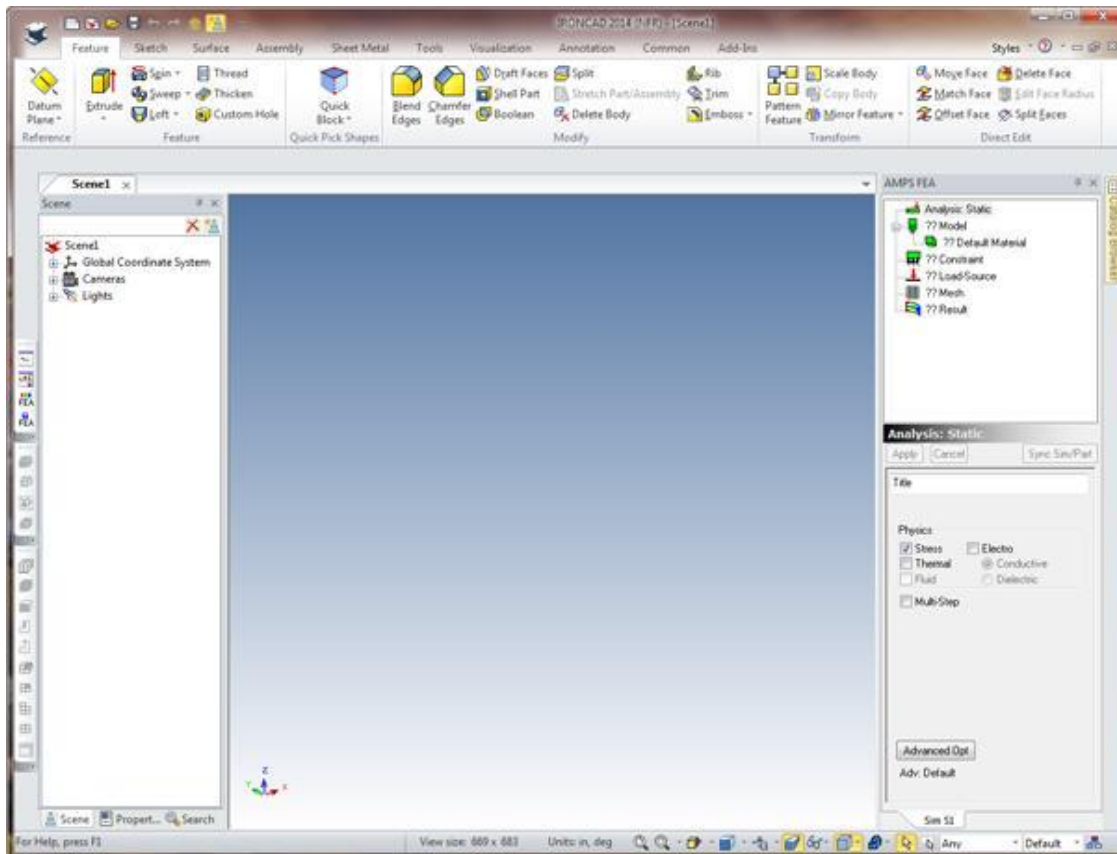


IronCAD MultiPhysics (Alpha Documentation)

6/12/2014

First time Installation for Alpha Release

- 1) You must have IronCAD 2014 SP2 installed before you can Install Multiphysics for IronCAD (MPIC).
- 2) Unpack the installation file into a temporary directory, and execute setup.exe. The installation will automatically identify the existing IronCad 2014 SP2 directory and installed the needed MPIC components into it with needed SP2 patches.
- 3) Start IronCAD, open a blank file and select the Add-In ribbon tab. Select 'Add-In Applications' on the Add-In Ribbon Bar and "IronCAD Multiphysics" should be available. Add a check to the box and close.
- 4) In "Add-on Tools" Toolbars section, there are three toolbars associated with MPIC (ICamps Main, ICamps FEA Display, ICamps FEA Selection). Select them, and position them so they are vertical and take up less space on the left. Then CLOSE IronCAD.
- 5) Re-open IronCAD with a blank file and click on 'Add FEA' (third icon down on the main toolbar – the only AMPS toolbar that is active). Then click OK in the Select Analysis Type dialog to start a static analysis. The FEA Window will open, but will be unattached, small and somewhere on your screen. Move this window to IronCAD and dock it on the right. Drag the edge of the window in so you can the full dialog. The screen should look like this:



To analyze an existing model, close the 'Scene1' model (without saving the model or FEA) and load a model that you would like to use for FEA analysis.

To create a simple block to analyze, click on the 'Hide FEA Window' button (top button on the main FEA toolbar), drag a block into the screen, resize the block and enclose it. Now click on Show FEA (if you had not yet click Add FEA, you would do that here).

Useful HINTS:

- 1) Right clicking on the Analysis leaf of the FEA tree provides a menu to change general preferences, copy a SIM (FEA simulation) or Delete the current SIM (FEA simulation).
- 2) The FEA tree names that start with a question mark, '?', should be examined or items should be added for a valid analysis. The one exception is that sometimes the 'Load-Source' item is not needed for an analysis if the Constraints describe the full condition. For example, setting the temperature at two faces of the model for a thermal analysis does not require a 'Load-Source'.
- 3) Boundary Conditions can be deleted or disabled by right clicking on the tree leaf and using the pop-up menu.

Structure of Documentation

Section A) Overview is an overview of using IronCAD for FEA analysis.

Sections B) through H) describe in details the pages and options associated with the top level leaves of the FEA tree: Model, Constraint, Load-Source, Mesh and Result.

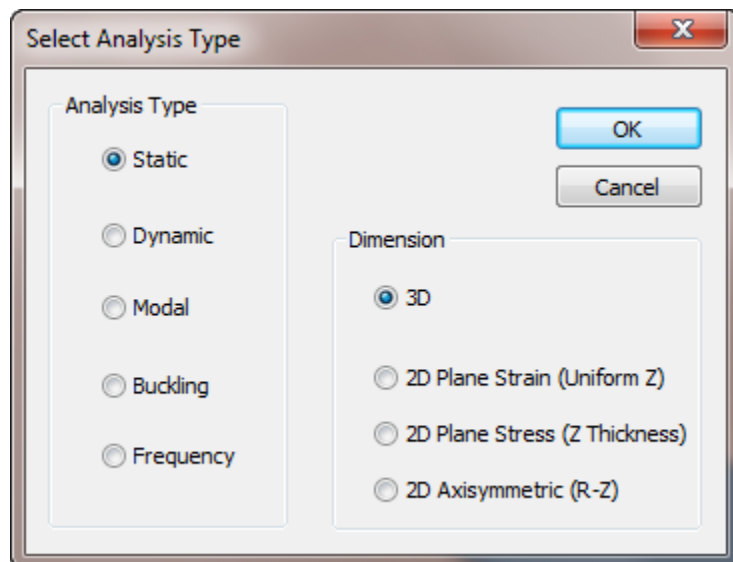
Section I) describes the right-click menus associated with FEA analysis tree leafs. There are right click menus for the Analysis, Material, Body, Constraint sub-leaves, Load-Source sub-leaves and MESH, Mesh>Refine leaves. There is also an FEA right-click menu for the model window that is available when the FEA mesh is displayed. It allows the user to hide/unhide FEA elements and control the display of FEA edges, nodes and inner faces.

A) Overview

The following steps are necessary to perform FEA analysis of a model.

1) Create a new SIM (or copy an existing SIM)

The **Select Analysis Type Dialog** is displayed by selecting Add FEA from the ICamps section of the Add-In ribbon bar or from the main FEA toolbar (3rd icon from the top).



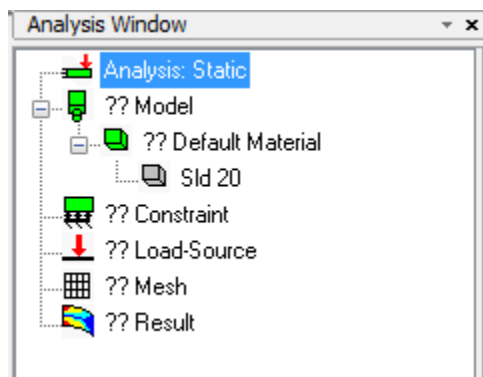
Static analysis is the default and is used for simulations that are not time dependent. Dynamic analysis is used for time dependent studies. Modal is used to display the shape and frequency of the resonances of the model (including BCs). Buckling is used to display the buckling modes (including BCs). Frequency analysis is used to display the response of a model to a sinusoidal load at various frequencies.

If one of the 2D analysis methods is used (and not 3D), then the CAD model must be surfaces in the XY plane (actually, when meshed, all nodes are projected to the Z=0 XY plane). For each of the 2D models, the XY surfaces represent a cross section of the actual 3D model and forces or variations in the 3rd (not

cross section) direction are not allowed. 2D Plane Strain models represent an XY cross section of a model assumed to be infinite in the Z direction. 2D Plane Stress models are flat plates (of a specified thickness) in the XY plane with forces limited to the XY plane. For 2D Axisymmetric models, the physical model corresponds to the XY plane cross section being rotated 360 degrees around the Y axis. For example, a circle of radius 1 at X=2 would correspond to a torus (donut) with a major radius of 2 and a minor radius of 1.

2) The FEA tree display

After the New Analysis type has been selected, the FEA dialog region is displayed. The top part of this region contains the FEA Tree and the bottom part contains the page for the selected FEA leaf.

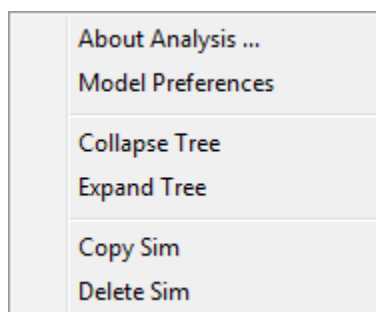


The FEA tree display provides a summary of the current simulation. Double question marks (??) indicate pages that haven't been used or examined. Starting at the top and working down works well in setting up most analyses.

Many tree leaves have a 'right click' menu associated with them. The Analysis leaf (top leaf) has a right click menu that can be used to copy or delete the entire simulation.

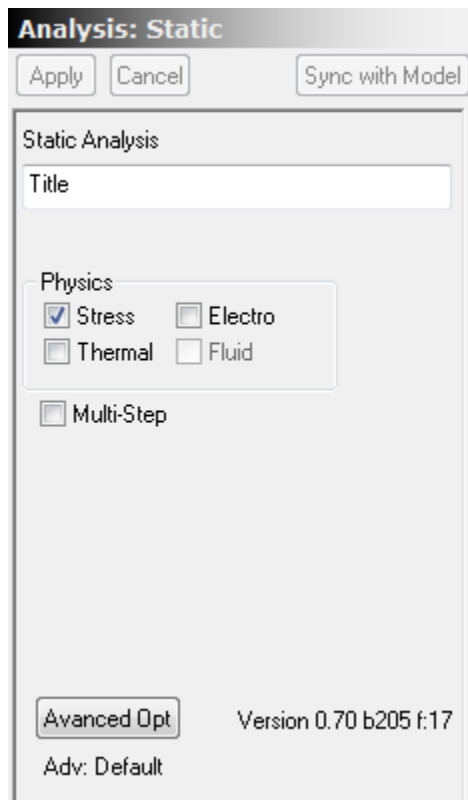
3) The Analysis Page

Right clicking on the Analysis leaf (top leaf) allows operations to be made on the tree and simulation.



The **Copy Sim** option is useful if you want to perform a different type of analysis but preserve the current materials, constraints, boundary conditions and mesh settings.

The Analysis page allows the type of physics used in the analysis to be set. Although usually only done when changing 2D Plane Stress models to 3D Shell models, the 'Dimension' option for the analysis can also be changed.



Also, for Static Analysis, the Multi-Step option can be used. If it is selected then a pseudo time is used so that displacements and loads can be applied slowly (dependent on the pseudo time). For non-linear static problems, Multi-Step should be turned on. This includes Contact analysis, thermal radiation analysis and analysis with non-linear material properties.

4) Model Page

The Model page (?? Model) is used to set the units and add new materials if necessary.

Part Length Unit: inch

Materials

Add New Material

Remove Unused

FEA Units

Length: inch

Unit System: USA

Temperature: F-Fahrenheit

Enable Visible Only

☐ Mesh Solid as Shell

Initially the units are based on the length unit chosen for the IronCAD model and all bodies are classified under one material. The **Add New Material** button is used to add more material pages and then the **Add Bodies** button on the material page is used to assign bodies to that material. The **Remove Unused** button is used to remove material pages that do not have any bodies assigned.

The **Mesh Solid as Shell** checkbox is a quick way of treating the surface of the solid as a shell FEA model. It should only be used if all the shells for each solid are of the same material and the same thickness. If multiple materials or thickness are to be used from one body, the faces of the solid body should be converted into surfaces. In IronCAD, click on one face until the face is selected. Then right click on the face and choose **Create** then **Extract Surface** from the pop-up menu. This creates a new part with the selected faces being converted into surfaces. If you have multiple faces that have the same thickness and material, you can hold down the SHIFT key and select multiple surfaces before selecting 'Create>Extract Surface' from the right click pop-up menu. This creates a new part with the selected surfaces. After you are done creating the surfaces you need, select the solid part (this can be done in the Scene tree) and delete that.

The **Enable Visible Only** is disabled and not used for IronCAD.

4.1) The Material page

The Material page, ?? Default material, allows the material properties, physics allowed for this material and the bodies associated with a material to be changed. Initially the material is set to default material. This is the first material listed in the 'AFEMaterial.csv' file that contains the library of material properties. Contact Kubotek about editing this file to add materials commonly used by your company.

Default Material

Apply Cancel Sync Sim/Part

Source KeMaterial Edit Materials

Category Steel

Name Carbon AISI 1040 Annealed

☐ Non-Linear ☐ Orthotropic

Material Physics

☒ Stress ☐ Electro

☐ Thermal ☐ Fluid

Add Bodies Defaults

Total: 0 No Defaults Apply

Depending on the type of analysis (solid or shell) and the physics used, default values can be set for all bodies associated with this material using the **Defaults** button. You can override these defaults for individual bodies on the body page.

4.2) The Body page

Right clicking on the body leaf allows you to Disable/Enable a specific body for the current simulation.

The body page allows the Defaults (if any) specified on the material page to be overridden for this specific body. If Override Material Defaults is not checked, then the body defaults are grayed and the material defaults are used.

If the analysis is a Shell Element analysis, the shell thickness can be set separately for each surface part. In a thermal analysis, the expansion reference temperature and the heat generated in the body can be specified for each part.

☐ Override Material Defaults

Shell Thickness

T(no expansion)

Heat Gen

5) The Constraint Page

The Constraint Page and the Load-Source page allow the boundary conditions (BC) of the simulation to be specified. These include any initial values, fixed values, and pressures or forces applied to the model.

The constraint page lists the types of constraints that can be applied based on the physics set on the Analysis page. If Stress, Thermal and Electro physics are allowed, the constraint page includes the following options:

The screenshot shows a software interface for defining constraints. It features three main sections: 'Stress' with a 'Position and Rotation' button, 'Thermal' with a 'Temperature' button, and 'Electro (Cond)' with a 'Voltage' button. At the bottom, there is an 'Advanced Constraints' button. The interface is clean and uses a light gray color scheme.

For only stress analysis, just the Position and Rotation constraints are available.

5.1) The Fixed Position page (Position and Rotation constraint)

This page is added if the **Position and Rotation** button is clicked on the Constraint page.

The screenshot displays the 'Fixed Position' configuration page. It includes a 'Displacements' section with a 'Units' dropdown set to 'inch'. Below this is a table for defining displacement constraints:

Dir	Use	Displacement
X	<input checked="" type="checkbox"/>	0
Y	<input checked="" type="checkbox"/>	0
Z	<input checked="" type="checkbox"/>	0

Below the table is a dropdown menu set to 'XYZ' and an 'Update Entities' button. At the bottom is a large empty rectangular box for entity selection.

It allows various parts of the model to be fixed. A check in the **Use** box indicates that the coordinate direction is constrained. If the displacement is not zero, then the constrained node is displaced before calculating the stresses on the model.

To specify which entities are constrained by this boundary condition, click on the **Update Entities** button. This changes the **Update Entities** button to **Accept** and allows you to toggle the selection of points, edges, faces and solids in the model. To save the current set of selections, click **Accept**.

Note: You can double-click on an Entity in the Update Entities list. This highlights the Entity and gives you an option to delete the entity.

As a shortcut, if you want to completely change all of the selections, you can 'pre-select' the new selections before clicking on **Update Entities**. You will be prompted to replace any old items with the pre-selected items. Then make any further changes and click **Accept** to keep the currently selected items. A 'Cancel Update' button appears above the Accept button. The 'Cancel Update' button can be used to cancel updating the selected entities.

For more hints about entity selection, see '**J) Selection Hints**' at the end of the Analysis documentation.

Constraint Directions: XYZ pull-down

The XYZ pull-down can be used to change the coordinate system associated with this constraint. Whenever you do not use the XYZ option, local coordinates are set up for the nodes associated with this boundary condition. There are several cautions when using these local coordinates.

Caution 1: Each FEA node can only have one set of local coordinates and the same set of local coordinates can be used by other loads and boundary conditions that don't use the global XYZ directions. If multiple local coordinates are applied to a node, the last set of local coordinates (e.g. assigned by the last BC that uses that node) is used. Also, if a Load-Source system (Normal, Cylindrical or Spherical) is set up for a node, that coordinate system will be overwritten by a Constraint Coordinate system if used on the same node.

Caution 2: Displacements are always in distance units and always linear. For example, if Cylindrical coordinates are used, a displacement of .5 (inch) in the Th (Theta) direction means a displacement of .5 inches along the local theta direction defined at the node and not a rotation of .5 radians around the axis.

The Right Click Menu

Right clicking on a BC (Fix ... etc.) allows the following options:

Disable/Enable: This allows the BC to be temporarily disabled and not used in the next analysis.

Move Down: This allows the BC to be moved down. If conflicting BCs are applied to the same node, the last BC is used. For example, if two faces sharing a common edge are assigned different temperatures, the nodes on the common edge would be assigned the temperature corresponding to the lower BC in the list.

Delete: This allows a boundary condition to be deleted.

6) The Load-Source Page

The Load-Source page is used to apply loads (forces and pressures) and sources (heat flux, electrical current) to the model. Depending on the physics selected on the analysis page, various buttons will be available.

The screenshot shows a software interface for applying loads and sources. It is organized into several sections:

- Stress:** Contains two buttons: "Force" and "Pressure".
- Thermal:** Contains four buttons: "Heat Flux", "Heat/Volume", "Radiation", and "Convection". Below these is a "Radiation View" button.
- Electro (Conduction):** Contains one button: "Current Density".
- Acceleration:** Contains one button: "Acceleration".
- Rotation:** Contains one button: "Rotation".
- Checkboxes:** At the bottom, there is a checkbox labeled "Loads not needed".

6.1) The Force Page

This page is added if the **Force** button is clicked on the Load-Force page.

Similar to all BC pages, the right click menu allows this BC to be disabled and not used for the next analysis.

Force ▾ Units

100 lbf ▾

Reverse Dir Set Direction

x= 1.0000 y= 0.0000 z= 0.0000

Fixed ▾

Update Entities

The **Set Direction** button allows the force direction to be set in global (XYZ) coordinates. On the Set Direction page, the **Get values from Edge** button allows the direction to be set from any straight edge in the model. The **Reverse Dir** button can be used to reverse the direction of the force.

The **Fixed** pull-down allows the force to be specified in Fixed, Cylindrical or Spherical coordinates.

Caution: If Cylindrical or Spherical coordinates are used, then local coordinates are defined for the node. Care should be taken to insure that these don't interfere with any other local coordinates defined at the same node. Constraint local coordinates will override Load-Source local coordinates.

7 Mesh

Before a FEA model can be analyzed the cad model must be converted into an FEA mesh. The mesh should be small enough to capture the important geometry of the model but not so small that the solution times become too large. The Number of nodes and Elements and the mesh size is displayed for the last mesh that was created. The **Create Mesh** button is used to set the mesh parameters and create the mesh.

The image shows a software interface for meshing. At the top is a button labeled "Create Mesh". Below it are two buttons: "Add Refinement" and "Show/Hide Mesh". Under these buttons, the text "Last Mesh:" is followed by "790 Nodes and 2731 Elements" and "Mesh Size:0.35". Below this is a section titled "For Non-Grouped Bodies" containing two radio buttons: "Glue" (which is selected) and "Independent". At the bottom of this section are two buttons: "Add Glued Group" and "Add Independent Group".

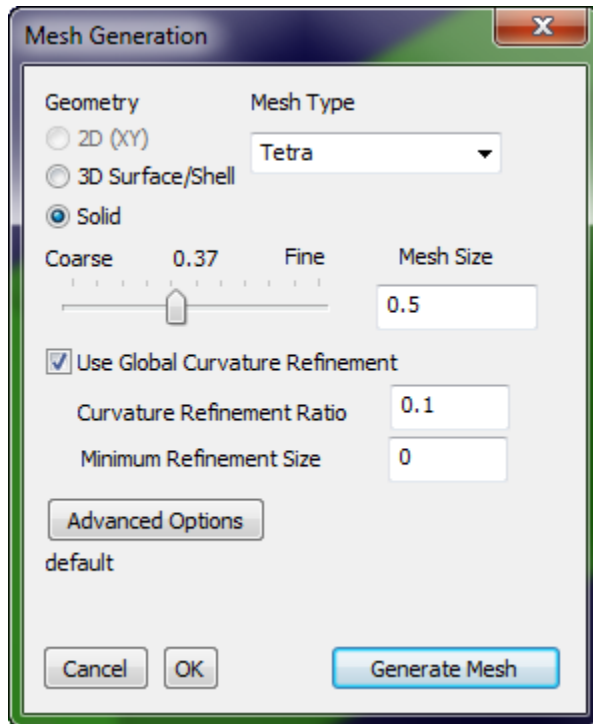
Glue/Independent option determines what happens at shared edges and surfaces. If **Glue** is selected, one set of nodes is used for both bodies on the common surface so they must move together. They act as if they were glued together. If **Independent** is specified, each body has a separate set of nodes. Unless 'a contact' or 'tie/glue' BC is applied, each body will act completely independently. They can separate or even pass through one another. If a 'contact' boundary condition is set, then the processor checks to see if the bodies overlap. If they do, the necessary force is added to keep the bodies apart. Contact works best when dynamic analysis is used as the inertia of the bodies helps to stabilize the solution. If a 'tie/glue' BC is applied, to coincident faces on two bodies, the faces act as if they were tied or glued together. An offset or transformation can be used before determining 'coincidence'. Tie/Glue with an offset provides a way of imposing periodic boundary conditions.

The **Add Glued Group** adds a Glued Group page under the mesh page. Bodies in the Update Bodies list for this page are glued together but not glued to any other bodies in the model. Similarly the **Add Independent Group** button adds a page that includes a body list of bodies that are never glued to other bodies – even if the **Glue** option is set.

The **Add Refinement** button can be used to refine (create a smaller mesh) specific edges or faces of the model. This is useful if a coarse mesh is adequate for most of the model, but a finer mesh is needed for a small section of the mesh. If the small mesh size were used for the entire model, excessive FEA elements would be created and the time (and computer memory) necessary to solve the simulation would be much larger.

7.1 Mesh Generation dialog

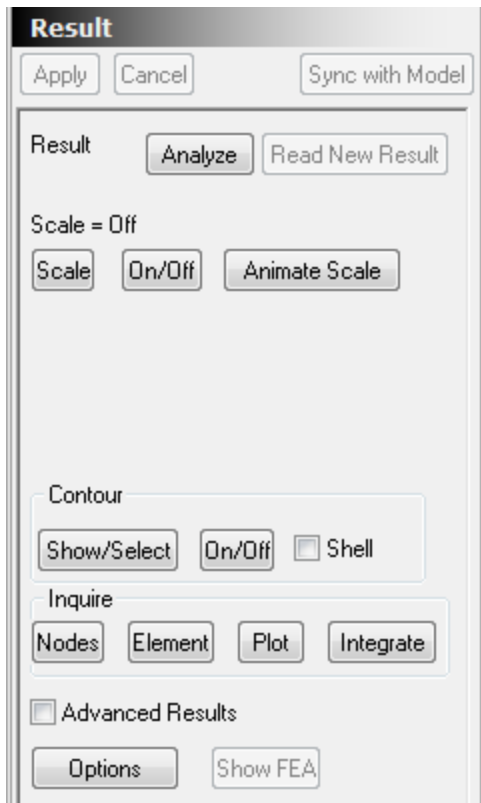
The Create Mesh button opens the main meshing dialog. This allows you to specify the mesh size and any curvature refinement. The **Generate Mesh** button on this menu creates a new FEA mesh for the model.



The Mesh Type pull-down allows you to select Tetra or Hexa element. Tetra elements are recommended if you are using Sefea™ technology to solve the model. If Hexa elements for solids or quad elements for shell bodies are used, the processor defaults to Standard technology and does not use Sefea™.

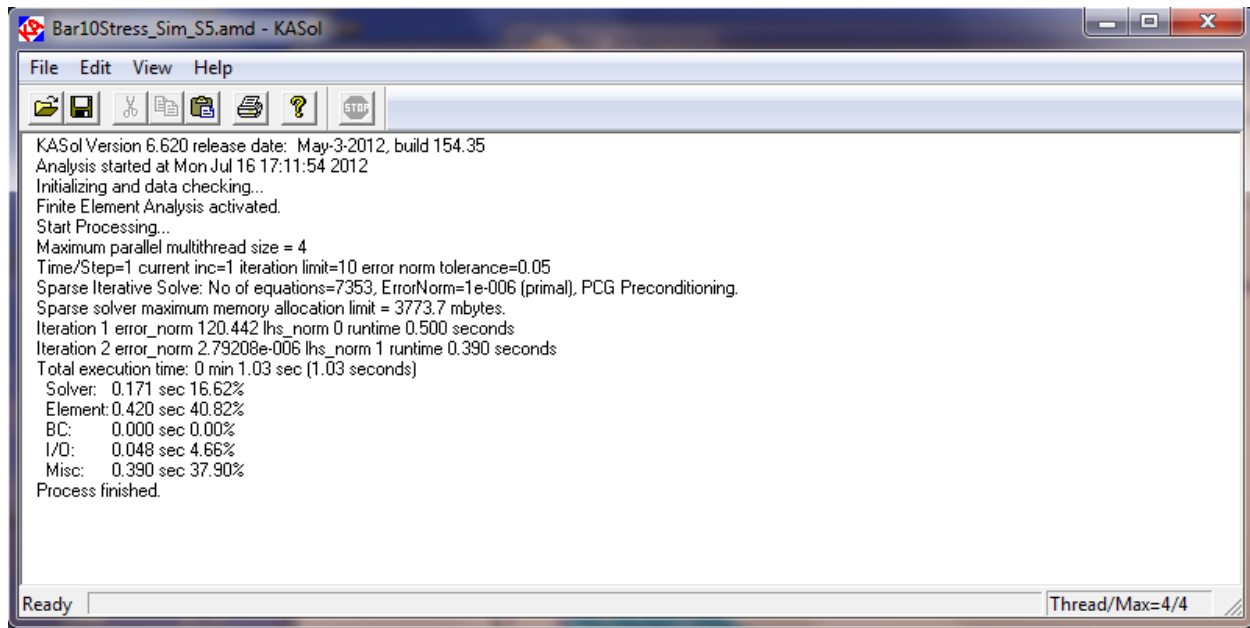
8 Results

Once the model has been meshed, the Result page is used to analyze (or re-analyze) the model and examine the results.



8.1 Analyzing the model

If the **Analyze** button is enabled, then the model has not been analyzed or has been changed since the last analysis. Clicking on the **Analyze** button starts the processor and begins analyzing the model. The processor displays a progress window (see below). The 'Strain-Enriched Finite Element activated' message indicates the Sefea™ is being used. The Iteration number and error_norm can be checked during analysis to verify that the analytic solution is converging.



Once 'Process Finished' is displayed, the processor (KASol) window can be closed.

