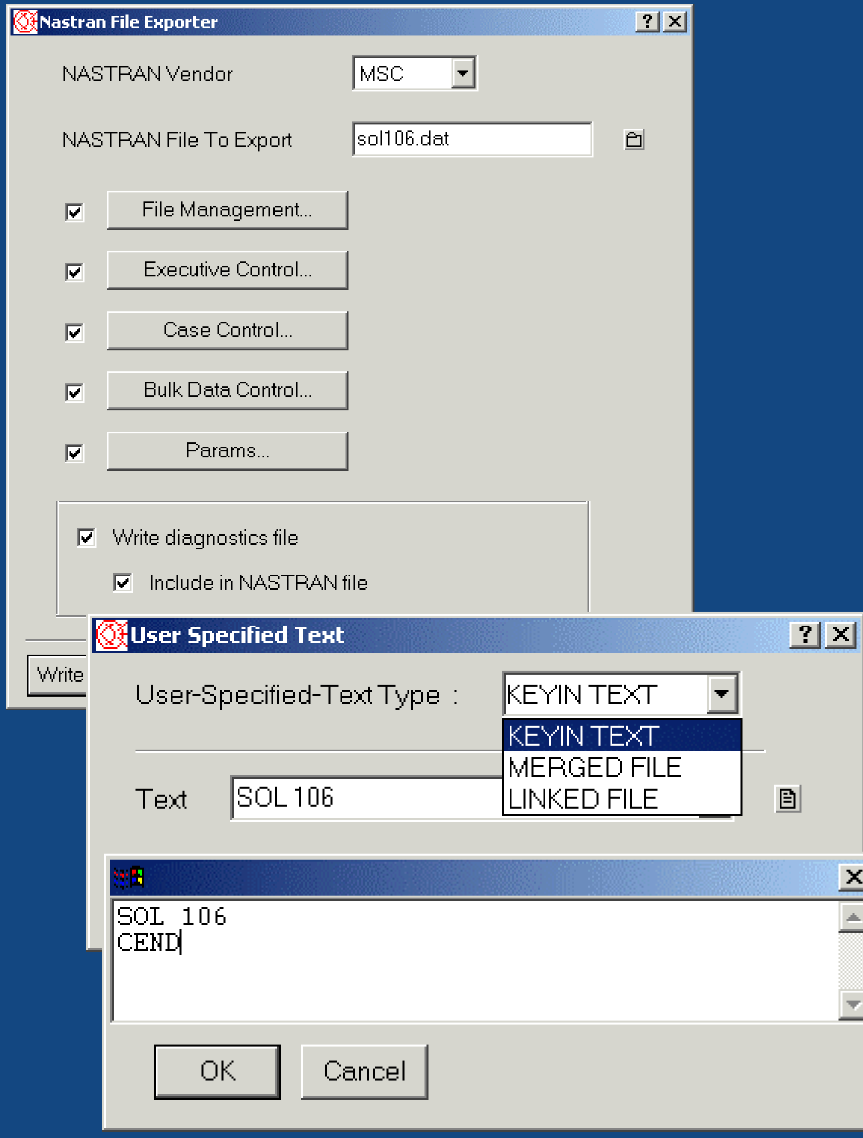
- Contact control
  - *Contact Control* provides you with a list of attributes that you can set by using the ON/OFF toggles, menu selection or inputting data.
- Contact Surface Interaction Parameter
  - This form is totally re-done so that there is a bigger capacity for information and it is more flexible to suite the user's needs.
- Contact pair parameter
  - The form has a very different look for Abaqus Standard type and Explicit type. Actually the form is new for ABAQUS Explicit.
- Abaqus 5.8 format input file
  - The exporter keeps supporting the Abaqus 5.8 format input file although its GUI has been re-designed based on analysis of Abaqus 6.2 format. The complex and complete migration work is done so the user will still be able to work on pre-I10 models.
- Boundary Conditions property assignment
  - Use the new property assignment to create the restraint set, constraint set and load sets within selected boundary condition sets.