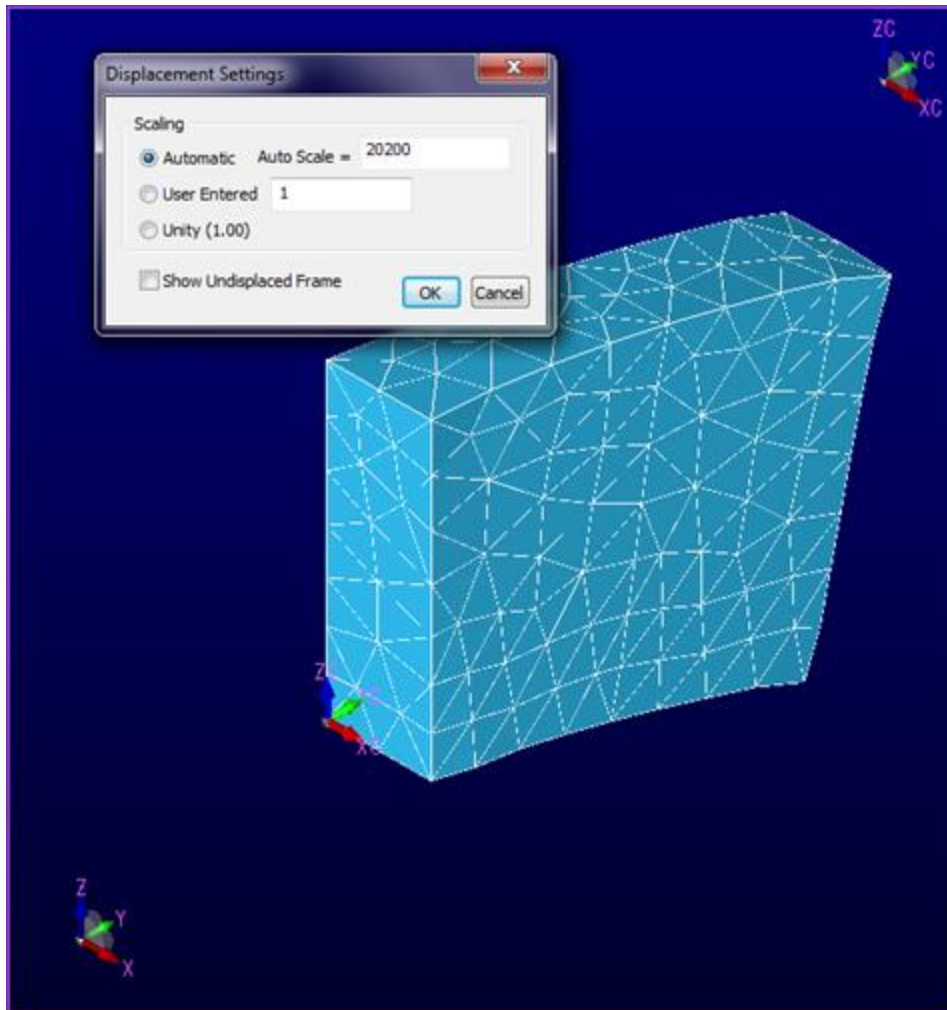
8.2 Re-Analyzing the model

There may be times when you want to re-analyze the model but the **Analysis** button is not enabled. This can occur if a previous analysis was aborted after it started. Click on the **Options** button on the Result page and then click **Re-Analyze** to run the analysis again.

8.3 Viewing the Scaled results

For a stress model it is often useful to exaggerate the deflections to make the deflected shape more obvious. The **Scale** and **Animate Scale** buttons are used for this. The **On/Off** button after the **Scale** button can be used to toggle the latest scaled deflections on and off. The Animate Scale button provides an animation by changing the scale factor for the displacements.

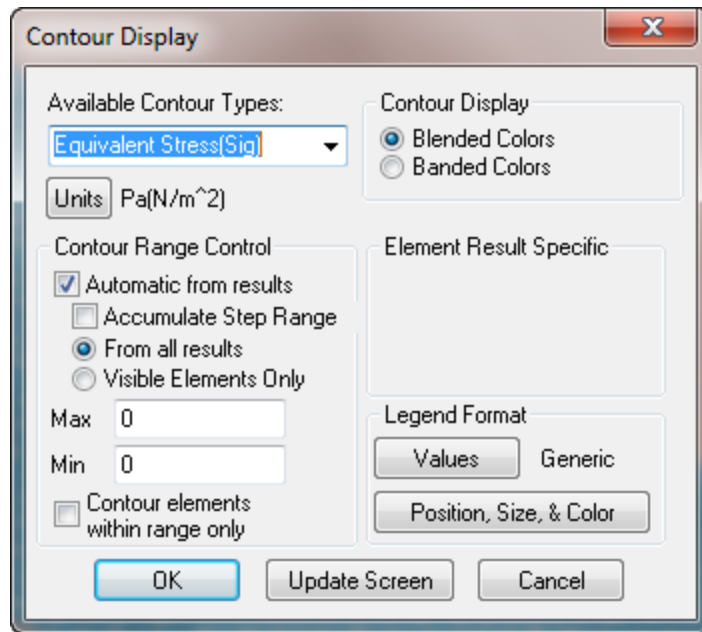
The Automatic option on the Scaling dialog works well to choose a reasonable scaling factor.



8.4 Contours of Output Values

The processor generates a file with output values at each node (the .oub file). The model can be colored with contours that represent these nodal values interpolated to the surface of the model or on cutting planes through a solid model.

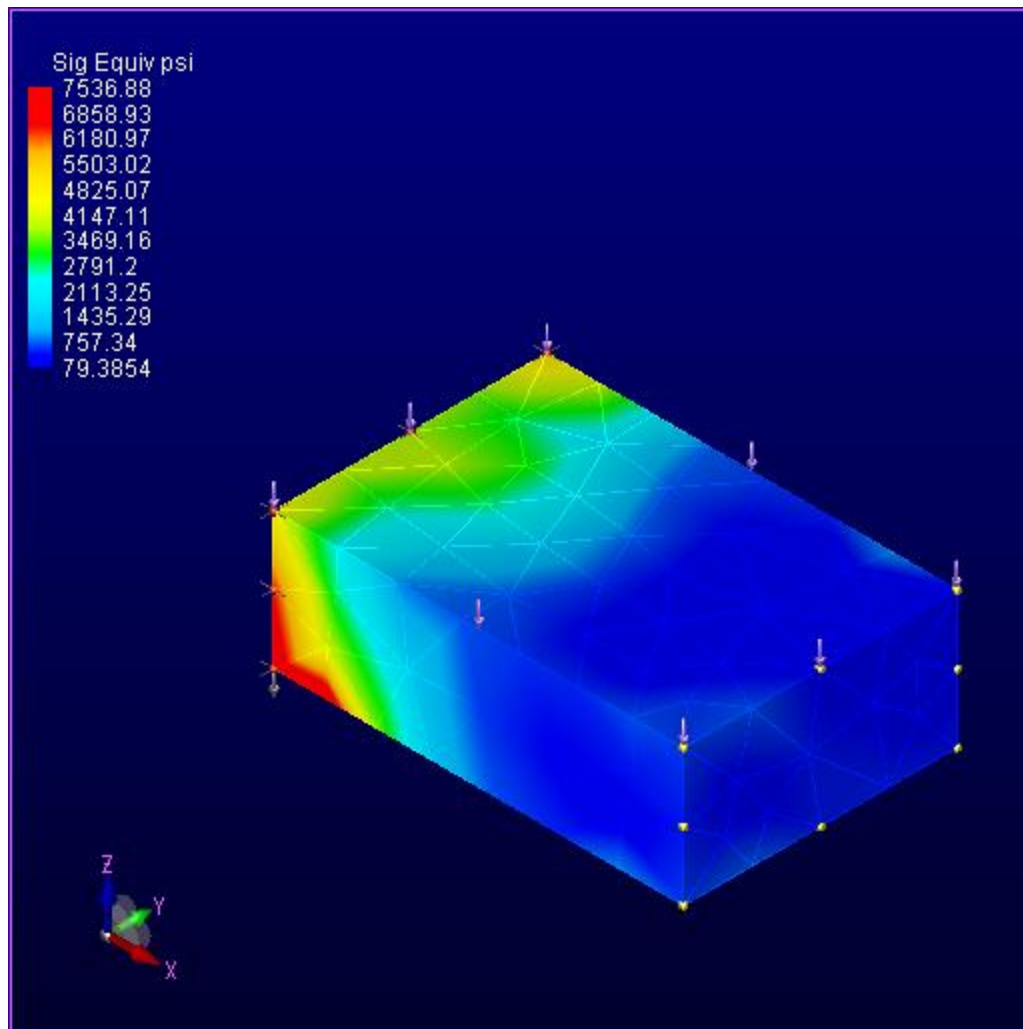
Select **Show/Select** under Contour to select the values to be contoured. The Contour Display Control dialog allows you to specify the output property to contour and other options that affect the location, format and range of values contoured:



The Contour Types pull-down is the control that you're most likely to change on this dialog. Initially this displays a list of the most common output options. If the **Advanced Results** checkbox on the Result page is checked, the list includes all the output options generated by the processor.

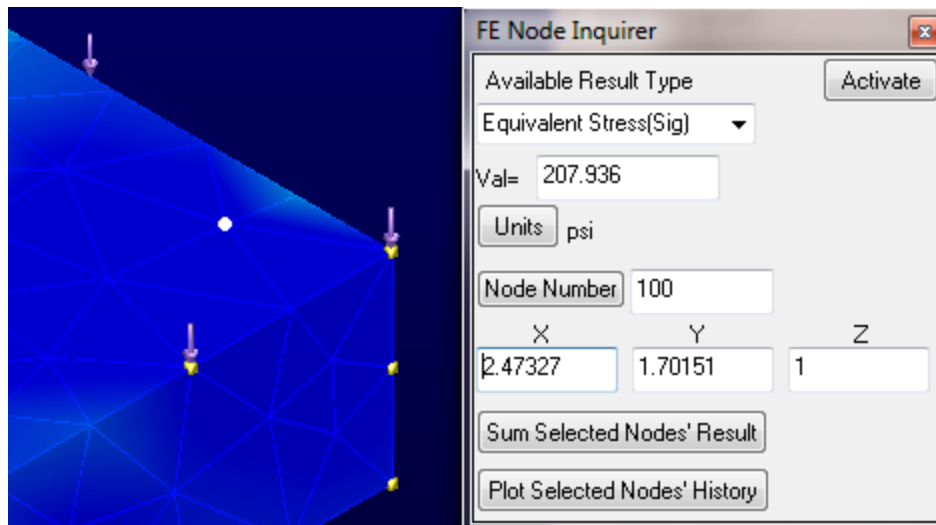
The Units button on the Contour dialog allows different output units to be used for the next contour. Updated output units are shared between the various results options (Node Inquire and Plot) and are saved as part of the simulation when the 'Save Simulation' button is used.

The following image shows the Sig Equiv (similar to Von Mises) stress for the model in psi. The Units button on the Contour dialog allows different output units to be used for the next contour. Updated output units are shared between the various output options (Node Inquire and Plot) and saved as part of the simulation when the 'Save Simulation' button is used.



8.5 Inquiring about Nodal Values

The **Nodes** button in the Inquire section can be used to open a dialog that allows values at specific nodes to be displayed using the mouse.



A white marker is used to highlight the node that corresponds to the values in the FE Node Inquirer dialog. The Result Type pull-down contains common output values if **Advanced Results** on the Results page is NOT checked and contains all the output values if **Advanced Results** is checked. Note that if Contouring is on, the FE Node Inquirer result type initially matches the contouring result type. But, it can then be changed independently of the contouring result type.

If no valid results are available, it is still possible to inquire about node positions using the FE Node Inquirer. To select and view a node based on the node number, just enter the value next to the **Node Number** button and click on **Node Number** (or press the <Enter> key after entering the number).

8.5.1 Summing Values at Selected nodes

The FE Node Inquirer dialog also allows the current Result value to be summed over the selected nodes. Use the selection toolbar:



To select the face or edge that contains the nodes to be summed. After you are done with selection, the **Activate** button on the Inquirer dialog can be used to change the mouse operation from selection to Node Inquiry.

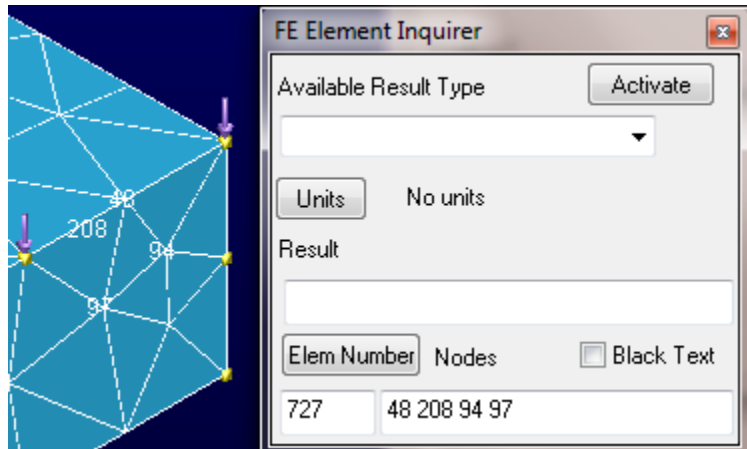
Note, if the Analysis Toolbar is not visible,

8.5.2 Plotting Values at Selected nodes

As a shortcut, an option to **Plot Selected Nodes' History** button is included on this dialog. This is more useful when a Dynamic or Multi-Step analysis is used so that output steps represent the values at the nodes as a function of time (or pseudo time). Other plotting options are available using the **Plot** button on the Result page.

8.6 Inquiring about Element Values

This option can be used (even without valid results) to display the nodes associated with each element. Just click on the **Element** button below Inquire to bring up the FE Element Inquirer dialog and move the mouse over an element.



This can be used to verify that common nodes (between elements of different materials) have the same node number when the 'Glued' option is used to mesh the model.

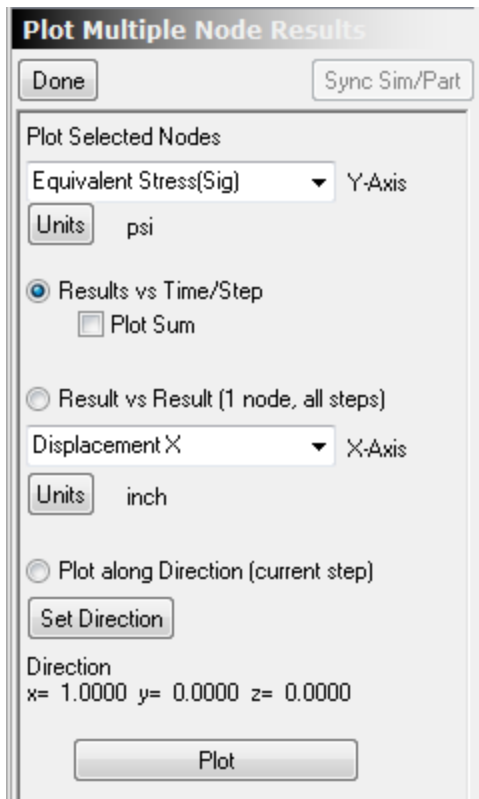
If analysis of a shell model is performed, the Inquire Element option can be used to display the top and bottom values associated with the shell elements.

8.7 Plotting nodal values

The **Plot** button in the Inquire section of the Result page can be used to bring up the plotting dialog (see below) to create various types of plots of nodal values. Three types of plots are available. The first two types are designed for multi-step (dynamic or non-linear) models that have multiple output steps. The last method plots the values of multiple nodes as a function of distance along a specified direction at the current time/step.

The selection toolbar should be used to select the desired geometry that contains the desired nodes before clicking the **Plot** button on the Plotting dialog:

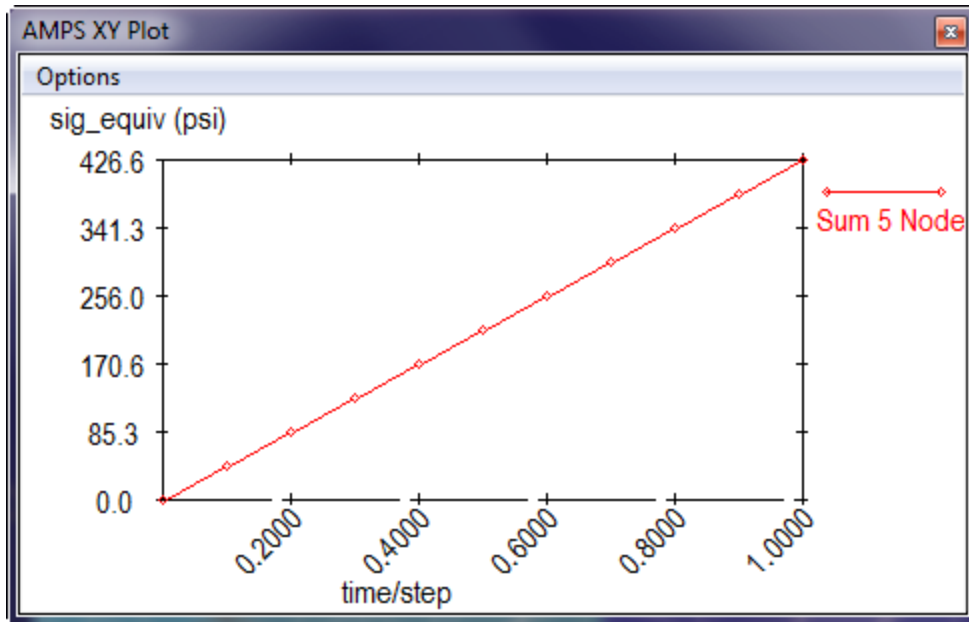




8.7.1 Results vs Time/Step

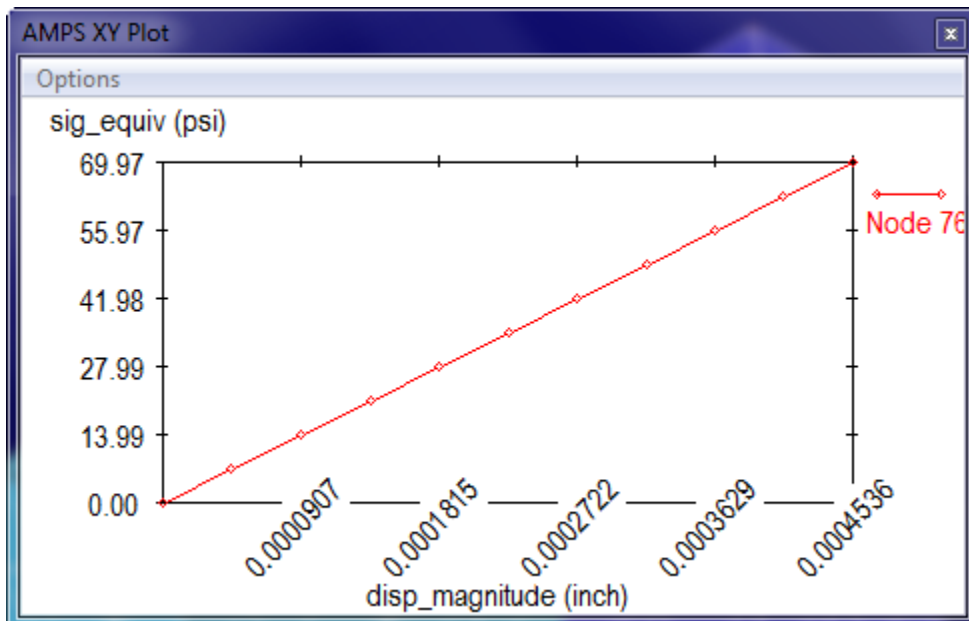
This option is the same as the **Plot Selected Nodes' History** shortcut button on the Inquire Node dialog. The pull-down allows the output parameter to be changed. If **Advanced Results** is checked, the pull-down displays all the output options generated by the processor.

The Plot Sum checkbox (not available from the Inquire Node plot option) allows the sum of the results for all the selected nodes to be plotted as a function of time:



8.7.2 Results vs Results (1 Node, all steps)

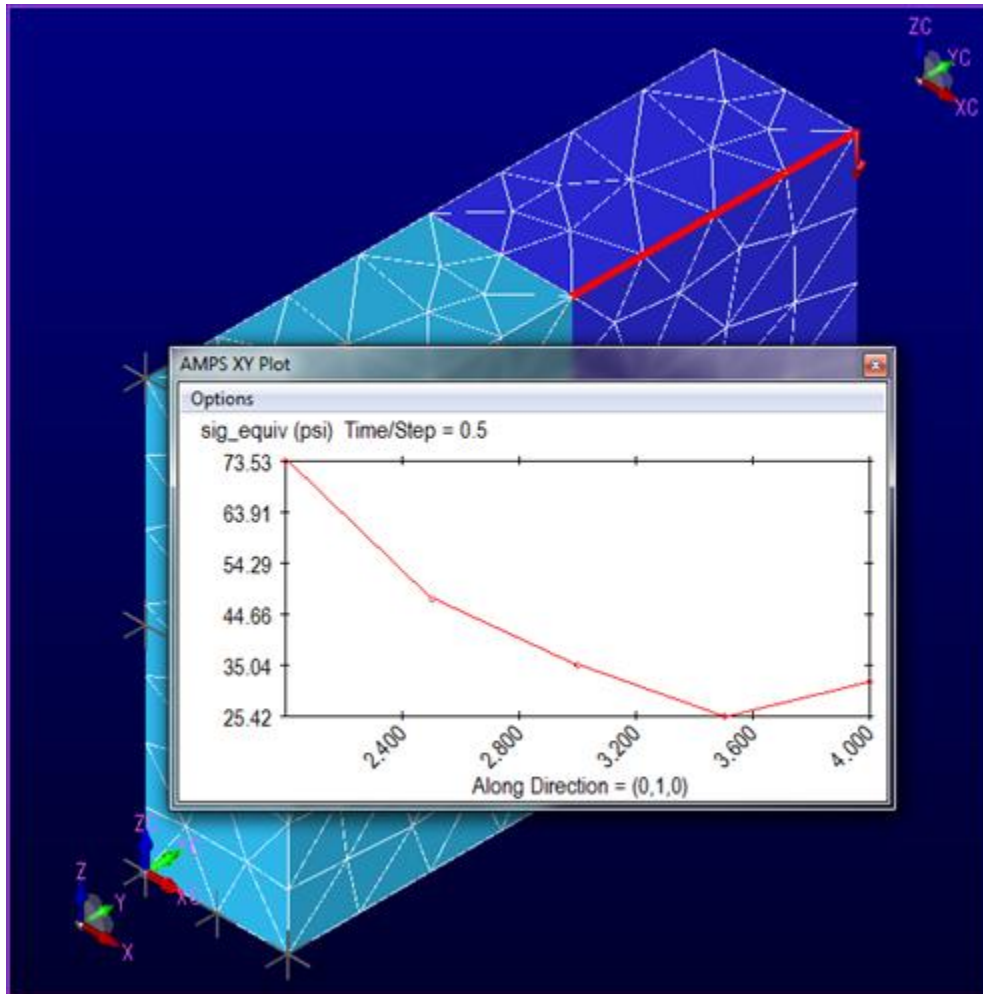
This option can be used to plot a curve of paired result values evaluated at one node at different time/steps. The vertical axis quantity is obtained from the top result pull-down and the horizontal axis quantity is specified by the second results pull-down. For example, a plot of sig_equiv vs disp_magnitude for node 76 for various time/steps is shown below:



8.7.3 Plot along Direction (current step)

If the Plot along Direction (current step) option is chosen, then the results specified in the top pull-down is displayed for all the selected nodes as a function of distance along the specified direction. The **Set Direction** button can be used to specify the direction manually, as an axis or based on an edge in the

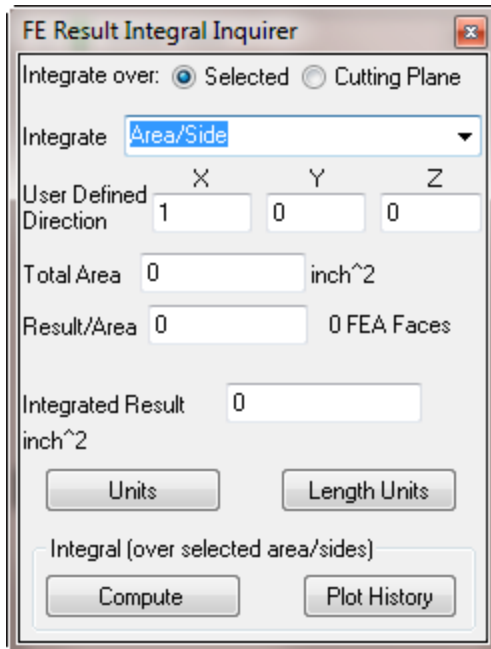
model. In the following plot, the sig_equiv values of the selected nodes are plotted as a function of the position of the nodes in the Y direction:



8.8 Inquire Integrate

The **Integrate** button below Inquire is used to bring up the Fe Result Integral Inquirer dialog. This dialog allows output quantities to be integrated over selected surfaces or over a single cutting plane through the model.

Even if no results are available, the Integration dialog can be used to calculate the surface area of a selected FEA surface or the area of the FEA model that intersects a cutting plane. Note that all integration in this dialog is based on the FEA model and not the original CAD model.



The Integrate pull-down includes not only the standard output from the processor but the following special quantities:

Area/Side

The Area/Side option is available even if no analysis has been performed. It provides the area of the selected surfaces.

Heat Flow (Surface Normal)

This is the heat flowing through the selected surfaces.

Heat Flow (UserDir)

This is the heat flowing in the specified direction integrated over the selected surfaces.

Electric Current (if Electro:Conductive physics used)

This is the total electric current through the selected surfaces (in Amps)

Electric Charge

This assumes the selected surface is an outer conducting surface (so the electric field inside is zero). It provides the charge on the surface.

CoulombForce (UserDir)

This assumes the selected surface is an outer conducting surface (so the electric field inside is zero). It provides the force due to the charge on the surface.

Normal Force (SurfaceNormal)

This is the force normal to the surface.

Shear Force (UsrDir)

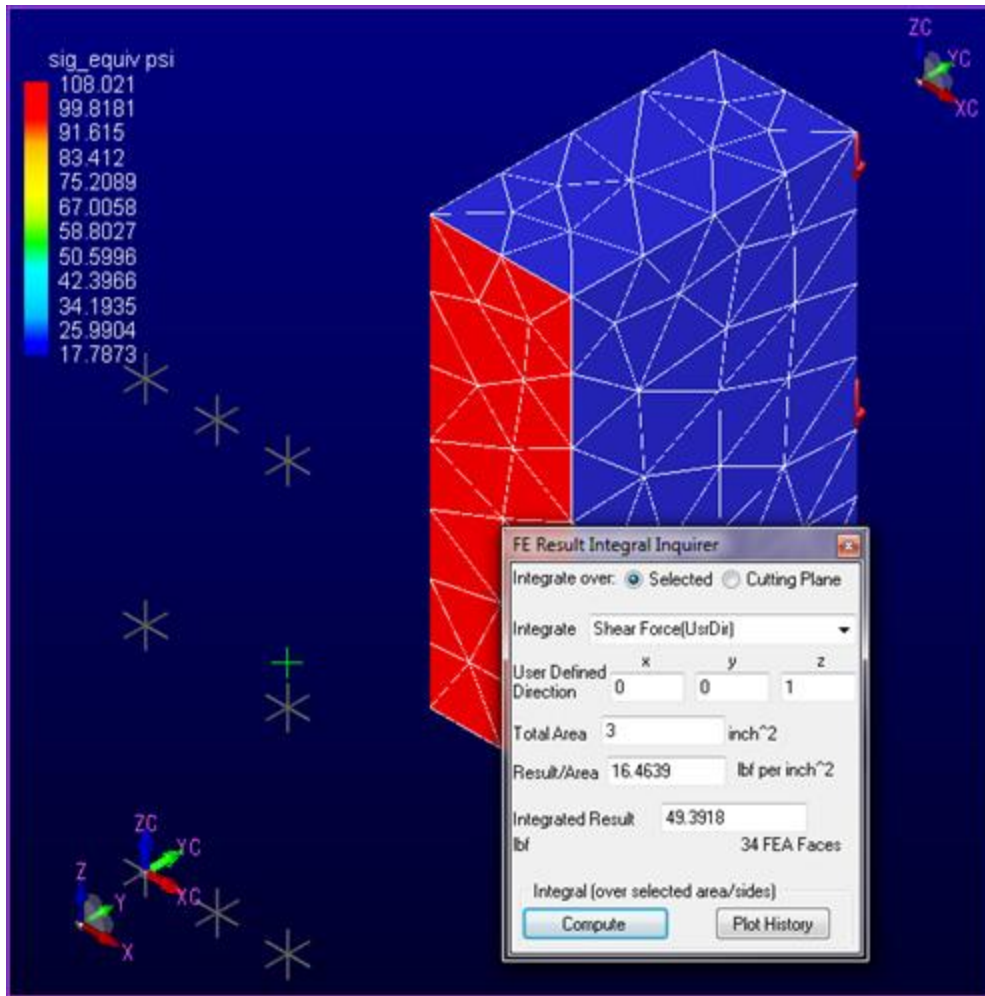
This is the component of the force in the specified direction integrated over the selected surfaces.