ABAQUS Solver Interface

- New Forms in the user interface to take advantage of these enhancements
  - User Specified Text
  - ABAQUS Amplitude Curve Set Manager
  - ABAQUS Amplitude Definition
  - Data file control form
  - Contact Interference form
  - Contact control
  - BC property assignment form

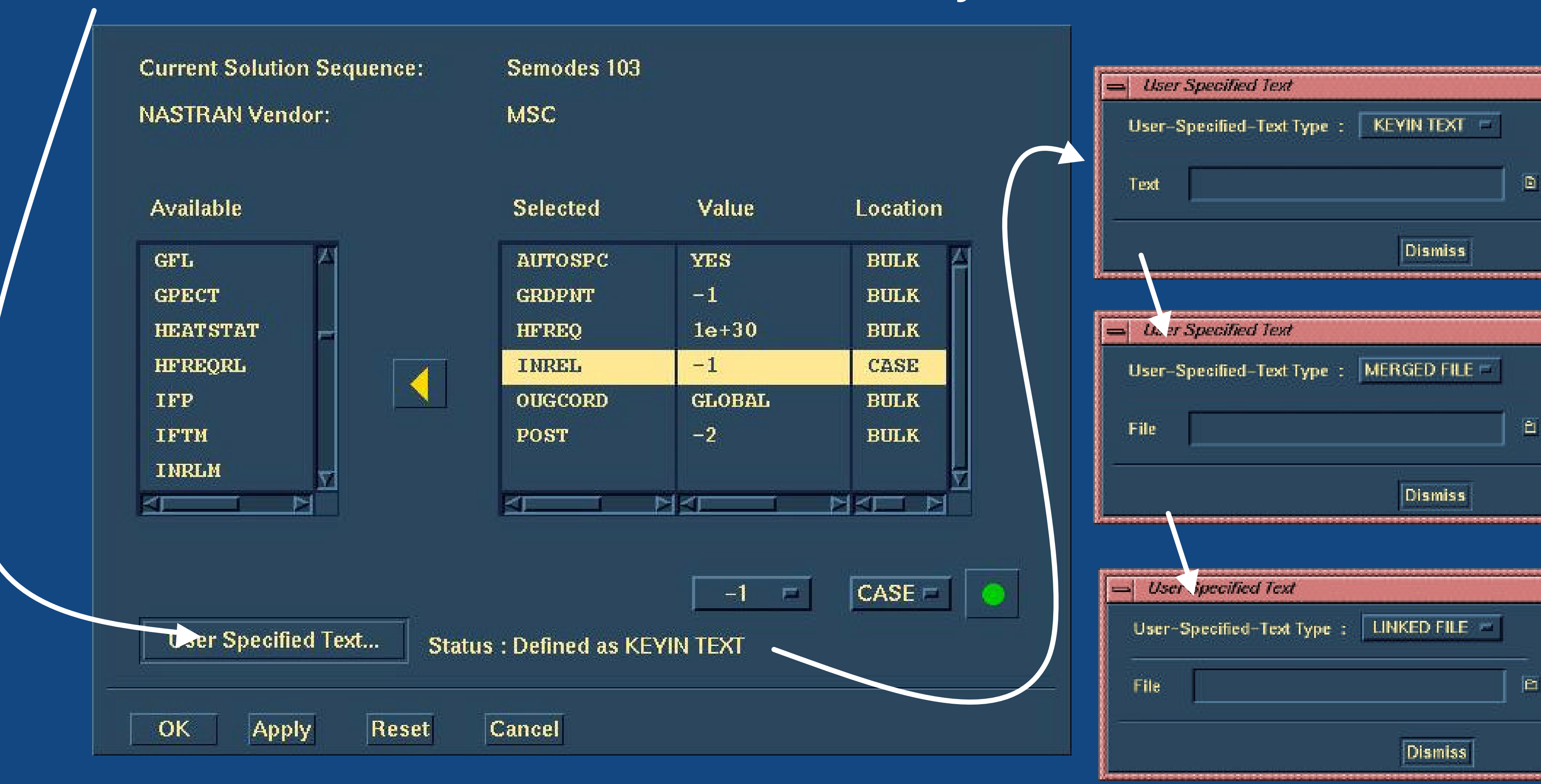
NASTRAN Exporter

- Export I-DEAS traction loads
- Support linear buckling solution 105
- Export user defined text
  - Keyin text, merged or linked files
- Write out I-DEAS groups as Case Control Sets
- Maximize small-field format significant digits
- Enhanced round-tripping
  - Permanent SPCs on GRID cards
  - CORD1/CORD2 cards
  - PLOTEL card labeling