Note: These special quantities are always available on the menu but display a warning if the associated physics is NOT included in the analysis.

8.8.1 Integrals on Selected Surfaces

The following figure displays the Shear Force in the Z direction integrated over an inner surface in a model made of two materials:

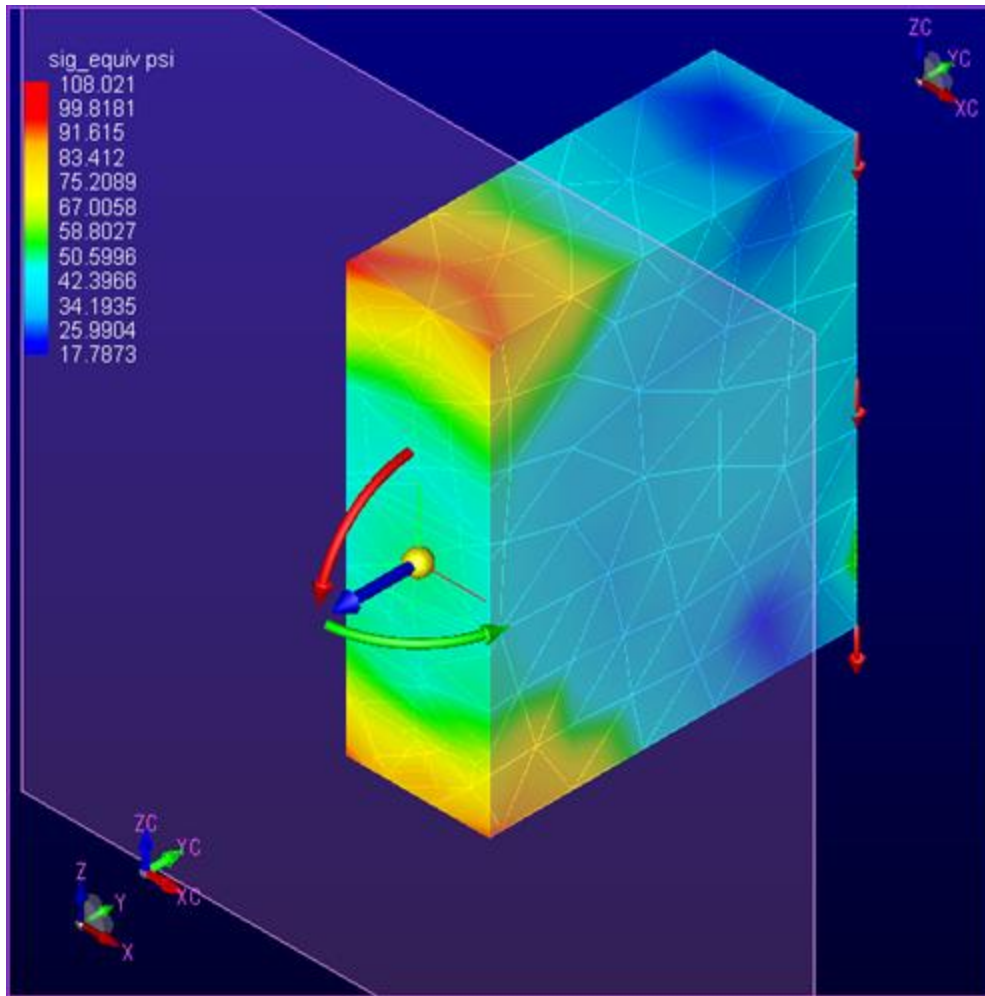
In order to select the inner surface, the solid body that covered the surface was selected (using the Sel Solid selector) and then hidden using the **Fe Hide Select** menu option from the pop-up menu displayed by right clicking in the model window. Then the Sel Face selection option was used to select the inner face. The **Fe UnHide All** command from the same right click menu can be used to re-display the parts of the model that were hidden.



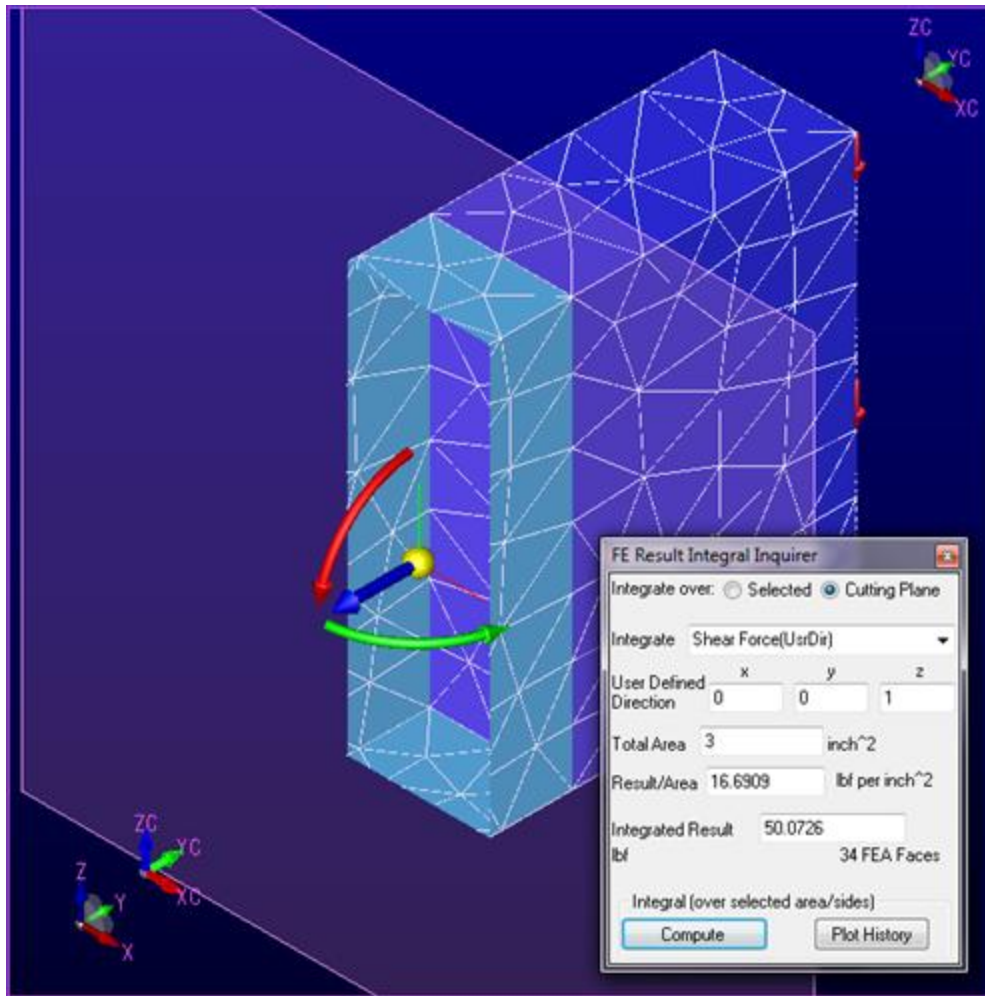
8.8.2 Integrals over a cutting plane

**** CUTTING PLANES are not yet available for IronCAD FEA analysis. ****

If contouring is turned on, the contour over the cutting plane is included and updated as the cutting plane is moved:



Even if contouring is turned off, the Integration dialog can be used to calculate the integral of the specified result value over the area of the model that intersects the cutting plane. To do this, you must have exactly one cutting plane and the **Cutting Plane** radio button in the top right of the integration dialog must be selected. The following figure shows the same Shear Force integral calculated at a cutting plane slightly offset from the inner plane that was previously selected:

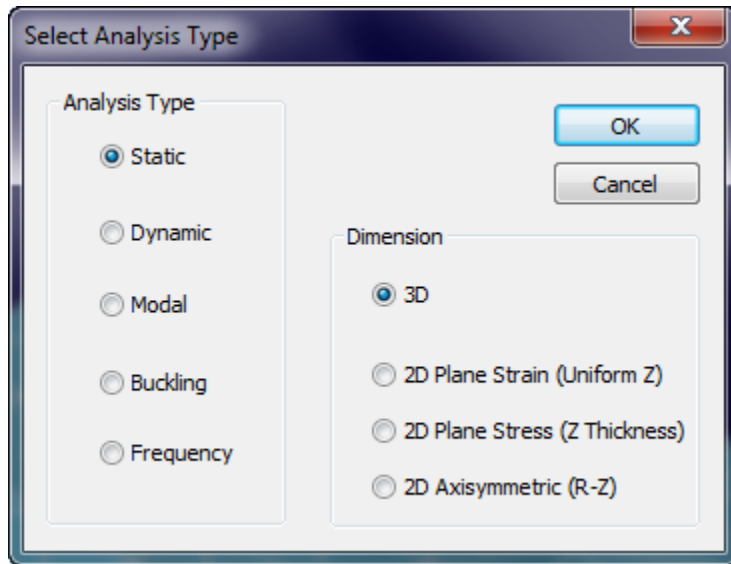


8.9 Result Options

The Options

B) Analysis Types Dialog

This dialog is displayed when you create a new FEA simulation or when you use the right click on the Analysis (top) leaf and select **Copy Sim**.



Analysis Types

1) Static

Static linear analysis is used when a static or steady state solution is desired. None of the output or input variables should be time dependent.

1.1) Non-Linear Static

If non-linear is checked on the Analysis Page, then a pseudo-time is used to correctly (usually slowly) apply the boundary conditions. The time dependency controls the magnitude of prescribed displacements, forces, pressures, temperatures, heats and voltages.

2) Dynamic

Dynamic analysis represents a step-by-step integration of the model as a function of time. The time dependency controls the magnitude of prescribed displacements, forces, pressures, temperature, heat and voltage.

3) Modal (includes Stress Stiffening)

This performs a standard modal analysis to extract the desired modes of vibrations. Before the modal analysis, a static analysis is performed and the stress stiffening effect is included for the modal analysis. The generalized eigen problem is $(\mathbf{K} + \mathbf{K}_s) - \lambda \mathbf{M} = 0$, where \mathbf{K}_s is the additional stress stiffness matrix computed during the initial static analysis.

Stresses can affect the modal response of the model. A Stress Stiffening option is available on the Analysis page for Modal Analysis. This, if available based on the user's license, allows modal analysis with stress stiffening.

4) Buckling

This calculates the linear Euler buckling modes of the system. The generalized eigen problem is $K - \lambda K_s = 0$, where K is the system stiffness and K_s is the stress stiffness matrix.

Buckling Analysis can be performed on 3D stress models with solid or shell elements.

5) Frequency

Frequency analysis displays the response of the system to sinusoidal forces applied at a range of frequencies. The frequency range start, end and increment are entered on the Analysis page in Radians/sec. Each output step corresponds to the response of the system to the forces applied at the specified frequency. The Animate Scale option animates the response as a function of phase angle of the sine wave.

Dimensions (3D)

3D is the option chosen for all analysis of cad models made up of solids or surfaces that are in more than one plane. Forces and responses can vary in all three directions.

Dimensions (2D)

The 2D options allow a cross section of a 3D model to be analyzed. For 2D analysis, the CAD model must be surfaces created in the XY plane. In cases where nothing changes in the 3rd direction (Z or the theta direction for cylindrical models), 2D analysis allows the FEA model to have significantly more resolution in the two dimensions of interest.

2D Plane Strain (Uniform Z)

2D Plane Strain models represent an XY cross section of a model assumed to be infinite in the Z direction. Examples would be the cross section of a dam or the cross section of a pipe. Boundary Conditions for static analysis are applied to the edges of the XY surfaces and are assumed to apply to the infinite surface created by extending the edge in the plus and minus z direction.

2D Plane Stress (Z Thickness)

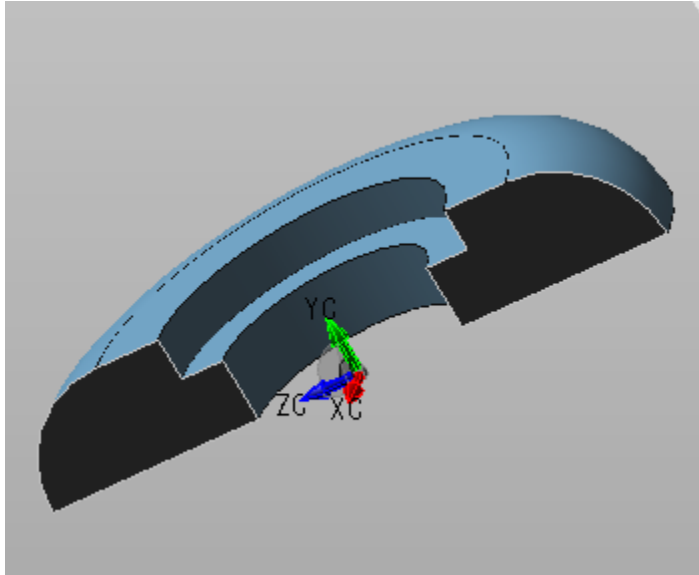
2D Plane Stress models represent an XY cross section of a model where all the stresses (and consequently deflections) are assumed to be in the XY plane. Each surface (sheet body) in the XY plane can be made up of different material and can have a different thickness. Boundary conditions for static analysis are applied to the edges of the XY surface.

The 'Copy Sim' command can be used to convert a 2D Plane Stress model to a 3D model (which will be a shell model). The edge pressures should be converted to edge forces for the 3D model and forces in the Z direction can be applied.

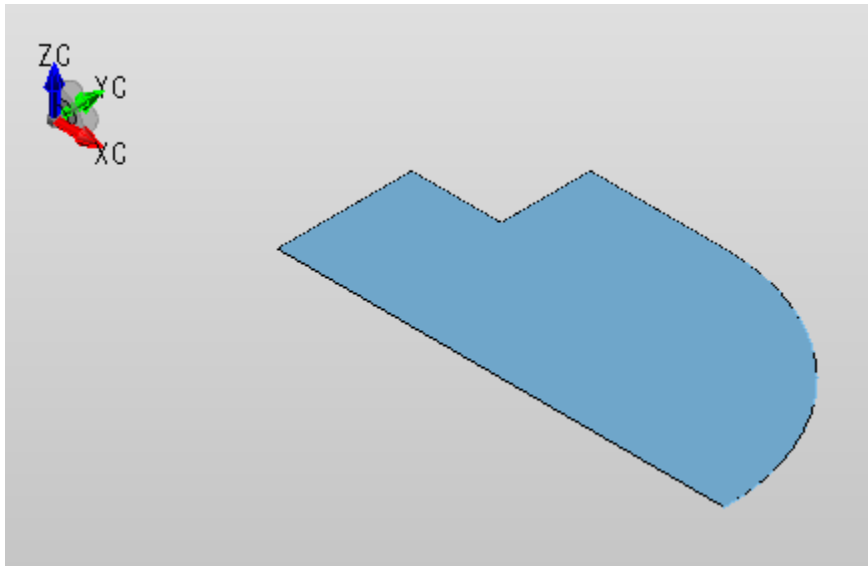
2D Axisymmetric (R-Z)

For 2D Axisymmetric models, the physical model corresponds to the XY plane cross section being rotated 360 degrees around the Y axis. For example, a circle of radius 1 at $X=2$ would correspond to a torus (donut) with a major radius of 2 and a minor radius of 1.

The following shows a cutting plane view of a gasket that is symmetric about the Y axis.



For 2D Axisymmetric analysis, the following XY surface would be used:



2D Axisymmetric analysis could be used to determine the expansion of the gasket if the bottom was fixed in the Y direction and the top subjected to a fixed pressure.

B1) The Analysis Page

This page is displayed with the top (Analysis) leaf of the FEA tree is selected. Based on the analysis being performed, the title and options change. For Linear Static analysis the following page is displayed:

Analysis: Static

Apply Cancel Sync Sim/Part

Title

Physics

☒ Stress ☐ Electro

☐ Thermal ☒ Conductive

☐ Fluid ☐ Dielectric

☒ Multi-Step ☐ Large Deformation

Non-Linear Pseudo-Time

Start 0

Finish 1

Increment 0.1

More Increments: 0

Advanced Opt

Adv: Default

The **Title** can be any single line description of the simulation being performed.

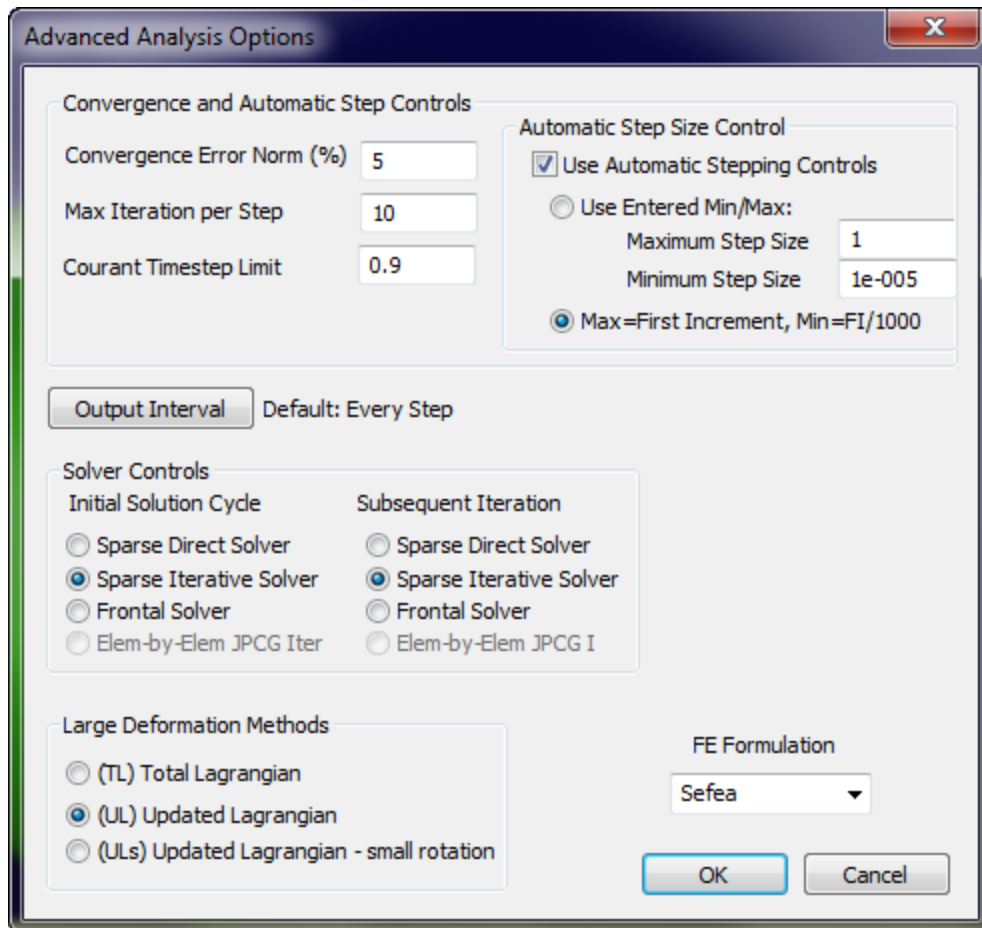
The **Physics** options represent the physics that is activated for this simulation. There should be at least one active material to support each physics option used in a simulation. Not all materials need to support all physics. For example, if joule heating is used, only the materials that represent the heating elements need to support 'Electro' physics. Usually each type of physics included in the analysis needs a constraint to provide a fixed value for that physics for the analysis.

If **Multi-Step** is checked, then a pseudo-time is used so that the BCs can be applied gradually. Also, large deformation of the elements is supported.

A **Large Deformation** option is available if Static with Multi-Step is used or if Dynamic analysis is used.

Advanced Analysis Options

This dialog is displayed by clicking on the **Advanced Opt** button of the Analysis page. It allows many of the advanced options associated with the FEA solution method to be changed.



Large Deformation Methods

The Large Deformation Methods radio button determines which geometry formulation is used when the Large Deformation box is checked on the Analysis Page. The default is Updated Lagrangian (UL) and this should be good for most non-linear problems.

Total Lagrangian (TL) and Updated Lagrangian (UL) (practical)

The TL and UL methods produce effectively the same results. The errors in the UL method are smaller but accumulate from step to step. The errors in the TL method can be larger (due to rotation approximations) but do not accumulate. The TL method is better for Hyper-Elastic Materials, but the FEA processor automatically switches from UL to TL if Hyper-Elastic Materials are used.

Updated Lagrangian (ULs) Small Rotation Approximation

This option causes the UL formulation to neglect the rigid body rotational matrix correction. This correction is typically small if the rotation between successive non-linear steps is small and ignoring the correction slightly increases the computation speed. On some models with very nonlinear behavior or multiple solutions, turning on *Small Rotation Approximation* will improve the numerical convergence by slightly smoothing these near singularity calculations.

Total Lagrangian (TL) and Updated Lagrangian (UL) (technical)

For large deformation and rotation, either Total Lagrangian Formulation (TL) or Updated Lagrangian Formulation (UL) can be used. The TL always uses the original state (displacements = 0) for the frame of reference, and UL uses the most recent state (latest displacements) for reference. In the TL formulation, the Green-Lagrangian strains and the 2nd Piola-Kirchhoff stresses are used, while the Cauchy stresses and the infinitesimal strains are used in the UL formation with respect to the most recent deformed state. Although the internal stress and strain calculated using these two methods are completely different, the converged results will be similar since the final result is expressed in the engineering Cauchy stresses and strains.

Max Iteration limit per step, Convergence Error Norm

This will limit the iteration during a equilibrium iteration in a single step. An equilibrium check is performed to ensure the error norm is less than the **Convergence Error Norm (%)**. The **Courant Timestep Limit** is only used for the (not available) Explicit solver.

Automatic Step Size Control

This is the automatic time step control. During an analysis step, if the analysis failed to converge in the specified error norm, and the **Use Automatic Stepping Controls** check box is selected, the program will automatically decrease the time step size and try to solve the problem again using this refined step size. This process will continue until a successful analysis can be established. The program will then gradually increase the step size to the maximum step size, the next display increment occurs or the analysis fails to converge.

If the **Max=First Increment, Min=FI/1000** radio box is checked, then the First Increment (on the analysis page) is used as the Maximum Step Size and the First Increment /1000 is used as the Minimum Step Size..

Optional Result Output Interval Controls

Normally, the processor will write a result data set for each successful time/step increment for FEA to display the results. Sometimes in the case of transient dynamic analysis with a small timestep, it is desirable to skip several simulations steps before saving the output results. This button provides a way to specify the output interval controls. In such cases, the processor will only write out the results for FEA after the time/step increment reaches the specified count.

The default is to generate output for every step.

FE Formulation (Standard, Sefea™)

This option allows the user to specify the Standard FE formulation or the new Sefea™ element formulation. The **Sefea™** option is a proprietary AMPS option that provides the same accuracy as highly optimized hexahedral (brick) elements, but uses the standard first order tetrahedral elements generated by automatic meshing systems.

It is no longer necessary to use hexahedral meshes that are carefully crafted in order to correctly solve many difficult problems. The new Sefea™ method provides the same accuracy using standard tetrahedral elements.

Also, you do not check *Generate 2nd Order Lagrangian Elements* as simple first order elements work best with the Sefea™ method.

The **standard** formulation is the formulation used in most FEA programs. The first order tetra (and triangular) elements are not as accurate as the 2nd Order Lagrangian Elements but can be used for a quick, rough estimate of the results. The 2nd Order Lagrangian Elements require much more memory and processing time than the first order elements.

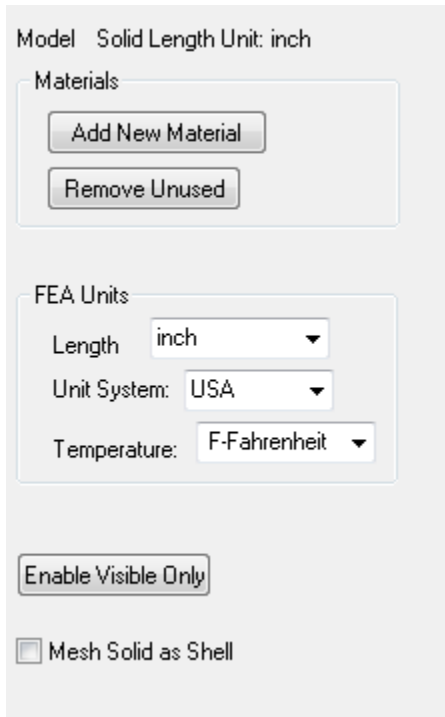
Solver Controls

The user is given the option to choose the type of procedure to be adopted in the initial iteration and subsequent iterations in each analysis step. The sparse direct solver uses the most up to date sparse matrix solution scheme. In general, sparse direct solver should be used since there is no approximation involved. The iterative sparse solver is the default solver and allows a solution speed up by a solution error tolerance control, and can usually achieve a faster solution time.

The frontal direct solver utilizes the element-by-element assembly and factoring technology to reduce the memory needed for solving large problems and is provided as a backup to the sparse direct solver. And for extremely large problem with limited memory constraint, the element-by-element Jacobian Preconditioned Conjugate Gradient (JPCG) solver can be used since it uses a matrix free iteration method, but is generally slower than the sparse iterative solver.

C) The Model Page

The Model Page is displayed by clicking on the Model leaf of the FEA tree.



The screenshot shows the 'Model' page in IronCAD. At the top, it says 'Model Solid Length Unit: inch'. Below this is a 'Materials' section with two buttons: 'Add New Material' and 'Remove Unused'. Further down is the 'FEA Units' section, which contains three dropdown menus: 'Length' set to 'inch', 'Unit System' set to 'USA', and 'Temperature' set to 'F-Fahrenheit'. At the bottom of the page is a button labeled 'Enable Visible Only' and a checkbox labeled 'Mesh Solid as Shell' which is currently unchecked.

Add New Materials and Remove Unused

The **Add New Material** button is used to add more material pages and then the **Add Bodies** button on the material page is used to assign bodies to the new material. The **Remove Unused** button is used to remove material pages that do not have any bodies assigned.

FEA Units

Initially the units are based on the length unit chosen for the IronCAD model and all bodies are assigned to one material (the first material in the AFEMaterial.csv file).

Enable Visible Only

*** Not implemented for IronCAD **

The **Enable Visible Only** button is used to enable only the bodies that are currently visible in the Solid Model.

Individual materials or bodies can also be enabled or disabled by right clicking on the tree leaf and selecting Disable/Enable to toggle the Disable flag.

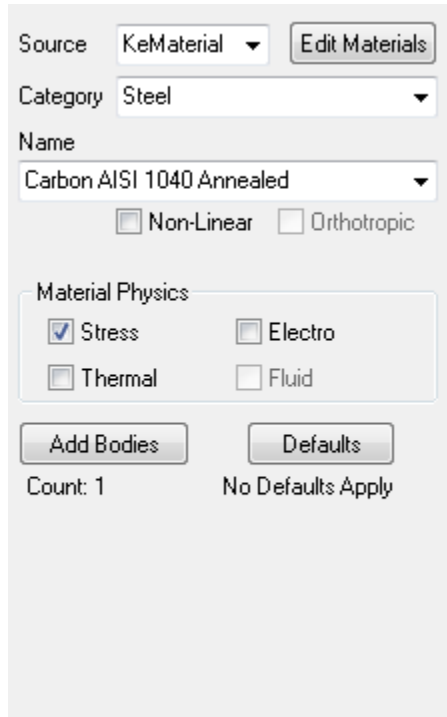
Mesh Solid as Shell

The **Mesh Solid as Shell** checkbox is currently the only way to do FEA shell analysis with IronCAD. It meshes the surface of the solid as FEA shell elements that enclose the solid.

D) The Material Page

The Material Page is displayed by clicking on any of the Material leafs located below the model page. After updating, the leaf name displays: (number of bodies) [E] physics-Material Name. Where physics is the first letter of the physics used for this material. The letter 'E' appears if the material has been edited and may not match the material from the AFEMaterial library.

The Material page allows the material properties, physics allowed for this material and the bodies associated with a material to be changed.



The screenshot shows a software interface for material properties. At the top, there is a 'Source' dropdown menu set to 'KeMaterial' and an 'Edit Materials' button. Below this is a 'Category' dropdown menu set to 'Steel'. The 'Name' section shows a dropdown menu with 'Carbon AISI 1040 Annealed' selected. Underneath the name, there are two checkboxes: 'Non-Linear' and 'Orthotropic', both of which are currently unchecked. A section titled 'Material Physics' contains four checkboxes: 'Stress' (checked), 'Electro' (unchecked), 'Thermal' (unchecked), and 'Fluid' (unchecked). At the bottom, there are two buttons: 'Add Bodies' and 'Defaults'. Below the 'Add Bodies' button, it says 'Count: 1'. Below the 'Defaults' button, it says 'No Defaults Apply'.

Source

Currently the source can be either the AFEMaterial or Edited. If the Edit Materials button has been used and the Material Edit dialog was not canceled, then the material properties are assumed to not match the AFEMaterial library and 'Edited' shows up as the source.

If you change the Source from 'Edited' to 'AFEMaterial', you are warned that the edited material will be discarded. Doing this allows you to select another material which is read from the AFEMaterial.csv file.

Material Physics

You can control what physics is used by each material in the FEA simulation.

Add Bodies

Click on the **Add Bodies** button and then click on bodies in the model to add them to this material. Then click the **Accept** (formerly Add Bodies) button to accept the changes. Clicking on a body a second time (before clicking Accept) will un-select the body so that it will not be added when Accept is clicked.

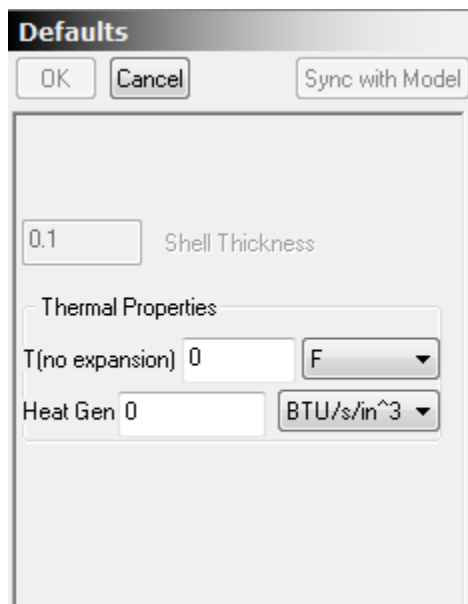
After clicking **Accept**, all the bodies associated with this material are selected. If you click the Cancel button (next to Apply under Material), you can cancel all material changes since the last **Apply**. This includes changes made with the **Add Bodies** button.

Defaults

Depending on the type of analysis (solid or shell) and the physics used, default values can be set for all bodies associated with this material using the **Defaults** button. You can override these defaults for individual bodies on the body page.

D1) The Material Defaults page

Click on **Defaults** button on the Material page to display this page. For Shell models, the default thickness can be set. For thermal models the temperature at which no expansion occurs can be set for this material: T(no expansion) and the Heat density (Heat Energy/volume) generated by bodies of this material can be set.



The screenshot shows the 'Defaults' dialog box. At the top, there are three buttons: 'OK', 'Cancel', and 'Sync with Model'. Below these buttons, there is a text input field containing '0.1' followed by the label 'Shell Thickness'. Underneath this, there is a section titled 'Thermal Properties'. Inside this section, there are two rows of controls. The first row has a text input field containing '0' followed by a dropdown menu showing 'F'. The second row has a text input field containing '0' followed by a dropdown menu showing 'BTU/s/in^3'.

D2) Material Edit (Linear)

Click on the **Edit Material** button on the Material Page with **Non-Linear** NOT checked to display this page. This allows constant material properties to be set for all physics models. If the names and units are grayed out, then the physics on the Material Page does not include those quantities. Also the enabled physics for this material are displayed. The enabled physics is set on the Material Page.

If OK is used instead of Cancel to exit this dialog, then the material is assumed to be edited and may not match the library values.

The image shows a 'Material Edit' dialog box with the following fields and values:

Property	Value	Unit
Name	Steel Demo	
Stress	<input checked="" type="checkbox"/>	
Thermal	<input checked="" type="checkbox"/>	
Electro	<input type="checkbox"/>	
Fluid	<input type="checkbox"/>	
Unit System	USA(ips)	
Youngs Modulus:E	2.90075e+007	psi
Poissons' Ratio:Vab	0.32	N/A
Mass Density:rho	0.283057	lb/in^3
Mass Damping Coef:	0	N/A
Stiffness Damping Coef:	0	N/A
Thermal Conductivity:	2.49894	BTU/(in.h.F)
Specific Heat Capacity:	0.000116079	BTU/(lb.F)
Linear Expansion Coef:	3.66667e-006	1/F
Electrical Conductivity:	151384	siemens/in
Fluid Viscosity:	0	/s

Name

This name is displayed in the FEA tree on the leaf associated with this material. Initially, the name matches the material selected from the AFEMaterial library.

Units System

The units can be displayed and edited in either USA (inches, pounds, seconds), SI (meters, kilogram, seconds) or SI (centimeter, grams, seconds). Internally, units are always stored in SI(mks) units.

Stress Material Properties

Basic properties are **Young's Modulus**, **Poisson's Ratio**, **Mass Density**. Mass Density is used only if inertia effects are included (Dynamic analysis type or an Acceleration or Rotational load is used).

Thermal Material Properties

Thermal Conductivity, **Specific Heat Capacity** and **Linear Expansion Coef** and Mass Density are used if Thermal is enabled.

Electrical Material Properties

Electrical Conductivity is used if Electro physics is enabled. If Electro and Thermo are both active, **JOULE HEATING** occurs where the current density is non-zero.

Stiffness and Mass Damping Coefficient (default 0)

These options are for advanced FEA users.

This is the damping coefficient applied to the mass matrix to for computing of the damping matrix. This hysteric damping coefficient can be found using the Rayleigh damping matrix formula from the damping factor. If the Rayleigh damp matrix C is

$$C = a_0 M + a_1 K$$

Where M is the FE mass matrix and K is the stiffness matrix, then the mass damping coefficient a_0 can be computed from

$$\xi = 0.5 (a_0/\omega + a_1\omega)$$

Where ξ is the damping factor from the test, and ω is the frequency of the vibration test. For more information, refer to "Structural Dynamics, An Introduction to Computer Methods", R. R. Craig, Jr., or other popular references.

D3) Material Edit (Non-Linear)

Click on the **Edit Material** button on the Material Page with **Non-Linear** checked to display this page.

This dialog supports various non-linear stress models including Plastic and Rubber. Future expansion will use the **Non-Linear Curves** buttons to support temperature (and other) dependent of the associated material quantities.

Non-Linear Isotropic Material Edit

Name

Steel Demo

OK

Cancel

☒ Stress
☒ Thermal
☐ Electro
☐ Fluid

Stress Material Models

☒ Non-Linear Curves
☐ Von Mises Plastic

☐ with Iso Hardening
☐ with Kinematic

☐ Mohr Coulomb Plasticity
☐ Hyperelastic Rubber
☐ Hyperelastic Foam

Quantity	Value	Non-Linear Multipliers	Unit System
Youngs Modulus:E	2.90075e+007	Temp+	psi
Poissons' Ratio:Vab	0.32	Sig	
Mass Density:rho	0.283057	Time	lb/in^3
Mass Damping Coef.	0	*----	
Stiffness Damping Coef.	0	-----	
Plastic Yield Stress	99989	-	psi
Isotropic Hardening	2.90075e+006	-	psi
Kinematic Factor (0-1)	0	-	
Thermal Conductivity	2.49894	*Temp	BTU/(in.h.F)
Specific Heat Capacity	0.000116079	-	BTU/(lb.F)
Linear Expansion Coef.	3.66667e-006	-	1/F
Electrical Conductivity	151384	-	siemens/in
Fluid Viscosity	0	-	/s

Name

This name is displayed in the FEA tree on the leaf associated with this material. Initially, the name matches the material selected from the AFEMaterial library.

Non-Linear Multipliers

These buttons allow the material value to be multiplied by a factor determined by the current value of one of the output variables. The factors can also depend on time (if not Frequency Analysis) or frequency (if Frequency Analysis).

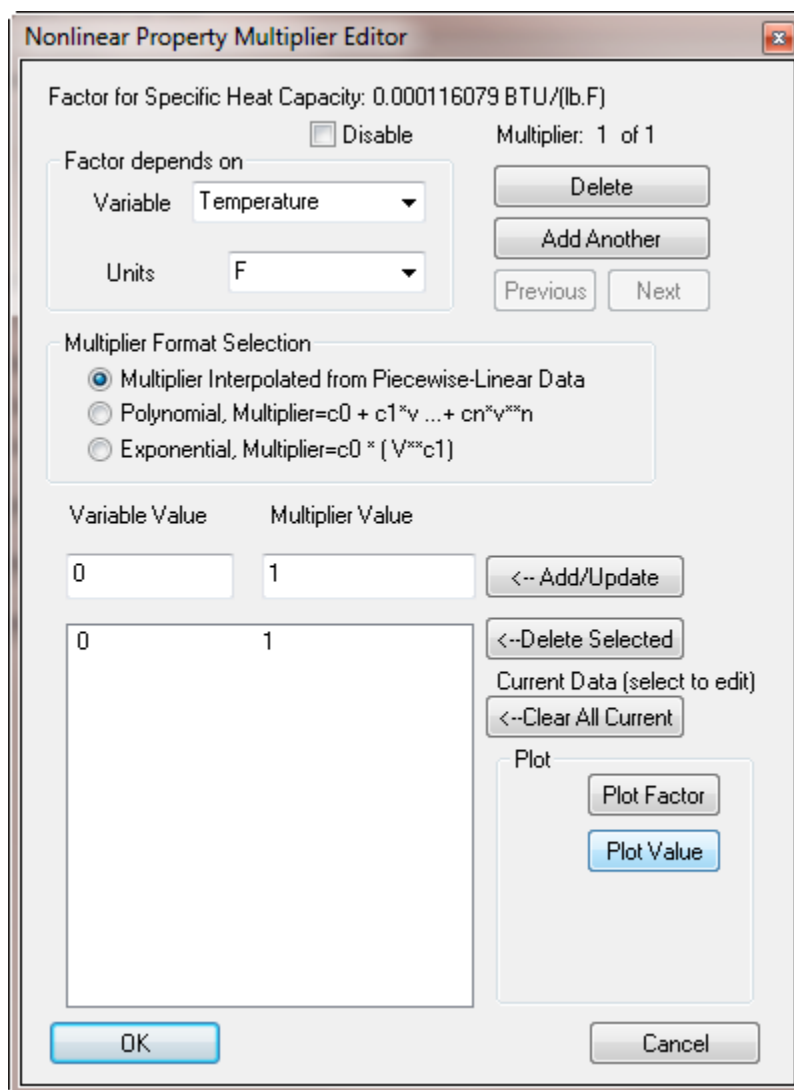
Non-Linear Button Notation

A minus sign, '-', indicates no non-linear multipliers are applied.

A name on the button (e.g. Temp, Sig, Time) indicates the output variable that the value depends on. If a '+' follows the name, then additional multipliers have been added. If a '*' is used before the name, then one of the additional multipliers is specified but unused because the current physics does not generate the necessary output variable. If a '-' appears before the name, then the user has disabled some of the property multipliers.

If '*----' is displayed, then no multipliers are applied because the physics does not generate the necessary output variable. Similarly, '- ----' indicates that no multipliers are applied because all the multipliers have been disabled by the user.

If the user clicks on one of the Non-Linear Multiplier buttons, the Nonlinear Property Multiplier editor is displayed:



The dialog box is titled "Nonlinear Property Multiplier Editor". It contains the following elements:

- Factor for Specific Heat Capacity: 0.000116079 BTU/(lb.F)
- ☐ Disable
- Multiplier: 1 of 1
- Factor depends on:
 - Variable: Temperature (dropdown)
 - Units: F (dropdown)
- Buttons: Delete, Add Another, Previous, Next
- Multiplier Format Selection:
 - ☒ Multiplier Interpolated from Piecewise-Linear Data
 - ☐ Polynomial, Multiplier= $c_0 + c_1*v + \dots + c_n*v^n$
 - ☐ Exponential, Multiplier= $c_0 * (V^{c_1})$
- Table with 2 columns: Variable Value, Multiplier Value.
 - Row 1: 0, 1
 - Row 2: 0, 1
- Buttons: <-- Add/Update, <--Delete Selected, Current Data (select to edit), <--Clear All Current
- Plot section with buttons: Plot Factor, Plot Value
- Buttons: OK, Cancel

Factor Depends on

This shows the Name, Value and Units of the property being edited. The output variable the factor depends on can be selected from the pull-down. If a ‘*u*’ precedes the variable name, then the current physics (or analysis type) do not allow that variable to be used. For example, Voltage and Frequency are not used in the following pull-down.

Temperature
Sig Equiv
Time
u Voltage
Eps Equiv
u Frequency

The ability to have unused multipliers defined makes it easier to ‘turn off thermal’ and run an analysis without thermal to see the effect without having to disable all multipliers dependent on temperature.

Multiplier Format Section

The default is **Piecewise-Linear** which means that the pairs of variables in the large list box specify the output value and multiplier associated with that value. If the output variable is less than the first Variable Value, then the first Multiplier Value is used. Similarly if the output variable is greater than the last Variable Value, then the last Multiplier Value is used. The *Plot Factor* and *Plot Value* buttons can be used to view either the Factor or the property value based on this multiplier.

If **Polynomial, Multiplier** is used, then the left variable is used for List Position (and not the polynomial coefficient index). So in the example below, the polynomial would be $5 - 3*v + 1.5 * v^2$:

Multiplier Format Selection

☐ Multiplier Interpolated from Piecewise-Linear Data
☒ Polynomial, Multiplier= $c_0 + c_1*v + \dots + c_n*v^n$
☐ Exponential, Multiplier= $c_0 * (V^{c_1})$

List Position: 5 Poly Coef: c0,c1,c2...: 1.5 <-- Add/Update

0	5
2	-3
5	1.5

<--Delete Selected

Current Data (select to edit)

<--Clear All Current

Plot

Plot Factor

Plot Value

Vmin: 0

Vmax: 1

OK Cancel

Exponential, Multiplier works the same way as polynomials in that the left variable is only used for list position. The first variable on the right (5 in the above box) is the factor and the second variable on the right (-3 in the above box) is used as the exponential. Any additional variables are ignored.

Changing the values in the large list box

Values in the large list box are changed by entering values in the two entry boxes on the Add/Update line. Clicking the **←Add/Update button** adds the values to the large list box. The new value is inserted in order based on the left variable. If an entry already exists for that value, that entry is replaced (e.g. the entry is updated).

Clicking on any line in the large list box brings the values to the Add/Update entry line. This provides a quick way to change one of the right values. Just click on it, change the right value on the Add/Update line and click the **← Add/Update button**.

To delete the selected item in the large list box, click **← Delete Selected**. To delete all items, click **←Clear All Current**.

Controlling the number of Multipliers for this property

The **Delete**, **Add Another**, **Previous** and **Next** buttons on the upper right control the number of Multipliers for this property. **Delete** will delete the current Multiplier and close the dialog. Add Another will allow you to add another multiplier for this property. This allows support for properties that depend on more than one output property. After the first Add Another, the line above delete changes to **Multiplier 2 of 2**. If more than one Multiplier is defined, the **Previous** and **Next** buttons allow you to change between multipliers.

Disable

The **Disable** checkbox can be used to disable a multiplier without discarding the data for the multiplier. Disabling a Multiplier and running the analysis again allows you to see the effect of the non-linear dependency on the simulation.

Non-Linear Stress Models

1) Non-Linear Curves (future version)

This option allows just the standard material data to be used (as with the Linear Material Edit) except that the material quantities can be temperature dependent.

2) Von Mises Plastic

This option is used for plastic materials. The Plastic Yield Stress is used. If **with Iso Hardening** is checked, then the Isotropic Hardening Modulus is used. If **with Kinematic** is checked, then the Kinematic Factor is used.

Isotropic Hardening Modulus

Specify the isotropic hardening modulus. This modulus can be computed from the uniaxial test result E and E_t . The tangent modulus E_t is the apparent slope of the stress-strain curve after the initial yield, and E is the initial modulus (Young's modulus). The hardening modulus H is defined as

$$H = E_t / (1 - E_t / E)$$

Effectively, the yield stress is then computed from

$$\sigma_y = \sigma_{y0} + H \epsilon^p$$

where σ_{y0} is the initial yield stress, and ϵ^p is the accumulated effective plastic strain computed.

Kinematic Hardening Factor

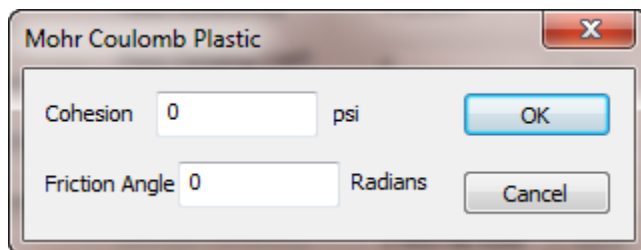
This factor β controls the kinematic Bauschinger hardening effect in the cyclic loading behavior. The effective yield stress is computed from

$$\sigma_y = \sigma_{y0} + (1 - \beta) H \epsilon^p$$

where σ_{y0} is the initial yield stress, ϵ^p is the accumulated effective plastic strain computed, and H is the isotropic hardening modulus. When $\beta=0$, the kinematic hardening effect is excluded, and when $\beta=1$, the full kinematic effect is applied.

3) Mohr Coulomb Plasticity

If this option is selected, the Mohr Coulomb Plastic model is used. Clicking on the Mohr Coulomb button brings up the following dialog to specify the Cohesion and Friction Angle.



The friction angle should be between 0 and 90 degrees.

4) Hyperelastic Rubber (incompressible Mooney-Rivlin)

This option specifies the hyperelastic Mooney Rivlin incompressible material model. Click on the **Hyperelastic Rubber** button to bring up the Hyperelastic Rubber dialog to set the C_{xx} values.

The dialog box is titled "Hyperelastic Rubber". It displays the following strain energy function:

$$W = C_{10}*(J_1 - 3) + C_{01}*(J_2 - 3) + (C_{20}*(J_1 - 3)^2 + C_{11}*(J_1 - 3)*(J_2 - 3) + C_{02}*(J_2 - 3)^2) + (C_{30}*(J_1 - 3)^3 + C_{21}*(J_1 - 3)^2*(J_2 - 3) + C_{12}*(J_1 - 3)*(J_2 - 3)^2 + C_{03}*(J_2 - 3)^3) + K/2 (J_3 - 1)^2$$

Below the equation, there are input fields for coefficients:

- Main Coefficients:**
 - C10 = 0
 - C01 = 0
- Additional for MR5:**
 - C20 = 0
 - C11 = 0
 - C02 = 0
- Additional for MR9:**
 - C30 = 0
 - C21 = 0
 - C12 = 0
 - C03 = 0

At the bottom, there is a button "Fit Coefficients from CSV file" and a dropdown menu for "Stress units:" set to "psi". "OK" and "Cancel" buttons are in the top right.

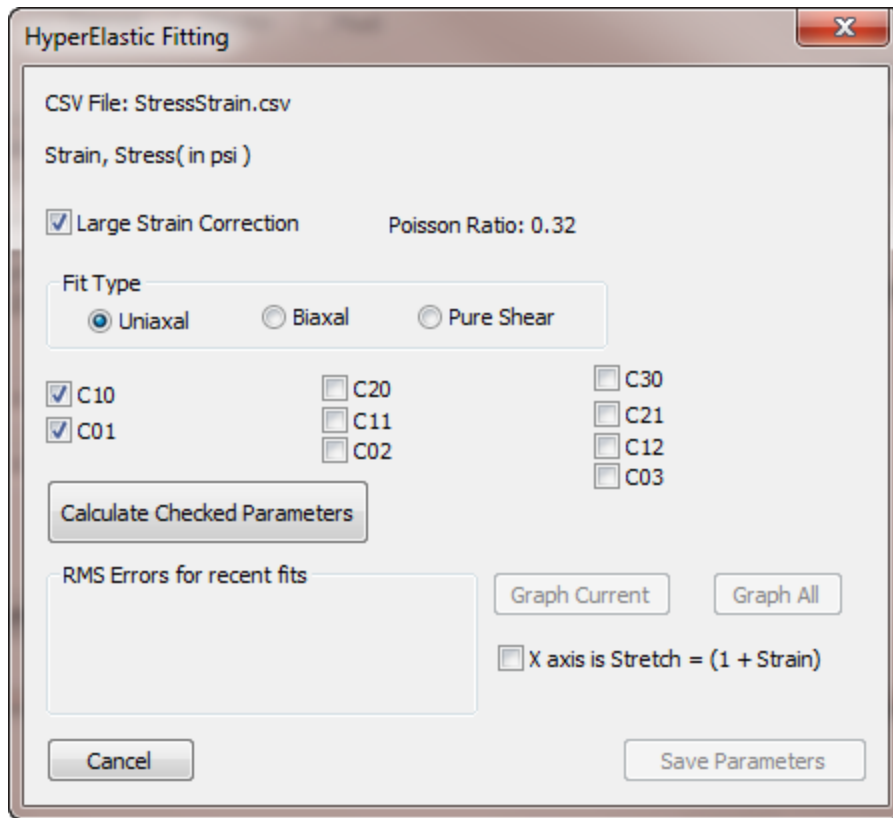
$$W = C_{10}*J_1 + C_{01}*J_2 + (C_{20}*J_1^2 + C_{11}*J_1*J_2 + C_{02}*J_2^2) + (C_{30}*J_1^3 + C_{21}*J_1^2*J_2 + C_{12}*J_1*J_2^2 + C_{03}*J_2^3) + K/2 (J_3 - 1)^2$$

where $J_1 = (I_1/I_3^{1/3}) - 3$, $J_2 = (I_2/I_3^{2/3}) - 3$, $J_3 = I_3^{1/2}$. I_1, I_2, I_3 are the Cauchy-Green strain tensor $C = [F]^T[F]$ from deformation tensor F . The volumetric effect is controlled by the bulk modulus K , and it is computed from ν , the Poisson's ratio.

More information about this material model can be found in "Finite Element in Nonlinear Continua", J. T. Oden, or other popular references.

The **Fit Coefficients from CSV** file button can be used to calculate the Mooney Rivlin Cxx parameters from a comma separated value file of strain, stress pairs. You should have create the comma separated file before using this command as the first step is to select the CSV strain, stress file from a file dialog.

Once the strain, stress CSV file is loaded, the following dialog appears:



To fit a comma separated value file of strain, stress data do the following:

1) Create a comma separated file of "strain, stress" pairs from experimental results for the hyperelastic material. You can use Notepad and rename the file from .txt to .csv or you can enter the data into the first two columns of Excel and save the data as a 'Comma Separated Value' file.

2) The **Large Strain Correction** should be applied to experimental data when the strain is larger than approximately 0.2.

3) The **Fit Type** should be specified.

4) The Poisson Ratio specified on the Stress page is displayed because it is used in the fitting calculation.

5) Check the coefficients to be non-zero in the fitting equation.

6) Click **Calculate Checked Parameters** to calculate fit the data with the checked coefficients.

The value of the coefficients is displayed by the coefficient and the root-mean-squared error of the approximation is displayed: $\text{RMS error} = \sqrt{\frac{\sum (\text{fit}(\text{strain}_i) - \text{data}(\text{strain}_i))^2}{\text{number of points}}}$.

7) Use **Graph Current** to graph the experimental results and the current fit approximation.

8) Try changing the checked parameters and clicking **Calculate Checked Parameters** again.

The RSM error for the current and last two fits are displayed. Also the **Graph All** button can be used to graph the current and last two fits. The RMS Error label and graph label indicates which options were used and which checked parameters were used. For example

U-LC 11 001 1000

Indicates that a Uniaxial fit (U) with the Large strain Correction (-UC) was used to calculate C10, C01, C02 and C30 (11 001 1000).

9) When you are satisfied with the fit, select Save Parameters and the current parameters are copied to the Hyperelastic Rubber dialog.

Hyperelastic Strain Energy Function (Technical Details)

In AMPS, the general hyperelastic strain energy function is expressed in the polynomial form

$$W = \sum_{i+j=1}^{i+j=n} c_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + \frac{K}{2} (J - 1)^2$$

Where \hat{I}_1 and \hat{I}_2 are the first and second deviatoric Cauchy-Green strain tensor $C = F^t F$ defined from the deformation gradient F , and J is the Jacobian of the C matrix. K is the bulk modulus controlled by Poisson's ratio. These deviatoric invariants can be shown to be

$$J = \det C = \lambda_1^2 \lambda_2^2 \lambda_3^2$$

$$\hat{I}_1 = J^{-1/3} (\lambda_1^2 + \lambda_2^2 + \lambda_3^2)$$

$$\hat{I}_2 = J^{-2/3} (\lambda_1^2 \lambda_2^2 + \lambda_2^2 \lambda_3^2 + \lambda_3^2 \lambda_1^2)$$

where λ_1, λ_2 and λ_3 are the principal stretch ratio.