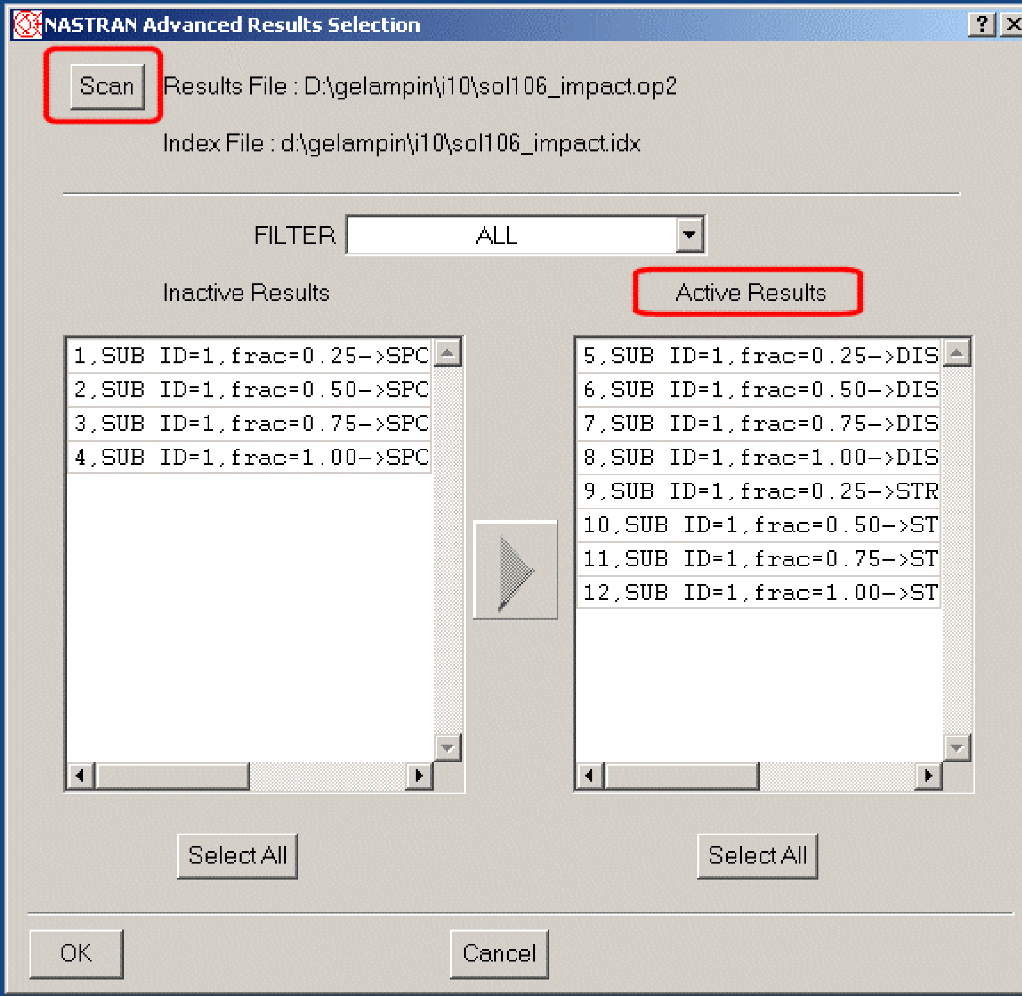
NASTRAN Exporter

- New PARAMs UI
- Enhanced 'Include File' functionality



NASTRAN Importer

- Selective import of results
- Non-linear stresses and strains
- Case Control import of:
  - Boundary Condition Sets
  - Sets
- SORT2 data imported as functions



New Solver Interface Products

- The simulation of complex structure model using explicit solvers such as LS-DYNA, RADIOSS and PAMCRASH generally requires an FE model with very large number of elements, complex contact definition and boundary conditions, as well as special modeling entities such as crash material models, airbags, seatbelts, sensors.
- The documentation for these products is not part of the I-DEAS online help system, and it cannot be accessed from the Help button in Ideas. Instead, access the online help for these products by first starting the application, then clicking on the Help button at the top of the main form, or at the bottom of any secondary form.

Crash Explicit Code Translators

- Initial release of crash translators
- LS-DYNA and RADIOSS
  - Operates via toolkit
  - File from I-DEAS
  - File to I-DEAS
  - Keyword tree
- PamCRASH in 10m1
- Continuing development with downloadable updated versions