Stress and Strain Correction for Large Deformation

When a uniaxial testing is conducted, the nominal/engineering stress and strain should be corrected to account for the Poisson's effect to compute the true stress and strain. If the engineering strain $e = (L - L_0)/L_0$ and stress $s = F/A_0$, with L, L_0, F, A_0 being the original test sample deformed length, original length, total force applied, and the original cross section area respectively, then the true stress and strain can be shown to be,

$$\varepsilon = \ln(1+e) = \ln(\lambda_1)$$

$$\sigma = s(1+e) = s \lambda_1$$

Stress calculations

The normal directional stress σ_i based on the hyperelastic function can be computed from the following formula

$$\sigma_i = \delta W / \delta \lambda_i = (\delta W / \delta \hat{I}_1) (\delta \hat{I}_1 / \delta \lambda_i) + (\delta W / \delta \hat{I}_2) (\delta \hat{I}_2 / \delta \lambda_i) + (\delta W / \delta J) (\delta J / \delta \lambda_i)$$

For uniaxial testing,

$$\lambda_1 = \lambda_U, \lambda_2 = \lambda_3 = \lambda_U^{-1/2}$$

$$\delta \hat{I}_1 / \delta \lambda_1 = 2 - 2\lambda_1^{-2} =$$

$$\frac{4(1+\mu)}{3} \left(\lambda_1^{\frac{1+4\mu}{3}} - \lambda_1^{\frac{-5-2\mu}{3}} \right)$$

$$\delta \hat{I}_2 / \delta \lambda_1 = 2 - 2\lambda_1^{-2} =$$

$$\frac{4(1+\mu)}{3} \left(\lambda_1^{\frac{-1+2\mu}{3}} - \lambda_1^{\frac{-7-4\mu}{3}} \right)$$

For equal biaxial testing,

$$\lambda_1 = \lambda_2 = \lambda_B, \lambda_3 = \lambda_B^{-2}$$

$$\delta \hat{I}_1 / \delta \lambda_1 = 4\lambda_1 - 4\lambda_1^{-5}$$

$$\delta \hat{I}_2 / \delta \lambda_1 = 4\lambda_1^3 - 4\lambda_1^{-3}$$

And for pure shear testing,

$$\lambda_1 = \lambda_S, \lambda_2 = 1, \lambda_3 = \lambda_S^{-1}$$

$$\delta \hat{I}_1 / \delta \lambda_1 = 2\lambda_1 - 2\lambda_1^{-3}$$

$$\delta \hat{I}_2 / \delta \lambda_1 = 2\lambda_1^3 - 2\lambda_1^{-3}$$

Curve Fitting Algorithm

The least-squares error function between the Cauchy stress σ derived from the constitutive model and the derived Cauchy stress σ^{test} from the experimental test is

$$E = \sum (1 - \sigma / \sigma^{\text{test}})^2$$

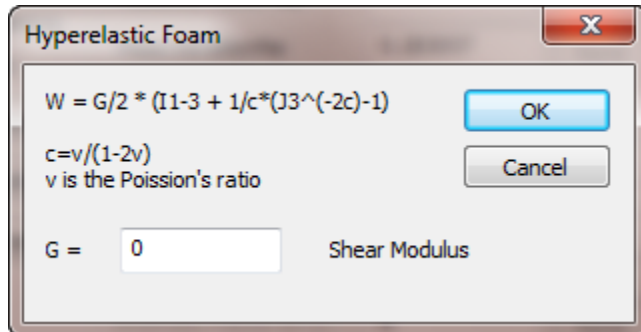
By minimizing the error function, we can find the C_{ij} coefficients from the equations

$$\delta E / \delta C_{ij} = 0$$

5) Hyperelastic Foam

$W = G/2 * (I1-3 + 1/c*(J3^(-2c)-1))$, where G is the shear modulus, and $c = v/(1-2v)$, and v is the Poisson's ratio. For Foam, the 'G' parameter is entered in the Hyperelastic Foam dialog and the Poissons' Ratio/Vab from this dialog is used to calculate c .

Clicking on **Hyperelastic** Foam brings up the following dialog so the G value can be entered.

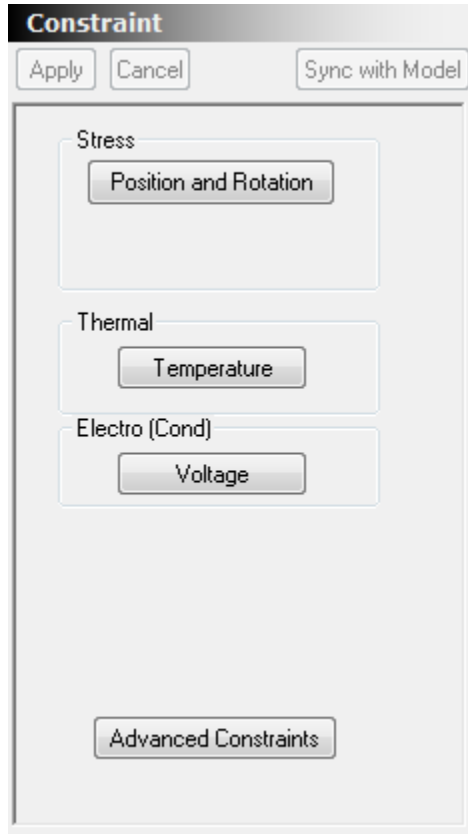


The dialog box is titled "Hyperelastic Foam" and contains the following text and controls:

- Equation: $W = G/2 * (I1-3 + 1/c*(J3^(-2c)-1))$
- Equation: $c=v/(1-2v)$
- Text: v is the Poisson's ratio
- Buttons: "OK" and "Cancel"
- Input field: "G =" followed by a text box containing "0"
- Label: "Shear Modulus"

E) Constraints

Clicking on the Constraint leaf brings up the Constraint page. Depending on the physics allowed on the Analysis page, different buttons will be available. The following constraint page corresponds to Stress, Thermal and Electrical physics:



E1) Position and Rotation Constraint

Clicking the **Position and Rotation** button on the Constraint page, adds a Position and Rotation Constraint. In shell analysis, the Fixed Position page includes options for fixing the rotation of a node.

Fix Position

Apply Cancel Sync with Model

Displacements

Units **inch**

Dir	Use	Displacement
X	<input checked="" type="checkbox"/>	0
Y	<input checked="" type="checkbox"/>	0
Z	<input checked="" type="checkbox"/>	0

XYZ

Update Entities

Units:

The pull-down specifies the units used for the displacement.

Coordinate system (XYZ pull-down)

The XYZ pull-down can be used to change the coordinate system associated with this constraint.

Whenever you do not use the XYZ option, local coordinates are set up for the nodes associated with this boundary condition. There are several cautions when using these local coordinates.

Caution 1: Each FEA node can only have one set of local coordinates and local coordinates the same set of local coordinates can be used by other loads and boundary conditions that don't use the global XYZ directions. If multiple local coordinates are applied to a node, the last set of local coordinates (e.g. assigned by the last BC that uses that node) are used.

Caution 2: Displacements are always in distance units and always linear. For example, if Cylindrical coordinates are used, a displacement of .5 (inch) in the Th (Theta) direction means a displacement of .5 inches along the local theta direction defined at the node and not a rotation of .5 radians around the axis.

If a coordinate system other than XYZ (global X, Y and Z) is used, then a Set Coordinates button appears to allow the user to set the coordinates for the coordinate system. The following coordinate systems are available

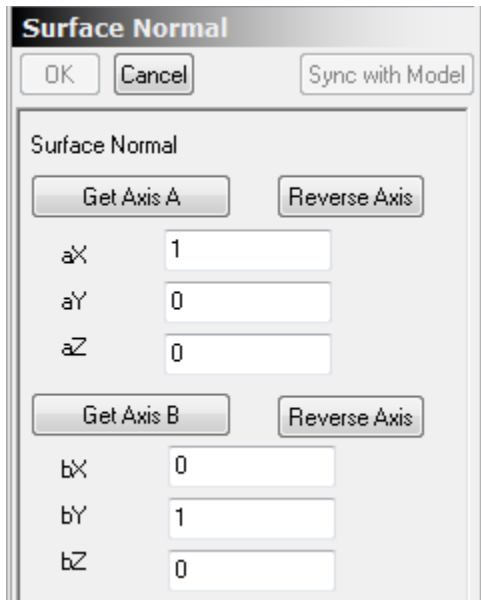
1) XYZ

Standard global X, Y and Z directions.

2) Normal

A coordinate system normal to a surface. The directions are N (normal), ab (A direction projected onto the surface. If A is parallel to the normal, then the B direction projected to the surface is used), c (a direction in the surface obtained by rotating the ab direction 90 degrees about the normal direction).

Selecting **Set Coordinates** brings up the following page:



Surface Normal

OK Cancel Sync with Model

Surface Normal

Get Axis A Reverse Axis

aX 1

aY 0

aZ 0

Get Axis B Reverse Axis

bX 0

bY 1

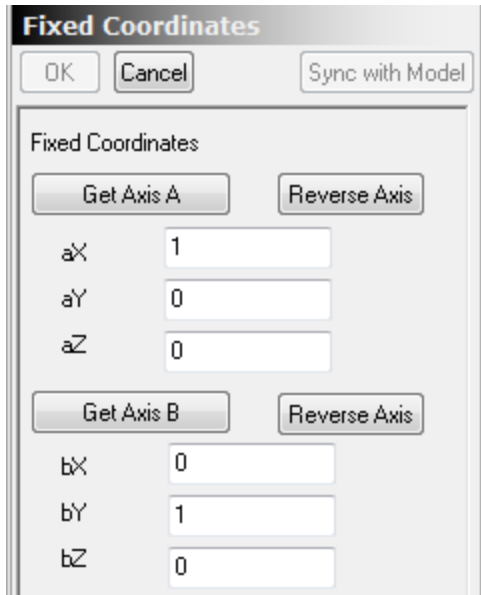
bZ 0

The A and B directions can be specified in global coordinates or the **Get Axis** buttons can be used to get the directions from a straight edge in the model.

3) Fixed

A coordinate system fixed in space but not aligned with the global X, Y and Z axes. This coordinate system is defined by two non-parallel vectors A and B. The a direction is along the A vector. The b direction is the component of B that is perpendicular to a. The c direction is perpendicular to both a and b. The directions a, b and c form a right hand coordinate system.

The Set Coordinates button brings up the following page:



The image shows a software dialog box titled "Fixed Coordinates". At the top, there are three buttons: "OK", "Cancel", and "Sync with Model". Below these, the title "Fixed Coordinates" is repeated. The dialog is divided into two main sections. The first section, labeled "Get Axis A", contains three input fields: "aX" with the value "1", "aY" with the value "0", and "aZ" with the value "0". To the right of these fields is a "Reverse Axis" button. The second section, labeled "Get Axis B", contains three input fields: "bX" with the value "0", "bY" with the value "1", and "bZ" with the value "0". To the right of these fields is another "Reverse Axis" button.

The A and B directions can be specified in global coordinates or the **Get Axis** buttons can be used to get the directions from a straight edge in the model.

Fixed coordinates usage

Fixed coordinates are used when you need to apply constraints in a coordinate system rotated from the standard X, Y and Z directions. For example, to constrain a node to move in the plane defined by $X = Y$, you could set the A vector to (1,1,0) and the B vector to (1, -1, 0). Then in the Fix Position Page, set a check for the b direction only. This means that the node can move freely in the a (1,1,0) direction and the c (0,0,1) direction but is constrained in the b (1,-1,0) direction. In this example, the B vector could be any vector in the XY plane that is not parallel to the A vector and it would be normalized to (1, -1, 0). If B represented more than a 180 degree rotation of A in the XY plane, the c vector would point in the -Z direction, but the constraint would still work.

4) Cylindrical

A cylindrical coordinate system defined by an origin and axis direction. The R (radial) direction is away from the axis. The Th (theta) direction is perpendicular to the R and axis direction. The H (height) direction is along the axis (e.g. the height of the cylinder direction).

Selecting **Set Coordinates** allows the origin and axis direction to be specified:

Cylindrical Coordinates

OK Cancel Sync with Model

Cylindrical Coordinates

Get Origin

cX = 0

cY = 0

cZ = 0

Get Axis Direction Reverse Axis

aX = 0

aY = 0

aZ = 1

Get Origin and Axis from Arc

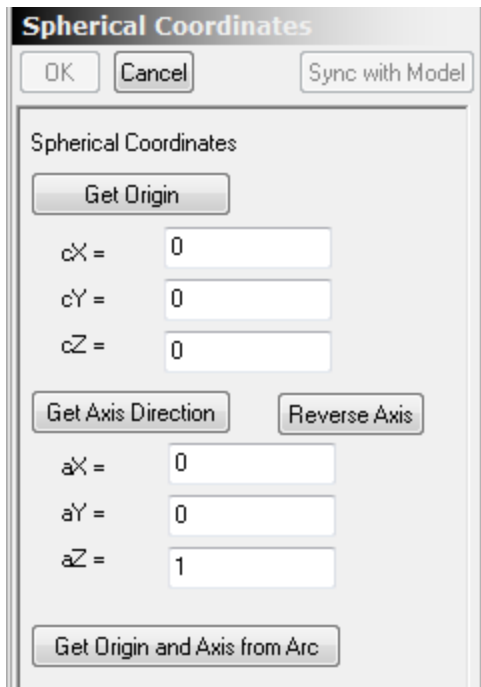
Selecting the **Get Origin and Axis from Arc** button allows you to set both the Origin and Axis direction by clicking on an edge in the solid model that is a circular arc.

Note that a displacement in the Th (Theta) direction is still specified in distance units and not radians. It represents moving the node the specified distance in the local Th (theta) direction.

5) Spherical

A spherical coordinate system defined by an origin and axis direction. The R (radial) direction is away from the origin. The Th (theta) direction is perpendicular to the R and axis direction. The Az (azimuthal) direction is perpendicular to both the R and Th directions. The R, Th and Az directions form a local right handed coordinate system.

Selecting **Set Coordinates** allows the origin and axis direction to be specified:



Selecting the **Get Origin and Axis from Arc** button allows you to set both the Origin and Axis direction by clicking on an edge in the solid model that is a circular arc.

Fixed Position and Shell models

Shell (and beam) models and solid models treat node rotation differently. For solid models, the nodes do not include orientation (rotation) information. To rotate a corner of a solid model, you must move adjacent nodes to represent the rotation. To apply a torque to a solid model, rigid elements must be used to simulate the torque being applied to multiple nodes or an actual solid must be used to simulate the shaft providing the torque.

Shell models approximate sheets with a fixed thickness. Rotating a node corresponds to bending the fixed width sheet around the node. Similarly, torques can be applied directly to nodes in shell (and beam) models.

To allow for this extra rotational degree of freedom in shell models, the Fix Position page includes an option to prescribe a fixed rotation at each node. For shell models, the following Fix Position dialog is used:

Fix Position

Apply Cancel Sync with Model

Displacements Rotation

Units mm Degrees

Dir	Use Displacement	Use Rotation
X	<input checked="" type="checkbox"/> 0	<input type="checkbox"/> 0
Y	<input checked="" type="checkbox"/> 0	<input type="checkbox"/> 0
Z	<input checked="" type="checkbox"/> 0	<input type="checkbox"/> 0

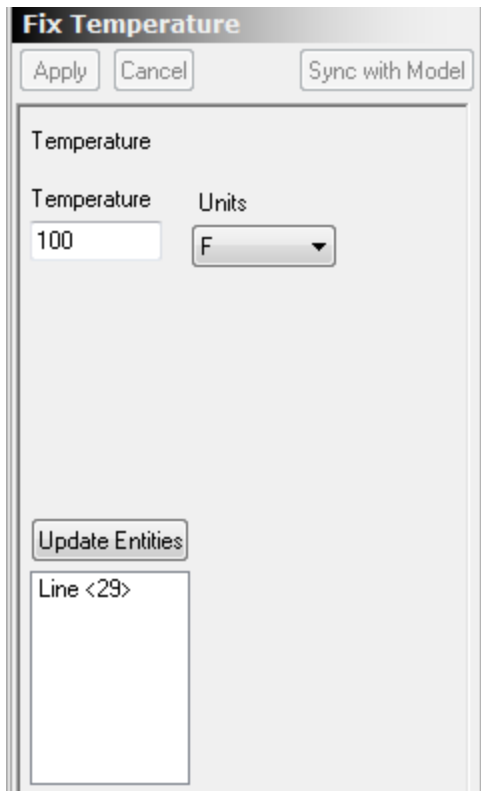
XYZ

Update Entities

The Use checkboxes next to the rotation can be used to fix the rotation of the node in the specified coordinate directions (X, Y and Z in this case). The displacement and Rotation coordinate systems are always the same. The **Units pull-down** for Rotation (Degrees pull-down) can be used to specify the rotation in either Degrees or Radians.

E2) Temperature Constraint

Clicking the **Temperature** button on the Constraint page, adds a Temperature Constraint leaf. This option is only available if Thermal physics is allowed on the Analysis page.

The image shows a software dialog box titled "Fix Temperature". At the top, there are three buttons: "Apply", "Cancel", and "Sync with Model". Below these buttons, the word "Temperature" is displayed. Underneath, there are two labels: "Temperature" and "Units". The "Temperature" label is followed by a text input field containing the number "100". The "Units" label is followed by a pull-down menu currently showing "F". At the bottom left of the dialog, there is a button labeled "Update Entities". Below this button is a text area containing the text "Line <29>".

Temperature Units

The Units pull-down allows the fixed temperature to be specified in the following units.

F) Fahrenheit

C) Celsius

K) Kelvin

R) Rankine

These temperature units are related by the following formulas:

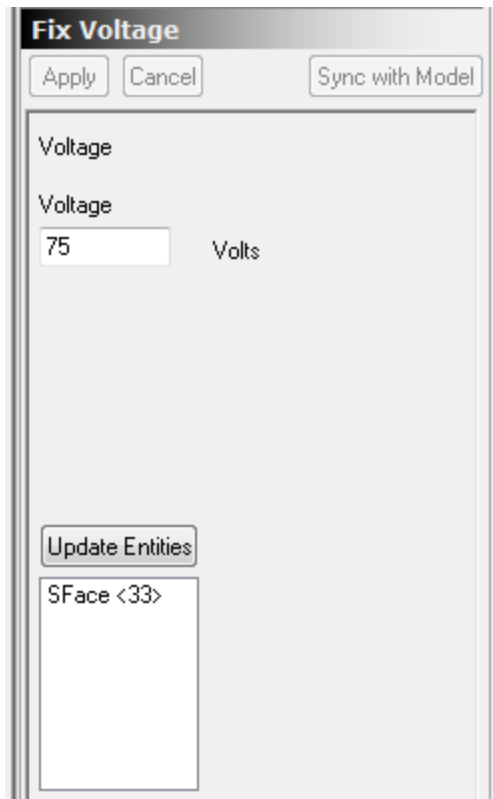
$$F = C * (9/5) + 32$$

$$K = C + 273.15$$

$$R = F + 459.67$$

E3) Voltage Constraint

Clicking the **Voltage** button on the Constraint page, adds a Voltage Constraint leaf. This option is only available if Electro physics is allowed on the Analysis page.



E4) Advanced Constraints

Clicking on the Advanced Constraints brings up the Advanced Constraints page to allow you to add advanced constraints to your model.

Currently **Surface Contact**, **Tie/Glue**, **Rigid Rotation** and **Nodal Stiffness**, **Damping** and **Mass** constraints are available:



E5) Advanced Constraint: Surface Contact

Selecting Surface Contact from the Advanced Constraints page adds a Contact constraint to the model and brings up the following Contact page:

Contact

Apply Cancel Sync Sim/Part

Friction

Static Coef. 1

Heating Coef. 0

Advanced Settings Phy=S

☒ All Except Selected
☐ Only Selected

Update Faces Geometric Planes

0 Defined

Because of the advanced contact capabilities of our FEA processor, the default settings on the contact page are sufficient for most contact situations. The **All Except Selected** indicates that all surfaces except the selected surfaces are to be checked for contact.

If you know that only a few surfaces will come in contact, you can slightly speed up the analysis by setting the Only Selected option and adding the bodies or faces of the objects that will come in contact.

Multi-step or Dynamic?

Contact analysis requires multi-step or dynamic analysis. For some models, the stabilizing effect of inertia included in dynamic models can allow the processor to converge to a solution even if convergence does not occur in multi-step analysis.

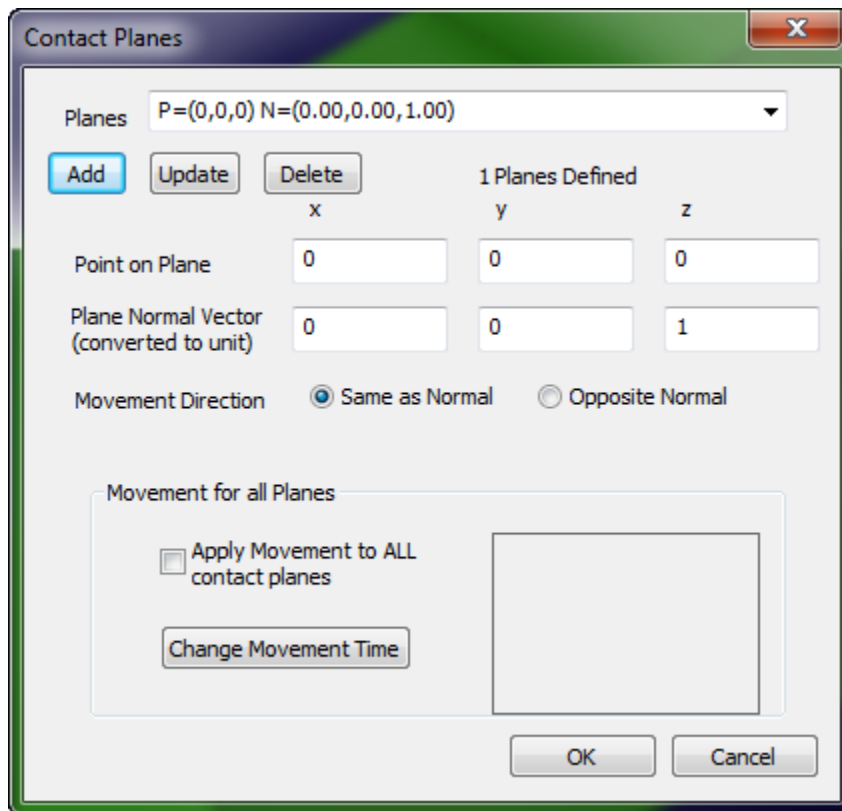
Friction Values

If Solid Continuum Contact (Stress) is selected on the Advanced page (default is on), then the **Static Coef.** is used. If the lateral force on a body is greater than the normal force times the static coefficient of friction, then the bodies can start slipping past each other.

If Thermal Contact and Solid Continuum Contact are selected on the Advanced page, then the **Heating Coef.** value is used to convert frictional energy to heat energy.

Geometric Planes

Along with faces of bodies, geometric planes can be used to provide either a stationary boundary or moving boundary for the contact surfaces. The **Geometric Planes** button brings up the following dialog:



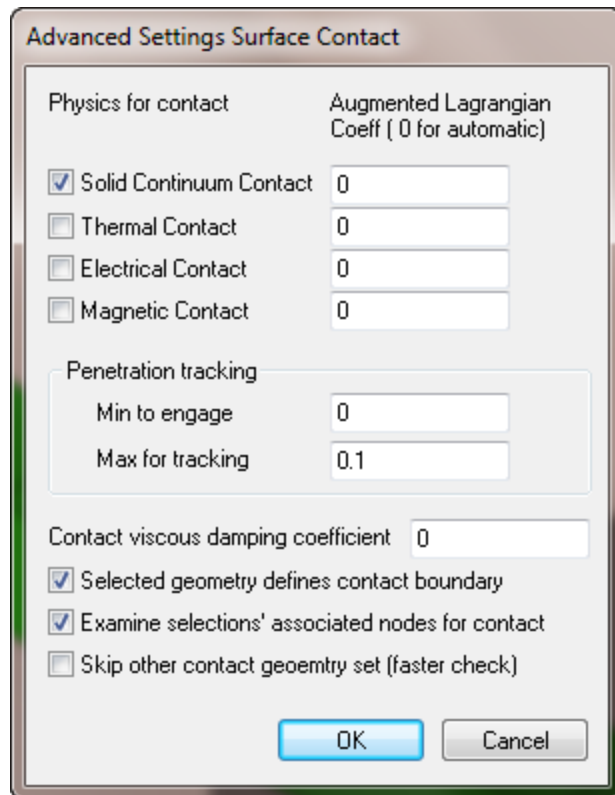
Add is used to add a plane to those used for this contact condition. Once it is added, the point and normal direction are displayed in the top pull-down. Once planes are defined, selecting the plane from the top pull-down updates the Point and Normal value in the dialog. A geometric representation of the current plane is shown in the model window. You can modify these values and click **Update** to update the values for the plane. The **Delete** button can be used to delete the plane currently displayed in the pull-down.

If **Movement for all Planes** is checked, then the movement time factor multiplied by a UNIT normal (in the length units specified on the Model page) current model units to specify the movement of the plane as a function of time. If **Opposite Normal** is checked, then the movement is in the opposite direction of the specified Plane Normal Vector.

All Planes associated with one Surface Contact condition move at the same rate. Multiple Surface Contact conditions can be used if multiple planes moving at different rates are needed.

Advanced Settings

The **Advanced Settings** button on the Contact BC page, brings up the Advanced Settings Surface Contact dialog:



This dialog allows the physics associated with the contact to be enabled and the other parameters related to the implementation of the contact BC by the processor.

Solid Continuum Contact

The solid contact detection is activated when this option is checked. The friction coefficient is used to compute the friction pressure from the normal pressure between the contacting surfaces. ALM (Augmented Lagrangian Multiplier) method is used to enforce both contact and friction behavior. The numerical method is preferred over the traditional Lagrangian Multiplier method as it preserves the SPD (symmetric Positive Definite) nature of the FE equations for faster iterative solution speed. In ALM, the multiplier value is automatically chosen if the ALM coefficient is left to be zero. In such case, the multiplier value is automatically computed based on the solid's stiffness at the point of the contact. Pending on the solution converge/diverge status, the augmented procedure tighten up the multiplier to enforce the error tolerance as requested. When solution converges, the multiplier value is effectively the needed normal pressure to enforce the non-penetration contact condition. If it becomes necessary to specify a more suitable multiplier value, the user can enter a non-zero scaling coefficient to increase/decrease the computed value. For example, enter a value of 0.1 will use only 1/10 of the computed multiplier value. Usually, if a value is too low, some slight penetration could be observed. A higher value usually will have smaller penetration, but at the cost of more numerical iterations, and leads to possible instability if the value is too high. Since the ALM implementation is very precise as it is computed from the actual physical condition during solution, it is recommended to always use the default augmented coefficient.

Thermal/Electrical Contact

When checked, thermal/electrical contact checking is activated. The thermal and/or electrical continuity is enforced through the contacting face if solid deformation contact is detected. When thermal physics is activated, since the contact friction will generate thermal energy and becomes a heat source, a frictional energy to thermal energy conversion constant can be supplied to take into this consideration. Since the contact formulation is based on the

ALM method, the multiplier value is automatically chosen if the ALM Coefficient is the default zero value. In such case, the multiplier value is automatically computed from the actual thermal/electrical property of the contacting solids during contact solution. If it becomes necessary to specify an alternative ALM value, the user can enter a scaling value to increase/decrease the computed ALM value.

Magnetic Contact

Not currently used but reserved for future expansion.

Penetration tracking: Min to engage

To avoid repeated contact open/close situation, when the contact penetration is more than the specified ratio, the contact mechanism will engage. The ratio refers to the penetration distance into the element bounding box size. The default value is 0.01%.

Penetration tracking: Max for tracking

When the penetration is more than the specified ratio, the contact tracking will not be engaged to and the contact condition will no longer be active. Not engaging the contact improves efficiency. In such case, usually a bad mesh or bad element aspect ratio is the culprit and a more refined mesh is suggested. The ratio refers to the penetration distance into the element bounding box size. The default value is ratio is 0.1.

Contact viscous damping coefficient

Viscous contact damping has been implemented to damp out oscillations during contact that sometime produces high frequency noise. The appropriate value is usually in the range of $1e-6$ to $1e-3$, depending on the desired behavior. If the value is too small, the contact region oscillate between being opened and closed and makes it difficult to generate a solution. If the value is too large, unnecessary energy loss occurs at the contact. The viscous damping is only applied to the local region engaged in contact so the energy loss is localized.

The coefficient value is the percentage of the body stiffness (stiffness matrix) that will be applied to the contact zone to stabilize the contact behavior. The *contact viscous damping coefficient* is similar to the *Stiffness Damping Coefficient* on the Stress page of the Material Edit Page except that it is only applied to the contact region.

This value usually should be a little higher than the actual material damping percentage in order to realistically reflect the contact physics.

Selected geometry defines contact boundary

This defines the selected geometry as the contact surface. In 2D analysis, the contact geometry can be geometry side, element side, analytical geometric line or planes. In 3D, this includes geometry surface, element face, or analytical geometric plane. These contact surface are used for penetration or sliding checking. Note that to be more efficient, not all nodes are checked against these contact surfaces. Only nodes specified as contact nodes are checked. By default, the nodes associated with the contact surface are also defined as nodes to be checked for contact. Note that once a geometry side/face is defined as a contact boundary, it will be used as a potential contact target in all other contact checking, even they are not in a same set.

Examine selections' associated nodes for contact.

This specifies that the nodes associated with the geometry selected are to be checked for contact penetration. In certain situations, it is necessary to specify additional nodes that are not associated with contact surface, but they should be checked against other contacting surface by using this assignment. Unless the contact surface will curl

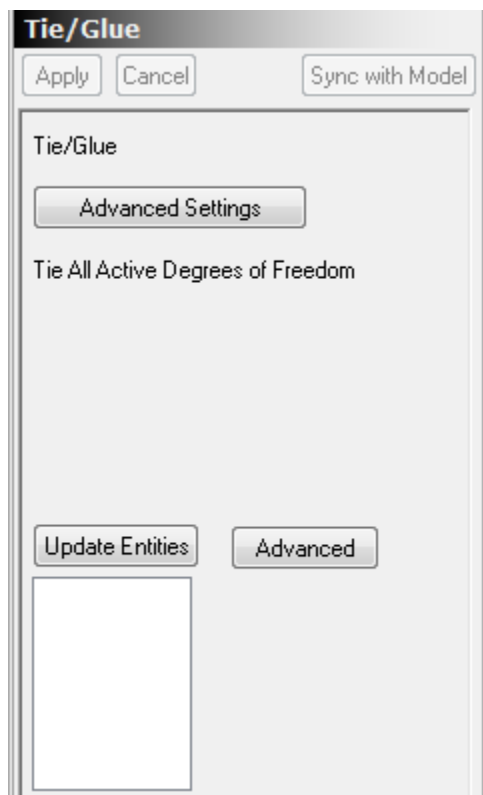
back and potentially penetrate/contact itself, it is not necessary to include the contact surface's nodes for contact checking. The default flag is to check this for self-contact.

Skip other contact geometry set (faster check)

In large problems, a quick proximity check is first performed for contact penetration test. The user can help speed up this process by putting contact boundary conditions in different sets, so the checking can be accelerated if this "skip other contact geometry set" check box is selected. The default setting is to perform the contact checking unless this check box is active.

E6) Advanced Constraint: Tie/Glue

Selecting **Tie/Glue** from the Advanced Constraints page adds a Tie/Glue constraint to the model and brings up the following Tie/Glue page:



The Tie/Glue constraint can be used to glue together two bodies that have been meshed independent and have independent nodes that do not match. It can also be used to apply periodic boundary conditions to a model. For periodic boundary conditions, it is necessary to set up the translation and/or rotation necessary to map the slave surface to the master surface.

The Tie/Glue condition

The Tie/Glue condition is often used to connect two separate geometric objects together when the meshes don't match. This is done by specifying a single "master" geometric entity along with one or multiple "slave" geometry objects to be tied/glued. If the master is a solid, surface or edge, the slave nodes (or transformed slave nodes) must be close to the master geometry to be tied. If the master is a single node, then all slave nodes are tied to the master

node and no proximity check is made. The proximity check and geometric tying coefficients are calculated based on the initial geometry and are not changed during the analysis.

In 3D analysis, the master geometry can be a solid (Solid Element or FE Element), a surface (Surface Elements or Solid Faces or FE Elem Face), or a single node (Vertex Node or FE Node). In 2D analysis, the master geometry can be a side (Edge Edges or FE Element Edge) or a single node (Vertex Node or FE Node). Only one master geometry per Tie/Glue condition can be used and the rest of the geometric objects are then tied to the master geometry based on geometric compatibility. The user can specify which of the slave/master DOFs are tied together. The user can also specify a Slave Ratio and have the tied slave DOF values multiplied by this Slave Ratio.

If the master geometry is a solid and initial contact/penetration is detected, the relative geometric ratio will be preserved by the tie/glue constraint. Also, unlike the contact condition, the definition of the master/slave geometry will be valid in the current set only, and will not influence other tie/glue condition.

Tie/Glue Advanced Settings

The screenshot shows the 'Tie/Glue' dialog box with the following settings:

- ☒ Tie all active DOF's (default)
- ☐ Dir 1 ☐ Dir 2 ☐ Dir 3 ☐ Rot 1 ☐ Rot 2 ☐ Rot 3
- ☐ Pressure (Also for Fluid Materials, Dir=Velocity and Rot=Vorticity)
- ☐ Thermal ☐ Electro
- Master to multiple-Slaves relationship
 - Master Entity (shown selected): [dropdown menu]
 - ☐ Scale DOF value by Slave Factor: 1
 - ☐ Apply transformation to the slave geometry.
(transformation applied before processing tie constraint)
 - Translation: x=0, y=0, z=0, units=meter
 - Rotation Origin: x=0, y=0, z=0, units=meter
 - Rotation Axis: x=0, y=0, z=1, units=meter
 - Rotation Angle (in degree, CCW about Rotational Axis is positive): 0

Buttons: OK, Cancel

Tie all active DOF's (default)

This specifies the tie/glue DOF's that should be enforced between the master/slave finite element data. For a geometric compatible tie/glue, the default "All DOF"s" tie should be used.

If **Tie all active DOF's (default)** is NOT checked, then only those DOFs check are constrained.

Master Entity (shown selected)

The **Master Entity** pull-down can be used to change the master entity from the first item in the Update Entities (geometry) list to any other item. The current displayed entity is shown as selected in the model so it can be identified.

Scale DOF value by Slave Factor

If Scale DOF value by Slave Factor is checked, then the DOF value (displacement, rotation, pressure, temperature or voltage) is scaled by the Slave Factor. If this is checked and the Slave Factor is 1.00, then the results are the same as if this wasn't checked.

Apply transformation to the slave geometry

This option will apply the specified translation/rotation to the slave geometry before processing the tie constraint. This is usually necessary when trying to perform some boundary constraint such as periodic or cyclic symmetric boundary conditions. In such cases, the master and the slave geometry items are on different areas of the model, and the proper constraint must be computed by translating or rotating the slave geometry before projecting it to the master geometry and checking the proximity.

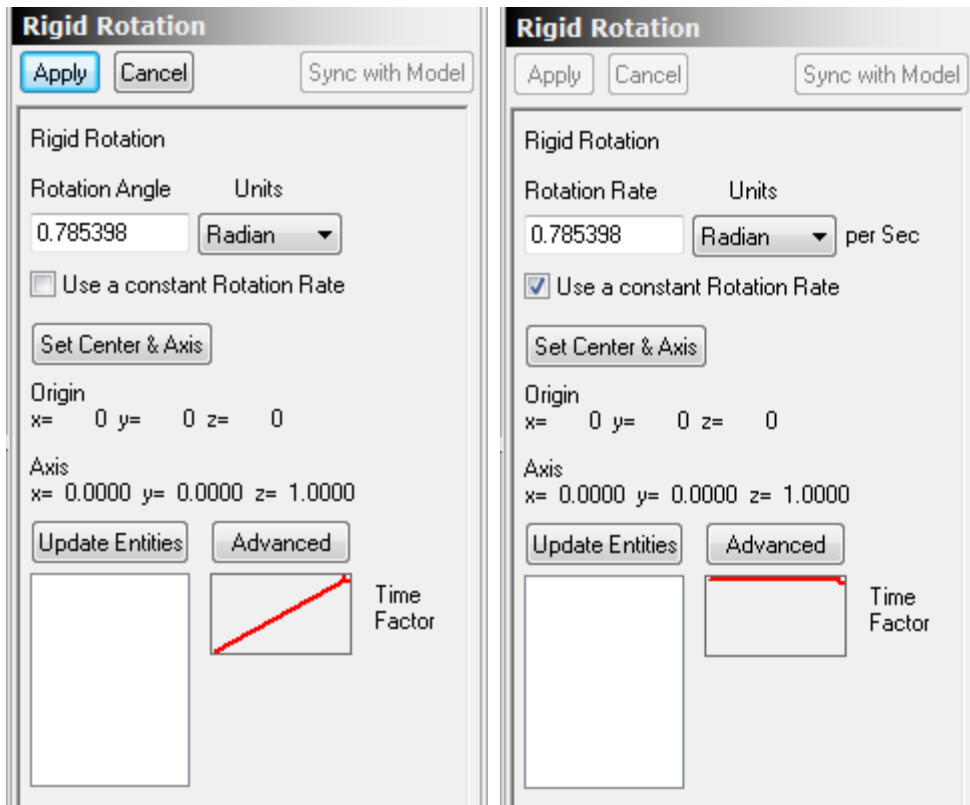
If rotational transformations are used, it is useful to use local coordinates for both the master and slave nodes so that Dir1, Dir2 and Dir3 on the master nodes correspond to the desired Dir1, Dir2 and Dir3 on the slave nodes.

Translation, Rotation Origin, Rotation Axis and Rotation Angle

These fields are used to entry the translation or rotation values associated with the transformation of the slave nodes before processing the tie/glue constraint. They are only used if **Apply transformation to slave geometry** is checked.

E7) Advanced Constraint: Rigid Rotation

Selecting **Rigid Rotation** from the Advanced Constraints page adds a Rigid Rotation constraint to the model and brings up the following Rigid Rotation page:



The nodes of the selected entities (Update Entities list) are constrained match a rotated position based on the Center and Axis of rotation.

Use a constant Rotation Rate

If *Use a constant Rotation Rate* is checked, then a fixed rotation rate per second is assumed and the angle of rotation is the simulation time multiplied by the Rotation Rate. The Time Factor is forced to a constant 1.00.

If *Use a constant Rotation Rate* is NOT checked, then the Rotation Angle is multiplied by the current Time Factor to determine the angle of rotation at a specific time.

Units

The Rotation Angle (or the Rotation Rate if *Use a constant Rotation Rate* is checked) can be entered in Degrees or Radians.

Set Center & Axis

This option can be used to set the rotation center and axis. The dialog page is similar to the page for setting the center and origin for cylindrical coordinates. The current center and axis are displayed on the Rigid Rotation page and a black double arrow icon is displayed on the screen at the rotation position. Also, the BC markers for the Entities that are selected for Rigid Rotation

are arrows in the direction of rotation based on the current Center, Axis and sign of the Rotation Angle/Rate.

Advanced: and ‘Use constant Rotation Rate’

If *Use a constant Rotation Rate* is checked, then the advanced options are ignored and the Time Factor is forced to a constant 1.0.

Advanced: and ‘Use constant Rotation Rate’ NOT checked

If *Use a constant Rotation Rate* is NOT checked, then the current Time Factor is multiplied Rotation Angle to determine the rotation angle for the current simulation time.

If *Use a constant Rotation Rate* is NOT checked, then the Off,On times from the Advanced page can be used to turn the constraint off. It is often difficult for the processor to converge if this constraint is turned off.

E8) Advanced Constraint: Nodal Stiffness

Selecting **Nodal Stiffness** from the Advanced Constraints page adds a Nodal Stiffness constraint to the model and brings up the following page:

Nodal Stiffness

Apply Cancel Sync with Model

Nodal Stiffness

Units lbf per inch

Dir Use Stiffness

X ☒ 0

Y ☒ 0

Z ☒ 0

XYZ

Update Entities Advanced

Time Factor

Nodal stiffness acts like a spring connecting the node to the original position of the node. The Stiffness entered represents the total stiffness for nodes on all the Entities in the Entities list. Based on the total

Stiffness, a nodal stiffness is applied to each node. For each 'Used' coordinate, a force equal to the nodal stiffness times the displacement in the 'Used' coordinate direction is applied to the node. This force is also multiplied by the Time Factor for multi-step and dynamic analysis.

Units

Both the force and the length units can be independently set.

Coordinate System: XYZ

The coordinate system of the nodes can be changed.

WARNING: Each node can have only one coordinate system. If a later (lower on the list) BC changes the coordinate system, that new coordinate system will be used for this BC also. The 'Move Down' menu item from BC right click menu can be used to move the current BC down and

Advanced

The Advanced button is used to change the Time Factor and also allows this BC to be turned off (and back on) as a function of time.

E9) Advanced Constraint: Nodal Damping

Selecting **Nodal Damping** from the Advanced Constraints page adds a Nodal Damping constraint to the model and brings up the following page:

Nodal Damping

Apply Cancel Sync with Model

Nodal Damping

Units **lbf** Sec per **inch**

Dir	Use	Damping
X	<input checked="" type="checkbox"/>	0
Y	<input checked="" type="checkbox"/>	0
Z	<input checked="" type="checkbox"/>	0

XYZ

Update Entities Advanced

SFace <38>

Time Factor

Nodal Damping creates a force proportional to the velocity in the opposite direction of the velocity. It tends to damp out motion in a simulation. The Damping entered represents the total Damping for nodes on all the Entities in the Entities list. Based on the total Damping, a nodal Damping is applied to each node. For each 'Used' coordinate, a force equal to the nodal Damping times the velocity in the 'Used' coordinate direction is applied to the node. This force is also multiplied by the Time Factor for multi-step and dynamic analysis.

Units

Both the force and the length units can be independently set.

Coordinate System: XYZ

The coordinate system of the nodes can be changed.

WARNING: Each node can have only one coordinate system. If a later (lower on the list) BC changes the coordinate system, that new coordinate system will be used for this BC also. The 'Move Down' menu item from BC right click menu can be used to move the current BC down and

Advanced

The Advanced button is used to change the Time Factor and also allows this BC to be turned off (and back on) as a function of time.

E10) Advanced Constraint: Nodal Mass

Selecting **Nodal Mass** from the Advanced Constraints page adds a Nodal Mass constraint to the model and brings up the following page:

Nodal Damping

Apply Cancel Sync with Model

Nodal Damping

Units **lbf** Sec per **inch**

Dir Use Damping

X ☒ 0

Y ☒ 0

Z ☒ 0

XYZ

Update Entities Advanced

SFace <38>

Time Factor

Nodal Mass creates mass to the nodes of the selected entities. The Mass entered represents the total Mass for nodes on all the Entities in the Entities list. Based on the total Mass, a nodal Mass is applied to each node. Note that a different mass can be used for each coordinate direction. Also the 'Use' button can be used to turn off the mass effect for any of the coordinate directions.

Units

Both the force and the length units can be independently set.

Coordinate System: XYZ

The coordinate system of the nodes can be changed.

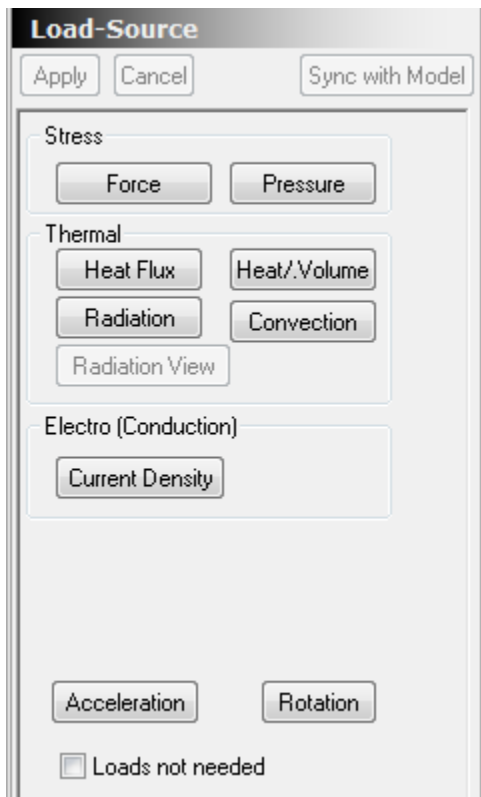
WARNING: Each node can have only one coordinate system. If a later (lower on the list) BC changes the coordinate system, that new coordinate system will be used for this BC also. The 'Move Down' menu item from BC right click menu can be used to move the current BC down and

Advanced

The Advanced button is used to change the Time Factor and also allows this BC to be turned off (and back on) as a function of time.

F) Load-Source

Clicking on the Constraint leaf brings up the Load-Source page. Depending on the physics allowed on the Analysis page, different buttons will be available. The following constraint page corresponds to Stress, Thermal and Electrical physics:



The **Loads not needed** checkbox is used to remove the '??' from before the Load-Source leaf name if you are using a simulation that does not require loads. Examples would be a dynamic simulation where initial velocities are used rather than forces to create the motion or a modal analysis where no loads are applied. It indicates that the person that designed the simulation designed it to be run without loads and did not just forget to add loads.

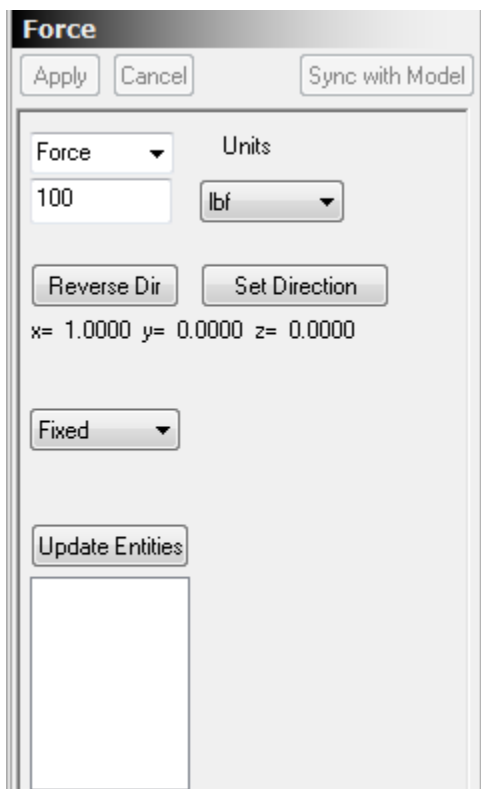
F-1) Force

Clicking the **Force** button on the Force-Load page adds a Force leaf under the Load-Source leaf and adds a Force page. The initial dialog corresponds to a 100 pound force in the global x direction. The **Update Entities** button must be used to specify the edges or faces on which the forces are applied.

Note: You can double-click on an Entity in the Update Entities list. This highlights the Entity and gives you an option to delete the entity.

The Force represents the TOTAL FORCE applied to all selected entities. If only surfaces are selected, the force is applied to the nodes weighted by area (like pressure). If only edges are selected, the force is applied based on length. It is not recommended that surfaces and edges are used for the same force BC. If they are, then the total force on all surfaces and edges will match the entered force.

The Load-Force>Acceleration option can be used is a mass dependent force (like gravity) is applied to the entire model.



The screenshot shows a dialog box titled "Force". At the top, there are three buttons: "Apply", "Cancel", and "Sync with Model". Below these, there are two input fields: "Force" with a value of "100" and "Units" with a pull-down menu showing "lbf". Below these are two buttons: "Reverse Dir" and "Set Direction". Below these buttons, the direction is specified as "x= 1.0000 y= 0.0000 z= 0.0000". Below this, there is a pull-down menu showing "Fixed". At the bottom, there is a button labeled "Update Entities" and a large empty rectangular box below it.

Units:

The pull-down specifies the units used for the force.

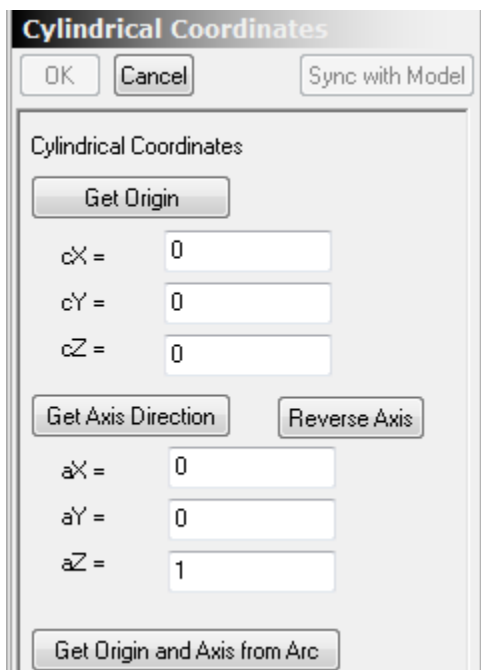
Coordinate System (Fixed pull-down)

The Fixed pull-down can be used to change the force from a fixed direction force to a force in Cylindrical or Spherical Coordinates. If Cylindrical or Spherical coordinates are used, the local coordinates at the selected nodes are changed to that coordinate system.

Caution 1: Constraints also use local coordinates if they are not applied in the global X, Y or Z directions. If a Load-Source>Force uses a coordinate system other than Fixed on a node that also has a local coordinate system defined, the Load-Source>Force coordinate system will take precedence over the earlier defined nodal coordinate system. Each FEA node can only have one set of local coordinates. Note that this does not cause a problem if the local coordinates match (e.g. cylindrical with same axis and both origins on the axis) and this is usually what the user wants.

Cylindrical and Spherical coordinates

If cylindrical or spherical coordinates are specified, the **Set Coordinates** button brings up a dialog that allows the origin and axis direction for the coordinate system to be set. The following page is used to set the origin and axis for cylindrical coordinates. A cylindrical coordinate system defined by an origin and axis direction. The R (radial) direction is away from the axis. The Th (theta) direction is perpendicular to the R and axis direction. The H (height) direction is along the axis (e.g. the height of the cylinder direction).



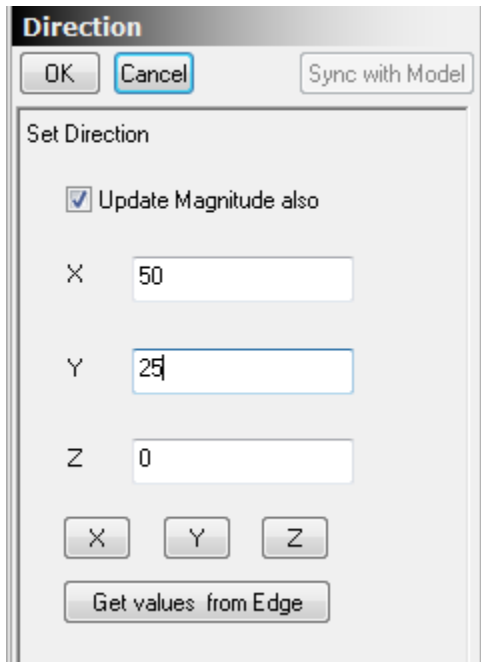
The screenshot shows a software dialog box titled "Cylindrical Coordinates". At the top are three buttons: "OK", "Cancel", and "Sync with Model". The main area of the dialog is titled "Cylindrical Coordinates" and contains several interactive elements. First is a "Get Origin" button, followed by three input fields labeled "cX =", "cY =", and "cZ =", each containing the value "0". Below these are two buttons: "Get Axis Direction" and "Reverse Axis". Following these are three more input fields labeled "aX =", "aY =", and "aZ =", with values "0", "0", and "1" respectively. At the bottom of the dialog is a button labeled "Get Origin and Axis from Arc".

Selecting the **Get Origin and Axis from Arc** button allows you to set both the Origin and Axis direction by clicking on an edge in the solid model that is a circular arc.

For Spherical coordinates, the allowable force component directions are R (Radial) away from the origin, Th (theta) right hand rule around the axis and Az (azimuthal) the R vector rotated 90 degrees in the R-axis plane towards the axis direction. R, Th and Az form a right handed coordinate system.

Set Direction

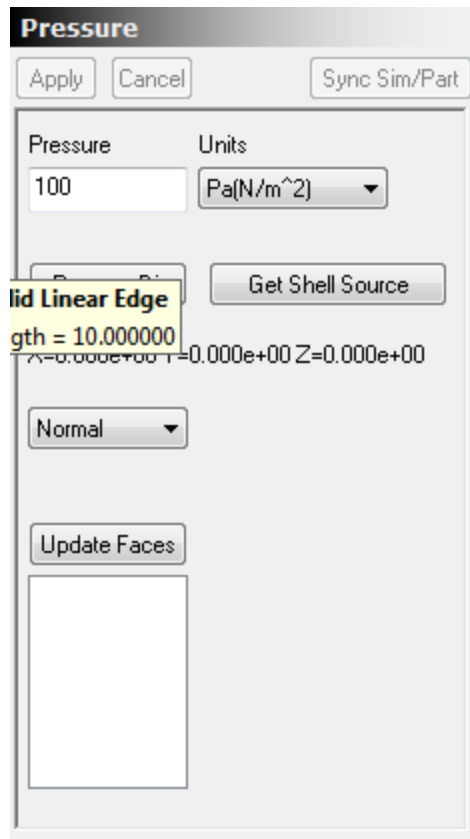
The set **direction button** is used to bring up the Direction page so that the direction of the force can be specified. Unless **Update Magnitude also** is checked, the X, Y and Z components are converted into a unit vector and the magnitude is not used. If **Update Magnitude also** is set, then the X, Y and Z values represent the force component in each direction. For example, with **Update Magnitude also** checked, setting the X component to 50 and the Y component to 45 creates a force of 50 in the X direction and 25 in the Y direction. This corresponds to a magnitude of 55.9017 and a direction of (0.8944, 0.4472, 0.0).



The **Get values from Edge** button can be used with Fixed direction to use an edge of the solid model to specify the force direction. Just click on the **Get values from Edge** button and then click on an edge in the model. The global X, Y and Z direction of the edge is used for the X, Y and Z components. If **Update Magnitude also** is checked, the magnitude of the force is not changed.

F-2) Pressure

Clicking the **Pressure** button on the Force-Load page adds a Pressure leaf under the Load-Source leaf and adds a Pressure page. The initial dialog corresponds to a 100 psi pressure in the normal direction. The **Update Entities** button must be used to specify the faces on which the pressure is applied.



1) Normal Direction

This option is used for pressure perpendicular to the surface.

Solid and Shell Normal Pressure Direction

For solids, positive normal pressure points in an outward direction.

For Shell models, positive normal pressure points away from the defined **Shell Point**. The **Get Shell Source** command can be used to set the Shell source Point. Currently for IronCAD, you must click somewhere on an existing part to create a shell selection point.

2) Fixed Direction

This applies a force proportional to the area in the fixed specified direction. The **Set Direction** button can be used to set the direction of the pressure. See [\(F1 Force Set Direction\)](#) for information on setting the direction.

3) Cylindrical and Spherical

This allows a force per area to be applied using cylindrical or spherical coordinates.. If Cylindrical or Spherical coordinates are used, the local coordinates at the selected nodes are changed to that coordinate system.

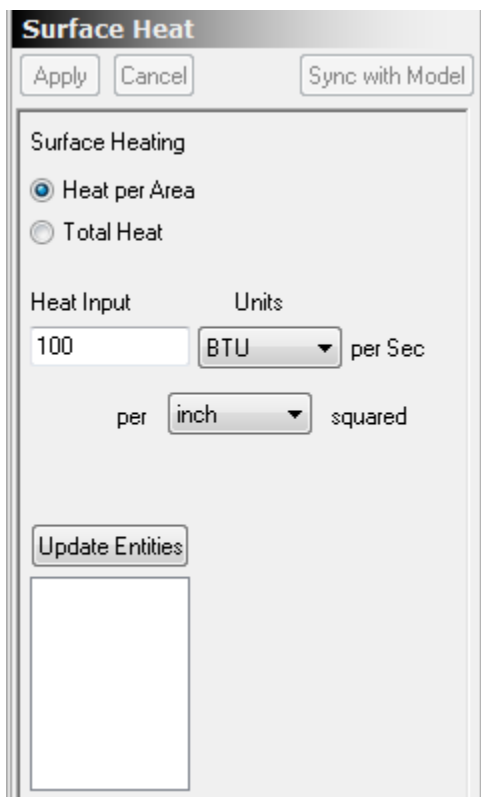
Caution 1: Constraints also use local coordinates if they are not applied in the global X, Y or Z directions. If a Load-Source>Force uses a coordinate system other than Fixed on a node that also has a local

coordinate system defined, the Load-Source>Force coordinate system will take precedence over the earlier defined nodal coordinate system. Each FEA node can only have one set of local coordinates. Note that this does not cause a problem if the local coordinates match (e.g. cylindrical with same axis and both origins on the axis) and this is usually what the user wants.

The **Set Coordinates** button can be used to set the parameters of the coordinate system. See [\(F1 Force Cylindrical and Spherical coordinates\)](#) for details.

F-3) Thermal Heat Flux

Clicking the **Heat Flux** button on the Force-Load page adds a Heat Flux leaf under the Load-Source leaf and adds a Heat Flux page. The **Update Entities** button must be used to specify the faces on which the heat is applied.



The screenshot shows the 'Surface Heat' dialog box. At the top, there are three buttons: 'Apply', 'Cancel', and 'Sync with Model'. Below these, the 'Surface Heating' section contains two radio buttons: 'Heat per Area' (which is selected) and 'Total Heat'. Underneath, the 'Heat Input' is set to '100' in a text box. To its right, the 'Units' section has a pull-down menu showing 'BTU', followed by the text 'per Sec'. Below this, there is another pull-down menu showing 'inch', followed by the text 'squared'. At the bottom of the dialog, there is an 'Update Entities' button and a large empty rectangular box for selecting entities.

The heat flux must be applied to a face (solid models) and the **Heat per Area** or the **Total Heat** can be specified. The units for both the heat can be specified using the **Units** pull-down. If **Heat per Area** is used, the units for the area can also be specified.

F-4) Thermal Heat/Volume

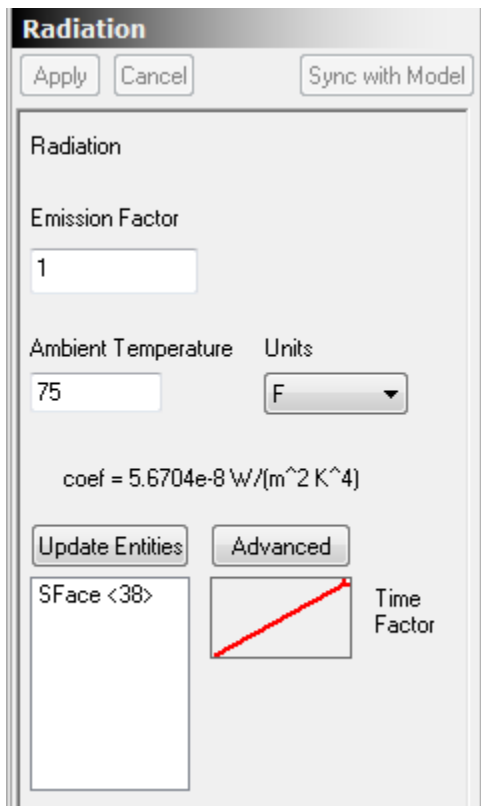
Clicking the **Heat/Volume** button on the Force-Load page brings up a reminder that Heat/Volume must be added as a material property and not a load. The page to set the default Heat/Volume is displayed by clicking on the Defaults button of a material page. The default values can be overridden by using the **Override Material Defaults** button on the body pages and entering a different Heat Gen value.

F-5) Thermal Radiation

Clicking the **Radiation** button on the Force-Load page adds a Radiation leaf under the Load-Source leaf and adds a Radiation page. The **Update Entities** button must be used to specify the faces that emit radiation.

The radiation uses the standard Stefan Boltzmann factor of $5.6704 \times 10^{-8} \text{ W}/(\text{m}^2 \text{ K}^4)$ (SI units).

The Emission Factor for the selected faces and the ambient temperature can be specified.



The screenshot shows a dialog box titled "Radiation". At the top, there are three buttons: "Apply", "Cancel", and "Sync with Model". Below these, the "Radiation" section contains an "Emission Factor" input field with the value "1". Underneath, there are two input fields: "Ambient Temperature" with the value "75" and a "Units" dropdown menu set to "F". Below these is a text label: "coef = 5.6704e-8 W/(m^2 K^4)". At the bottom left, there is an "Update Entities" button and an "Advanced" button. Below "Update Entities" is a list box containing "SFace <38>". To the right of the list box is a "Time Factor" input field, which is currently empty and has a red diagonal line drawn across it.

F-6) Thermal Convection

Clicking the **Convection** button on the Force-Load page adds a Convection leaf under the Load-Source leaf and adds a Convection page. The **Update Entities** button must be used to specify the faces that are affected by convection.

The **Convection Coefficient** and **Ambient Temperature** for convection is specified on the Convection page:

The image shows a software dialog box titled "Convection". At the top, there are three buttons: "Apply", "Cancel", and "Sync with Model". Below these buttons, the word "Convection" is displayed. The main section of the dialog contains two input fields. The first is labeled "Convection Coefficient: Units" and has a text box with the value "1" and a dropdown menu showing "BTU/(s in^2 F)". The second is labeled "Ambient Temperature Units" and has a text box with the value "60" and a dropdown menu showing "F". At the bottom of the dialog, there is a button labeled "Update Entities" and a large, empty rectangular box below it.

F-7) Electrical Current

For the Current button to be available, Electro with the Conductive option must be checked on the Analysis page.

Clicking the **Current** button on the Force-Load page adds a Current leaf under the Load-Source leaf and adds a Current page. The **Update Entities** button must be used to specify the faces on which current is applied.

If a material supports Thermal and Electrical Current properties, then joule heating is automatically included in the simulation.

The **Total Current** (on all the selected faces for this BC) or the **Current Density** can be specified. If Current Density is used, the area units can also be specified:

Electrical Current

Apply Cancel Sync with Model

Current

☒ Current Density
☐ Total Current

Current Units

100 Amps

per inch squared

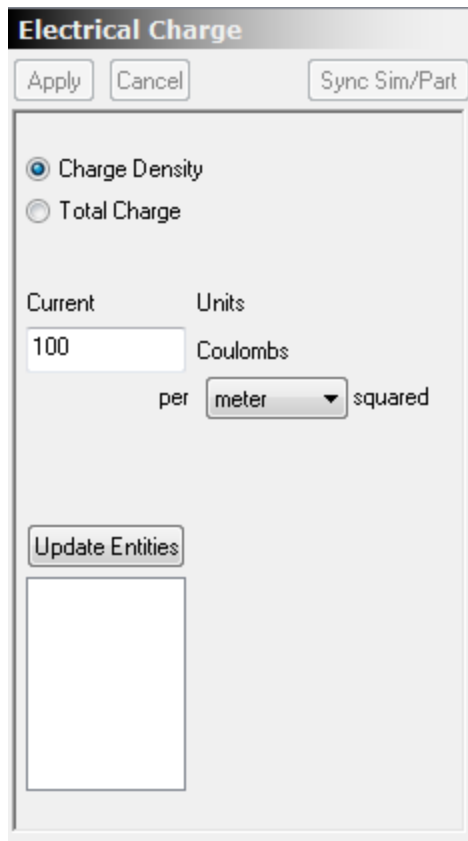
Update Entities

F-7a) Electrical Charge

For the Charge button to be available, Electro with the Dielectric option must be checked on the Analysis page.

Clicking the **Charge** button on the Force-Load page adds a Current leaf under the Load-Source leaf and adds a Charge page. The **Update Entities** button must be used to specify the faces on which current is applied.

The **Total Charge** (on all the selected faces for this BC) or the **Charge Density** can be specified. If Current Density is used, the area units can also be specified:



Electrical Charge

Apply Cancel Sync Sim/Part

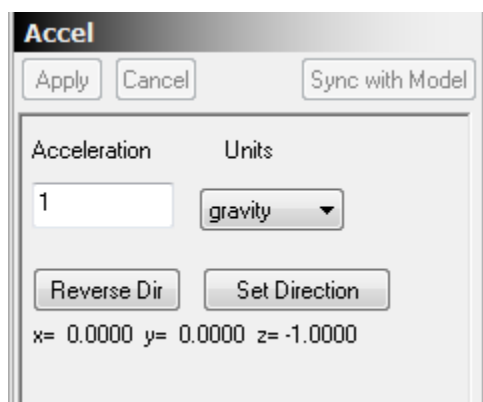
☒ Charge Density
☐ Total Charge

Current Units
 100 Coulombs
 per meter squared

Update Entities

F-8) Acceleration

Clicking the **Acceleration** button on the Force-Load page displays the Acceleration leaf under the Load-Source leaf. This is used to add a constant linear acceleration (like Gravity) to the entire model. You can only have one Acceleration leaf so if you click on the **Acceleration** button a second time, it just displays the single Acceleration Page:



Accel

Apply Cancel Sync with Model

Acceleration Units
 1 gravity

Reverse Dir Set Direction
 x= 0.0000 y= 0.0000 z= -1.0000

Units

The Units pull-down allows you to select gravity or a length unit. If a length unit is entered, then the acceleration units are that length unit per second squared. For example if meter was chosen, the units are meters/Sec²:

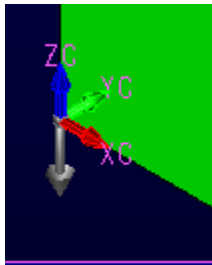
Acceleration	Units
9.8	meter /Sec^2

If gravity is used as the units, then the acceleration value is multiplied by 9.80665 meters/sec^2 (standard surface gravity).

Direction

The **Set Direction** and **Reverse Dir** buttons work the same as the similar buttons on the Force BC page for a **Fixed** force direction.

A gray arrow in the direction of the acceleration is added to the Axis at the origin:



F-9) Rotation

Clicking the **Rotation** button on the Force-Load page displays the Rotate leaf under the Load-Source leaf. This is used to add rotational forces to the model as if the frame were rotating. You can only have one Rotate leaf so if you click on the **Rotate** button a second time, it just displays the single Rotate Page:

Rotate	
Apply	Cancel
Sync with Model	
Rotation	Units
60	rpm
Get Center	
X=0.000e+00 Y=0.000e+00 Z=0.000e+00	
Reverse Dir	
Set Direction	
dX= 0.0000 dY= 0.0000 dZ= 1.0000	
<input checked="" type="checkbox"/> Include Coriolis Effect	

Units

The units for the rotation value can be rpm (revolutions per minute), rotations per second, radians per second or degrees per second.

Include Coriolis Effect

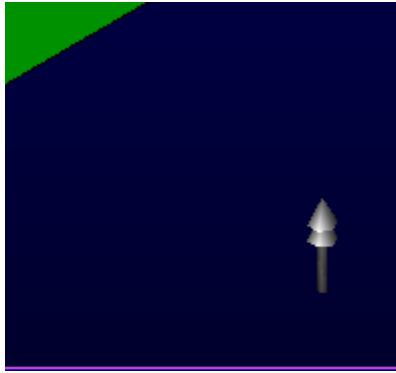
If checked then the Coriolis effective force is included.

Rotation Center and Direction

The **Get Center** button is used to change the center of rotation. In IronCAD, you can click on a node or anywhere on an existing part to select a point.

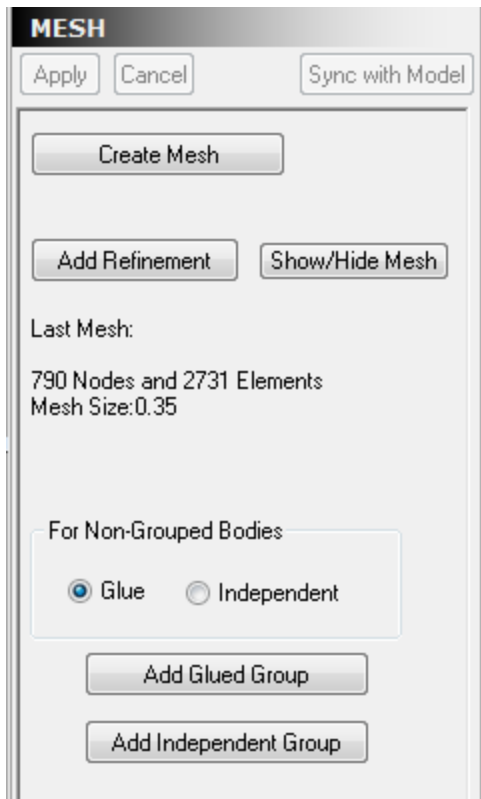
The **Set Direction** and **Reverse Dir** buttons work the same as for Fixed forces.

A gray double headed arrow is used to display the center and direction of the rotation in the model window:



G) Mesh

The solid model must be approximated by a mesh of tetrahedral (or hexahedral) elements in order for the FEA processor to calculate the results of the simulation. If the model has been previously meshed, the Mesh leaf name in the FEA tree displays the number of Nodes (N:), Elements (E:) and the mesh size (S:) used in creating the mesh.



Mesh Size

An appropriate mesh size should be chosen for the model. As larger mesh sizes require longer times to solve, the initial set-up and analysis is often done with a coarser mesh (larger mesh size) than the final analysis. Also, refinement is used to allow a smaller mesh to be used in critical sections of the model where the results or geometry varies on a smaller scale. Curvature refinement for the entire mesh is available on the **Mesh Generation** dialog obtained by clicking on the **Create Mesh** button. Refinement of selected surfaces and edges can be performed by using the **Add Refinement** button to add refinement leafs and pages to the Meshing section of the FEA tree.

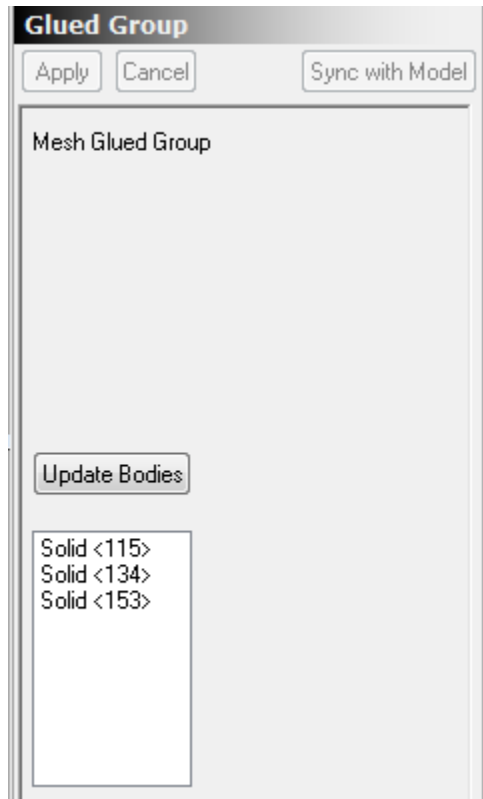
Connections between bodies

For most models, independent Solid Modeling bodies are used to represent different material. These bodies are 'glued' together and not expected to separate from adjacent material. The **Glue** option insures that all but the selected bodies are glued together. The **Independent** option, meshes each body separately. If the Independent option is used, a Constraint>Advanced>Contact boundary condition is often used so independent bodies do not 'pass through each other' during the simulation. Occasionally when the Independent option is used, a Constraint>Advanced>Tie/Glue boundary condition is used to tie two independent bodies together.

Add Glued Group (and Add Independent Group)

The Glue groups allow you to glue together the bodies in the Update Bodies list. These bodies are NOT glued to any other Bodies in the Model. This is useful if you have an independent part of your model

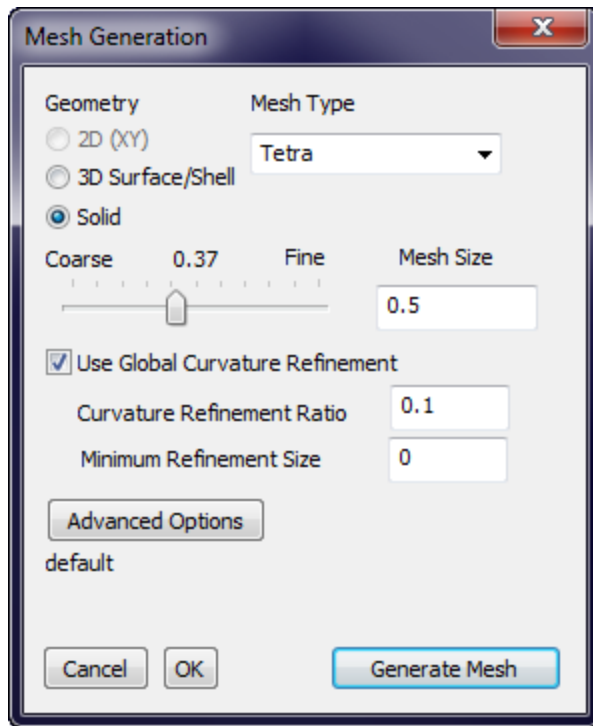
made up of multiple materials. Use the Glued Group to glue the bodies that represent different materials together.



Add Independent Group generates an Independent Group page. The bodies on Update Bodies list are NOT glued together and not glued to any other bodies in the model. If you have a model with many bodies glued together and a few bodies free to move, use the Glue option and then create an Independent Group for the bodies that are free to move.

Creating a new Mesh

The **Create Mesh** brings up the Mesh Generation dialog that is used to create a new mesh.



Cancel (x), OK, Generate Mesh

These are the three different ways to terminate this dialog. Generally, Generate Mesh is used. **Generate Mesh** causes the values to be saved and the mesh to be generated. **OK** saves the entered values for this session but does not generate the mesh. **Cancel** (or clicking the X in the upper right of the dialog) discards any changed and does not generate the mesh.

Geometry

If a model contains only Surface Bodies (and no Solids) it must be meshed with the **3D Surface/Shell** option. If a model contains Solid Bodies it must be meshed with the **Solid** option unless the **Mesh Solid as Shells** option is specified on the Model page. Because of the complexities in coupling shell node rotation and solid nodes (with no rotation) in the same model, creating a model with both Solid and Shell elements is not allowed.

All 2D analysis models must be meshed with the 2D (XY) option. The original model should be surfaces in the XY plane. If a surface is not in the XY plane (e.g. there is some Z component), the Z component is zeroed out after the mesh is created so that the mesh is in the XY plane.

Mesh Type

For Solid meshes, the user has the option of using Tetra (recommend) or Hexa (from Tetra) elements. Hexa (from Tetra) is used mostly for fluid problems (currently not supported) where 8 node hexa elements work better. Currently Sefea™ is not compatible with hexa elements. If Sefea™ is set as the FEM (on the Analysis>Options page) and a Hexa mesh is used, Standard FEM will be used for the analysis.

For Shell meshes, the following types are available:

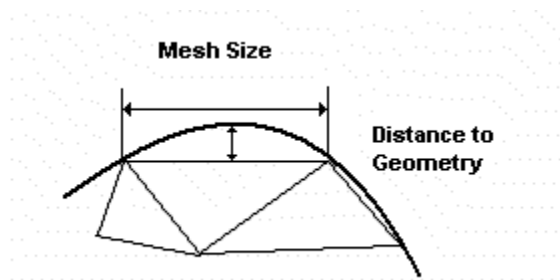
- 1) Tri Only (only triangles, recommended for Sefea™)
- 2) Mixed Tri and Quads (a mixture of triangles and quads)
- 3) Quads Only (only quad elements)

Mesh Size

The mesh size can be set by moving the mesh size slider control or by entering a Mesh Size value in the entry box under **Mesh Size**. The default value (shown between Coarse and Fine on the slider) is a rough estimate of mesh size based on the enclosing box of the model.

Use Global Curvature Refinement

If Use **Global Curvature Refinement** is checked, then the **Curvature Refinement Ratio** and **Minimum Refinement Size** are used to determine if the edge mesh needs to be refined. The Curvature Refinement Ratio is the ratio of the largest distance from an element to the actual solid geometry divided by the element's size:

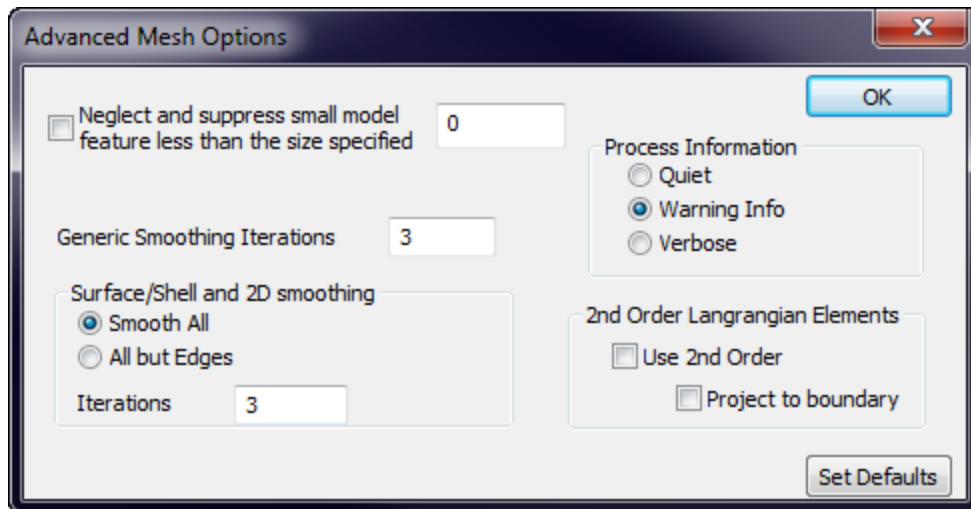


If this value is larger than the entered **Curvature Refinement Ratio** then the mesh is refined to reduce the error. The **Minimum Refinement Size** represents a minimum mesh size below which no further refinement is attempted. If **Minimum Refinement Size** is 0.0, then the default size of 5% of the mesh size is used.

Typically, the **Curvature Refinement Ratio** is between .1 and .4.

Advanced Options

The **Advanced Mesh Options** dialog is displayed when the **Advanced Options** button on the **Mesh Generation** dialog is clicked.



Neglect and Suppress Small Model Features

The small feature suppression function allows the user to detect small features in the model and suppress them. "Small Features" would be undesirable model faces/edges/regions created due to the unioning/imprinting of multiple model entities. Consider for instance two distinct parts that are supposed to touch on an edge but instead slightly overlap. Unioning/imprinting of these parts will cause a "sliver" face to be created at the area of overlap. This face is undesirable and is usually not intended. Small feature suppression allows the user to "erase" this sliver edge.

Consider a sphere of diameter equal to smaller feature size specified, moving along the length of a model face. If the face falls completely within the sphere, it is considered small. Model edges shorter than the tolerance are also considered small. Whenever possible, these model features will be skipped during the meshing process. These identified features will be highlighted during the meshing process. If the user decide to keep some individual feature, a specific mesh size must be specified on such feature (face, curve, or vertex) so the meshing process will respect these small features.

Generic Smoothing Iterations

This controls internal smoothing within the mesher.

2D and Shell smoothing

The **Smooth All** and **All But Edges** are currently not used. Edges are not smoothed to preserve any refinement of the model.

The **Iterations** count determines the number of smoothing iterations used.

The **Process Information** option controls the amount of information displayed during meshing. The **Process Information** options are currently not used.

Quiet: Displays very little information

Warning Info: (Default) displays warning information.

Verbose: Displays more detailed information (mostly for FlexMesh solid meshing).

2nd Order Lagrangian: Use 2nd Order

If checked, then 2nd Order Lagrangian Elements are created. If 2nd Order Lagrangian elements are used, then Standard element formation and not Sefea™ element formation is used.

2nd Order Lagrangian: Project to boundary

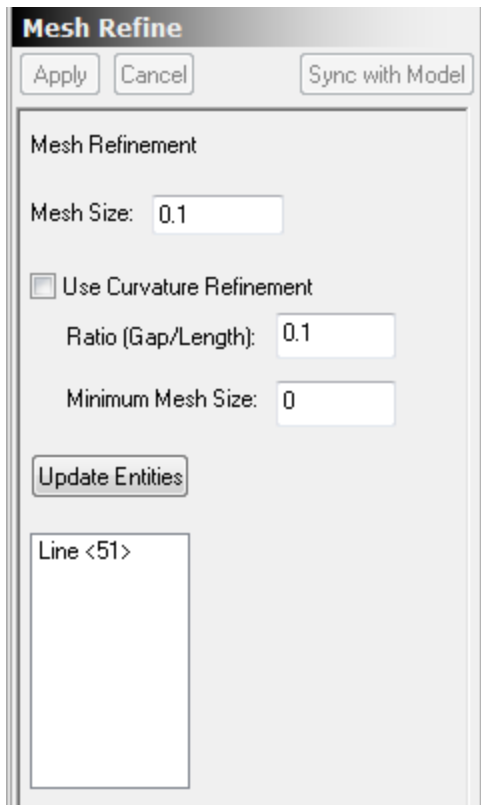
If 2nd Order Lagrangian Elements are generated and the *Project to boundary* option is checked, then all inserted 2nd order nodes on the boundary are projected to the associated geometric boundary face or edge. This provides more accurate boundary geometry for the 2nd Order Lagrangian elements. However, it can cause problems for some thin elements on highly curved boundaries. In this case, projecting the point to the actual boundary can move the point past the opposite edge and ‘invert’ the element. If this happens, the solver is likely to have difficulties in converging to a solution. An error is generated in the <model>.log file with the following form:

Warning J<=0! el=.....

This error can be eliminated by unchecking the *Project to boundary* option or by refining the mesh in the highly curved region.

Add Refinement

Clicking on the **Add Refinement** button of the Mesh page adds a Refinement leaf to the FEA tree in the Mesh section along with an associated **Mesh Refine** page. The newly added **Mesh Refine** page is displayed. The **Update Entities** option must be used to associate an Edge, Face or Solid with the refinement.



The image shows a 'Mesh Refine' dialog box with a title bar. At the top are three buttons: 'Apply', 'Cancel', and 'Sync with Model'. Below them is a section titled 'Mesh Refinement'. This section contains a 'Mesh Size' input field with the value '0.1'. Below that is a checkbox labeled 'Use Curvature Refinement', which is currently unchecked. To the right of the checkbox are two input fields: 'Ratio (Gap/Length):' with the value '0.1' and 'Minimum Mesh Size:' with the value '0'. Below these fields is an 'Update Entities' button. At the bottom of the dialog is a text area labeled 'Line <51>' which is currently empty.

Mesh Refine

Apply Cancel Sync with Model

Mesh Refinement

Mesh Size: 0.1

☐ Use Curvature Refinement

Ratio (Gap/Length): 0.1

Minimum Mesh Size: 0

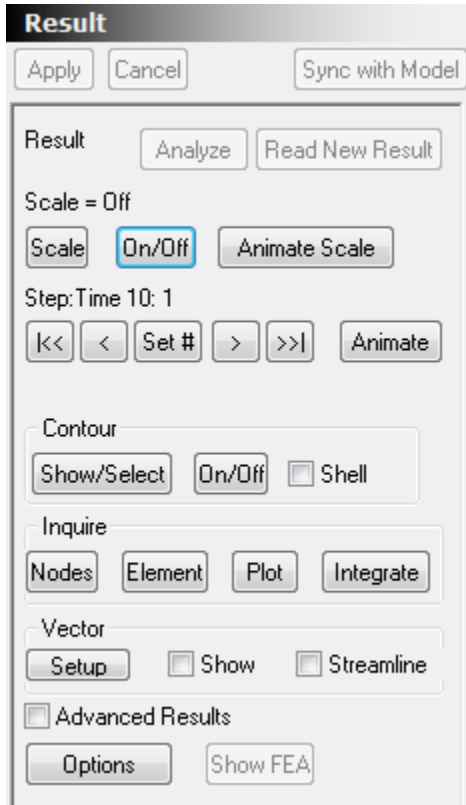
Update Entities

Line <51>

The **Mesh Size** is used as the mesh size for the associated entities unless **Use Curvature Refinement** is checked. If **Use Curvature Refinement** is checked, then the **Ratio** and **minimum mesh size** are used if they generate smaller elements than the global curvature refinement (if used) and the **Mesh Size** specified on this page.

H) Results

The Result page is used to analyze the model and examine the results of the analysis. If a change has been applied to the FEA model since the last analysis, the **Result** leaf name becomes **?&Result** to indicate the results are not valid.

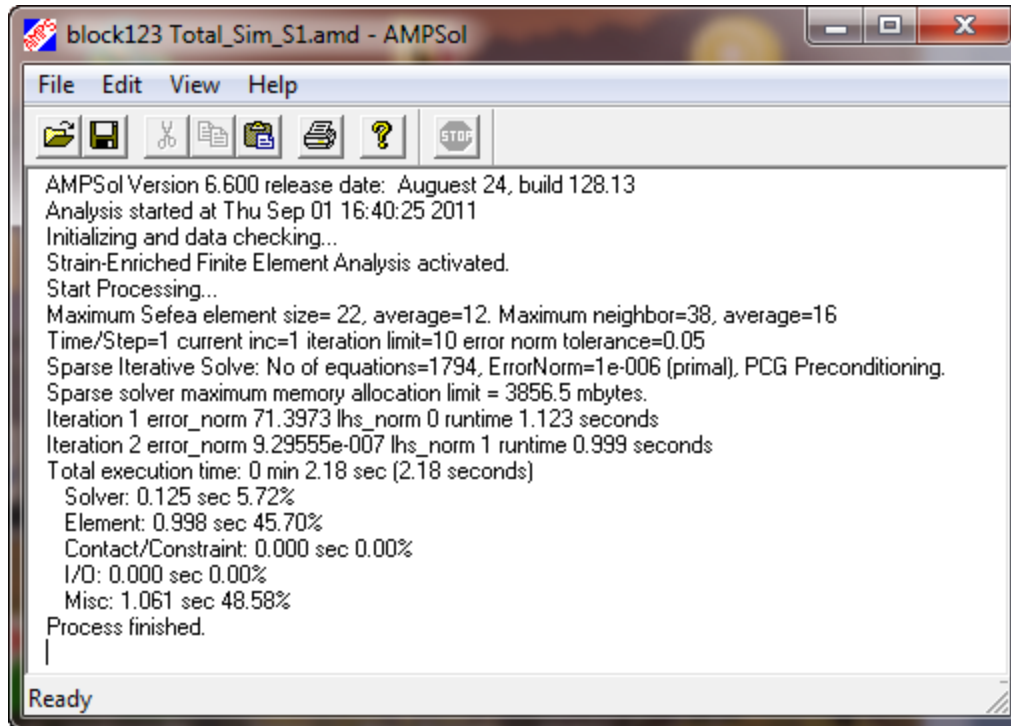


H-1) Analysis

The **Analyze** button is active if a change has been made in the FEA model since the last FEA analysis. If you need to Re-Analyze the model and the **Analyze** button is not active, click Options and then **Re-Analyze** on the options page. This is useful in the following situations:

- 1) You stop an analysis and then later decide to re-run the analysis without changes.
- 2) You have upgraded the FEA module and want to re-analyze some old models without changing the FEA information.
- 3) You feel the Analyze button should be available for whatever reason – but it is grayed out.

Clicking on the **Analysis** button starts the processor and opens the processor window:



If **Strain-Enriched Finite Element Analysis activated** is displayed, then the Sefea™ technology is being used. Sefea™ is not used if hexahedral elements, quad shell elements or 2nd order elements are used.

In multi-step analysis, the processor solves multiple steps. At each step, the error_norm, listed after Iteration #, should decrease and become less than Convergence Error norm on the Analysis>Advanced Opt dialog.

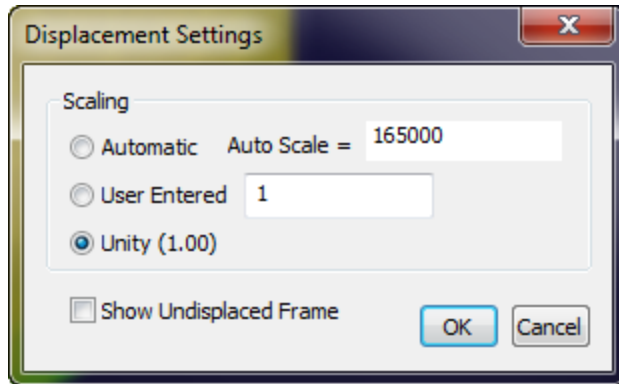
Once Process Finished is displayed, you can close the AMPSol processor window.

Read New Results

During multi-step analysis, the **Read New Results** button will be highlighted if new steps are available. Clicking on this button reads in the currently completed steps. The **Read New Results** button is not needed for single step analysis as the first time any of the result display buttons are pressed (Scale, Animate, Contour or Inquire), the latest results are read in.

H-2) Scale

For models with stress physics, the scale options allow a deformed view to be displayed with scaled displacements added to the node positions. Clicking on **Scale** brings up the Displacements Settings dialog:



Select **Automatic** if you want the program use an automatically calculated scale factor that scales the largest displacement to be about 30% of the model size.

Select **User Entered** if you want to enter your own Scale factor. This is useful if you are comparing several simulations of the same model and want the same scale for the displacements of all the models.

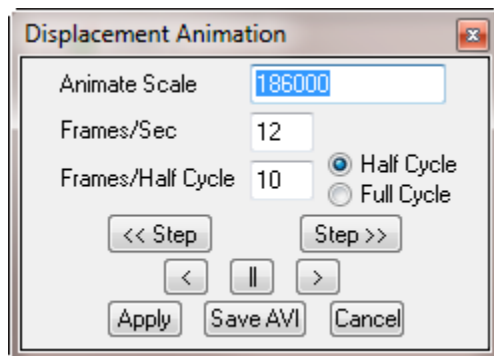
Select **Unity (1.00)** if you want to view the actual displacement. This is useful for Dynamic simulations where you are interested in the actual motion of parts of the model.

Click **OK** to have the displacement scaling applied to the model.

The current scaling is displayed above the **Scale** button on the Result page.

Animate Scale

The **Animate Scale** button brings up the Displacement Settings dialog. Usually you'll just use the Automatic option and the model will be animated by changing the scale factor from 0 to the Auto Scale value. Click OK to start the animation and open the Displacement Animation dialog:



The Displacement Animation dialog is used to change the parameters for the scale animation. The **Animate Scale**, **Frames/Sec** and **Frames/Half Cycle** can be changed by entering a new value and then clicking on the Apply button at the bottom left of the dialog.

Half Cycle and Full Cycle

If AnimScale is the animation scale, **Half Cycle** changes the scale factor from 0, to + AnimScale and then back to 0. **Full Cycle** changes the animation scale from -AnimScale to +AnimScale and then back to AnimScale.

Other Animation Controls

Increasing the **Frame/Sec** will increase the speed of the animation until the limits of the hardware are met.

Increasing **Frames/Half Cycle** will increase the number of intermediate frames in the animation and make it appear smoother. It will also increase the period of the animation cycle.

<< Step and Step >>

The step controls are used for multi-step models. They allow the previous (<<Step) or next (Step>>) to be displayed.

Previous, Pause and Next

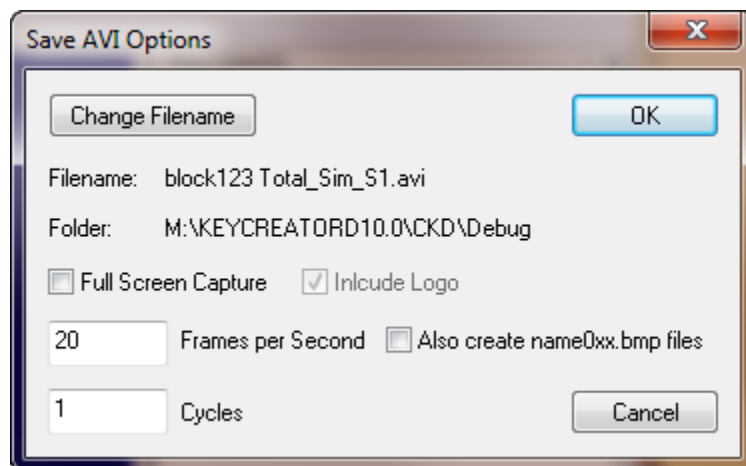
Clicking on the pause button, ||, pauses the animation. Once the animation is paused, the previous (<) and next (>) step controls can be used to change the scaling for the previous and next scaling in the animation sequence.

Save AVI Options

**** NOTE Save AVI has not been implemented for the IronCAD Alpha version. ****

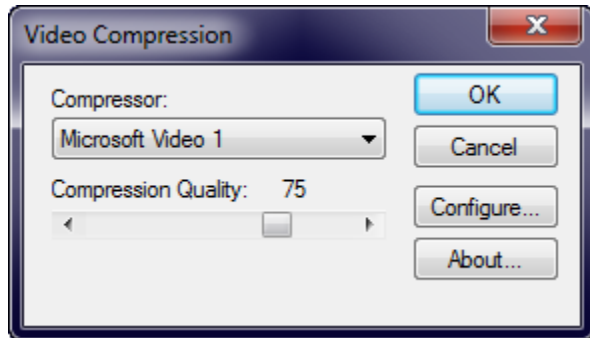
Note: If the AVI option does not work on your computer, try a different graphics mode. Change the pull-down in the upper right of the Tools>Options>Display>Graphics Card dialog. You will need to open a new window for the new driver to take effect. After creating the AVI file, you can change back.

Clicking on Save AVI displays the Save AVI Options dialog:



Use **Change Filename** to change the filename and/or folder of the .avi file (The default is to use the same filename and folder as the simulation file).

Clicking OK displays the **Video Compression** dialog. The **Microsoft Video 1** option is usually a good compromise between file size and quality.

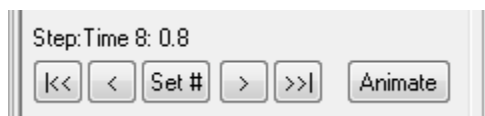


Click OK to create the animation and the AVI Completed dialog is displayed.

If you used the defaults, there should be a .avi in the same directory as the model.ckd file. It will typically have _Sim_X# append to the name where X is the first letter of the type of analysis and # is the number of simulation for this model.

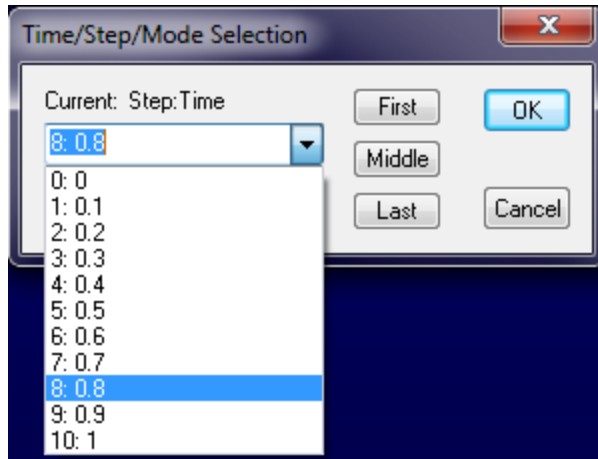
H-3) Multi-Step Results

If your simulation included multiple FEA steps, the multi-step controls are displayed on the Result Page between the Scale controls and the contour controls:



The current step and time is displayed above the controls. The buttons are First Step [**<<**], Previous Step [**<**], Set a specific step [Set #], Next Step [**>**] and Last Step [**>>**]. The **Animate** button starts a step animation that animates by increasing the step number.

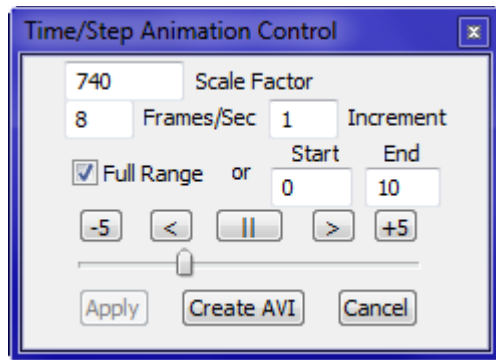
Clicking on Set # brings up a dialog that allows individual steps to be selected from a pull-down:



It also includes buttons to go to the First, Middle or Last step.

Clicking on the **Animate** button starts the step animation and displays the Step Animation Control dialog. If stress is used and the scale is zero (or Off), then the Displacement Settings dialog is displayed.

NOTE: The Auto Scale value suggested is for the current step. If you want to use Automatic scaling, you should pick one of the steps (usually the last) with the largest deflections.



The **Step Animation Control** allows you to change the Scale Factor, Frames/Sec or Step Increment used for the animation. If any of these fields (or the Start and End fields) are changed, the Apply button must be clicked for the change to affect the current animation.

The **Full Range** option is the default and allows the full range of steps to be used in the animation. If you are doing a step animation of a simulation in progress, the last step is the last step read in the last time the **Read New Results** button was clicked. To view a limited range of steps, enter the values in the **Start** and **End** fields, uncheck **Full Range** and click **Apply**.

When Pause is pressed

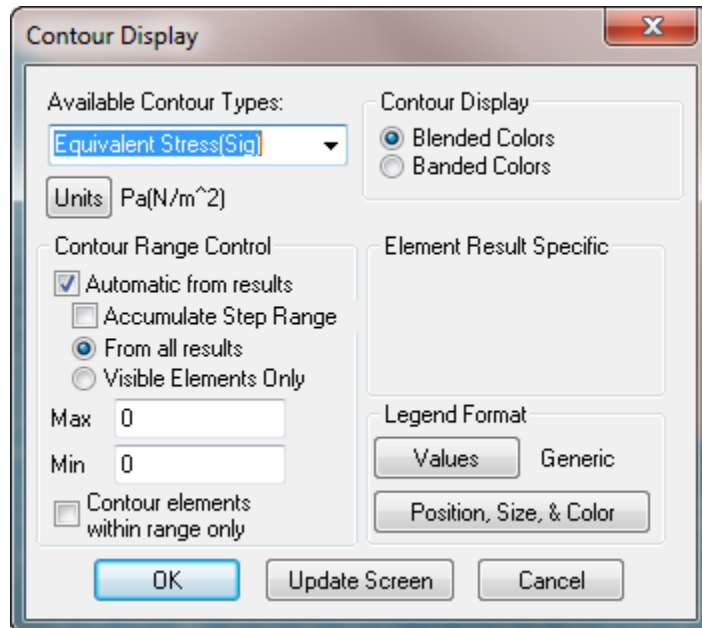
When Pause (||) is pressed, you can use the -5 or +5 buttons to move backward or forward 5 steps and you can use the Previous (<) or Next (>) buttons to go to the previous or next step. You can also use the slider control to change the step.

Create AVI

The Create AVI button works the same as the Animate Scale Create AVI button. See [\(Create AVI in Animate Scale\)](#) for more details.

H-4) Result>Contour

Select **Show/Select** under Contour to select the values to be contoured. The Contour Display Control dialog allows you to specify the output property to contour and other options that affect the location, format and range of values contoured:



The Contour Types pull-down is the control that you're most likely to change on this dialog. Initially this displays a list of output options that users are most often used. If the **Advanced Results** checkbox on the Result page is checked, the list includes all the output options generated by the processor.

If Shell elements are used and the (Contour) Shell checkbox is checked, then the Back Side, Front Side and Paint Visible Side Only are available – see Shell Contours at the end of this section.

Units Pull-down

The Units button on the Contour dialog allows different output units to be used for the next contour. Updated output units are shared between the various results options (Contour, Node Inquire and Plot) and are saved as part of the simulation when the 'Save Simulation' button is used.

Blended and Branded Colors

If **Blended Colors** is selected dithering is used to create a smooth/blended transition between colors. If **Branded Colors** is selected, the pure colors are displayed as bands.

Contour Range Controls


If **Automatic from results** is selected then the range represents the full range of the result type selected from the Contour Types pull-down and these values are displayed in the **Max/Min** entry fields the next time the Contour Control is displayed. If **Automatic from results** is NOT selected, then the values in the Max/Min entry boxes are used for the contour legend range.

For Step Animation with contouring, the default is for **Accumulate Step Range** to be checked so that after one cycle of animation, the contouring range represents the Max and Min of all the steps. If you are interested in viewing the Max/Min range for each step, turn the **Accumulate Step Range** option off and manually cycle through the steps.

If **Visible Elements Only** is checked, then nodes from elements hidden using the **FEA Hide** options from the model window Right-Click menu are not included in the automatic Max/Min range. Elements hidden only because they are off the screen from the current view are still included in the range.

Contour elements within range only

This option is usually used with Automatic from results turned OFF and the Max and Min values adjusted to show a limited range (usually top or bottom). FEA elements for which all the node values for the current Contour Type are hidden so that each FEA element displayed has at least one node with results in the Min/Max range. This option is useful when you want to identify the section of a large model that

has results within the specified range. You may want to turn off **Show Mesh Edges**, , when you use this command.

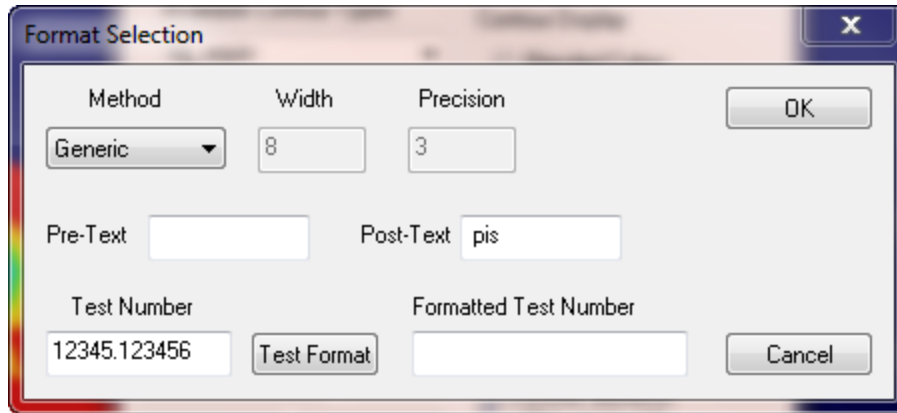
To redisplay all the elements after using this command, turn off **Contour elements within range only** and re-contour the model. Or, use the model window right click menu and select FE Unhide all.

Update Screen

This button updates the screen without closing the Contour Display Control. It is useful when you want to adjust the Max/Min range, the Format, the Legend position or just trying different Contour types without restarting the Contour Display Control.

Format

The Format option of the Contour Display Control brings up the Format Selection dialog and allows the format of the legend values to be changed:



Method

Select the method for formatting. All but the BestFit method are based on the C-style formatting options.

Fixed (Width, Precision)

Enter the width for the number and the number of places after the decimal point (Precision). If the formatted number doesn't fit within the Width, a larger width is used.

Exponential (Precision)

Enter the number of places after the decimal point (Precision). The number is displayed in the following format d.pppppe+XXX where the number of p's equals the precision and XXX represents the exponent.

BestFit (Width must be 6 or greater)

This is a good format if you are limited on room. It uses fixed or exponential precision whichever provides the most precision for the width. The exponent is reduced to 2 digits.

Generic

This corresponds to a C-programming language "%g" format. It will generally use 6 digits of precision and, if necessary, an e+0xx or e-0xx exponential added to the number.

Pre-Text and Post-Text

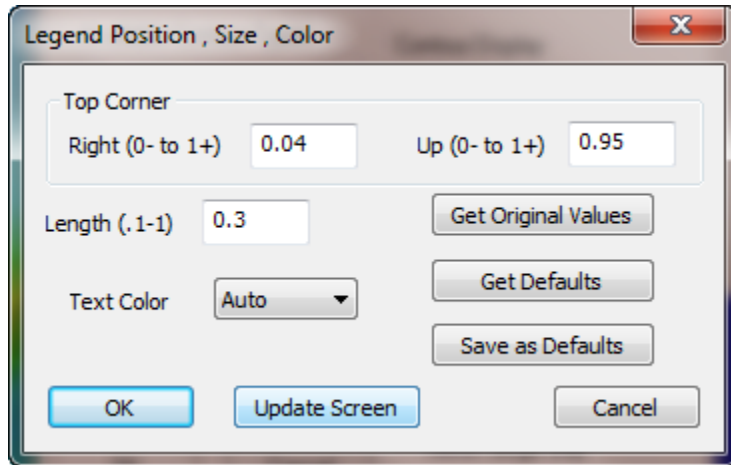
Text entered into the Pre-Text field is displayed before each contour legend number. Text entered in the Post-Text field is displayed after each contour legend number. These can be used to add units or other annotations to the contour legend for documentation screen shots or videos.

Testing Options

When the **Test Format** button is pressed, the **Test Number** is formatted based on the current format settings and displayed in the **Formatted Test Number** field.

Legend

The Legend button of the Contour Display Control brings up the Legend Position, Size, Color dialog that allows the contour legend properties to be changed:



Right (0- to 1+) and Up (0- to 1+)

These values determine the location of the upper left corner of the legend.

The coordinate system is based on a unit square centered in the model window and enclosed by the model window. For example, if the model window has a height of 10 and a width of 20, the up value would go from 0 to 1 (smallest dimension) and from -.5 to 1.5 (larger dimension). If the model window aspect ratio is changed, part or all of the legend may be outside the model window, but the relationship with the enclosed square remains the same.

Length (.1-1)

This value determines the length of the legend information.

Update Screen

This button updates the screen showing the position of the legend based on the current values.

Get Original Values

This button restores the original default values of $R = 0.04$, $U = 0.95$ and $L = 0.3$.

Get Defaults

This button restores the saved default values for the legend.

Save as Defaults

This button saves the current values as the new default values that are used each time AMPView is started.

Cancel

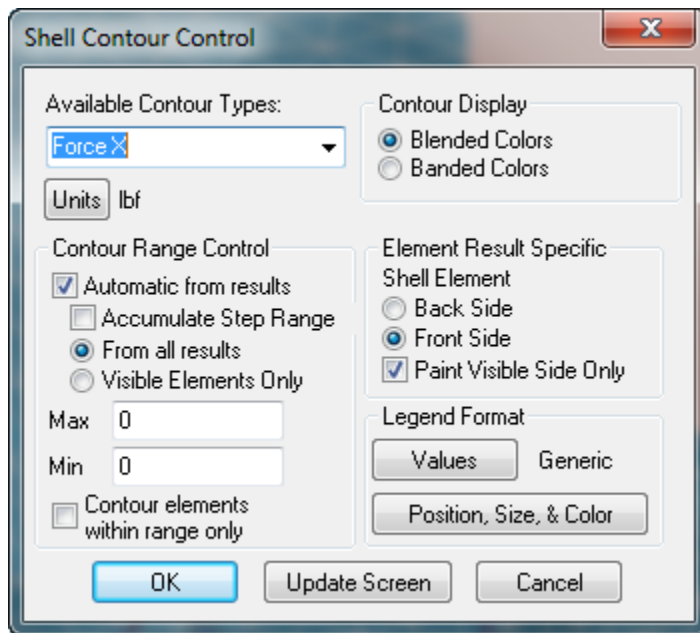
This cancels the Legend position and size changes change and reverts to the values in effect when this dialog was displayed.

OK

This button exits the dialog and saves the current legend position and size.

Shell Element Contouring

Shell elements provide element output at the nodes of each shell element. If the Result>(Contour) Shell checkbox is checked, then the Shell Contour Control is used:



This control is identical to the standard nodal contour control (Contour Display Control) except that the Contour Types pull-down contains results that are generated for shell elements and the **Element Results Specific Shell Element** options are active.

Back Side and Front Side

Shell Elements have a front side and back side based on element node ordering. Because shells have a finite thickness, the results on the front and back side can differ. If **Back Side** is selected, the contouring corresponds to the back side values. If **Front Side** is selected, the contouring corresponds to the front side values. If **Paint Visible Side Only** is checked, then only the front side (if **Front Side** is active) or the back side (if **Back Side** is active) face is painted. When the view is rotated so that the 'wrong' side is visible, the face is not displayed. This provides an easy way of identifying the front and back sides of shell elements.

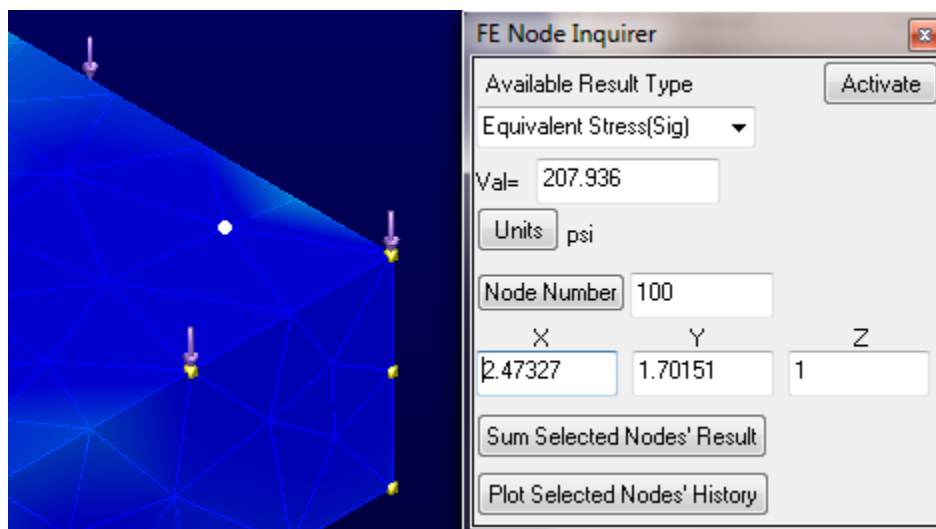
Discussion of FEA Element Results

Since the finite element results are usually nodal based (deformation, temperature, voltage, etc), those results involving derivative such as stress, strain, flux, etc. are computed after the basic nodal solutions are obtained. These derivative type results are then computed based on volume averaging from different elements' projection. Element based results are directly from the analysis without any extrapolation, as compared with the nodal results that are based on the volume averaging. They usually have slightly more oscillation than the nodal based result, but will give out more specific information for a specific element of interest. The shell element results also contain the global element end-forces, or forces and moments in the global direction. This is convenient for design purpose.

Note that for all element-based result, you can also inquire each element's corner value using the Result>Inquire by Element dialog to find out the exact value of each element.

H-5) Result>(Inquire) Nodes

The **Nodes** button in the Inquire section can be used to open a dialog that allows values at specific nodes to be using the mouse.



A white marker is used to highlight the node that corresponds to the values in the FE Node Inquirer dialog. The Result Type pull-down contains common output values if **Advanced Results** on the Results page is NOT checked and contains all the output values if **Advanced Results** is checked. Note that if Contouring is on, the FE Node Inquirer result type initially matches the contouring result type. But, it can then be changed independently of the contouring result type.

If no valid results are available, it is still possible to inquire about node positions using the FE Node Inquirer. To select and view a node based on the node number, just enter the value next to the **Node Number** button and click on **Node Number** (or press the <Enter> key after entering the number).

Summing Values at Selected nodes

The FE Node Inquirer dialog also allows the current Result value to be summed over the selected nodes. Use the selection toolbar:



To select the face or edge that contains the nodes to be summed. After you are done with selection, the **Sum Selected Nodes' Result** button can be used to sum specified results for the nodes associated with the selected geometry. If multiple faces are selected, nodes on the edges that are part of two selected faces are only included once in the sum.

The **Activate** button on the Inquirer dialog can be used to change the mouse operation from selection to Node Inquiry.

The Sum Selected Nodes' Results option can be used even if no results are available. In this case, it just counts the selected nodes.

Plotting Values at Selected nodes

As a shortcut, an option to **Plot Selected Nodes' History** button is included on this dialog. This is more useful when a Multi-Step (Dynamic or Non-Linear) analysis is used so that output steps represent the values at the nodes as a function of time (or pseudo time). Other plotting options are available using the **Plot** button on the Result page.

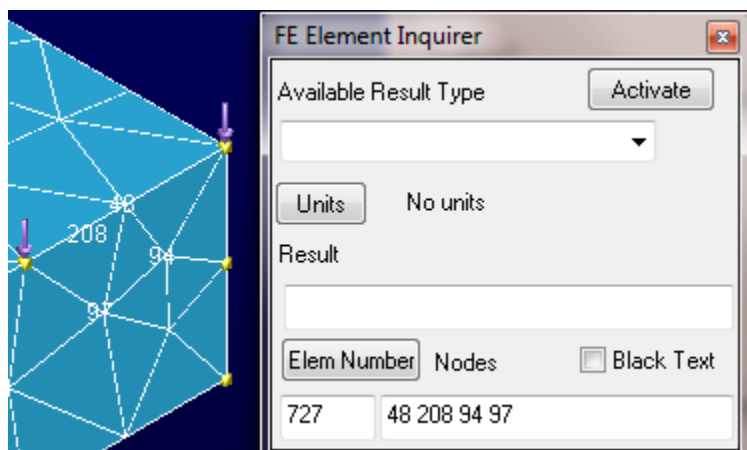
To use this option, use the selection toolbar:



to select the nodes to be plotted, and then click **Plot Selected Nodes' History** to plot the results at the selected nodes as a function of Time/Step.

H-6) Result>(Inquire) Elements

This option can be used (even without valid results) to display the nodes associated with each element. Just click on the **Element** button below Inquire to bring up the FE Element Inquirer dialog and move the mouse over an element.

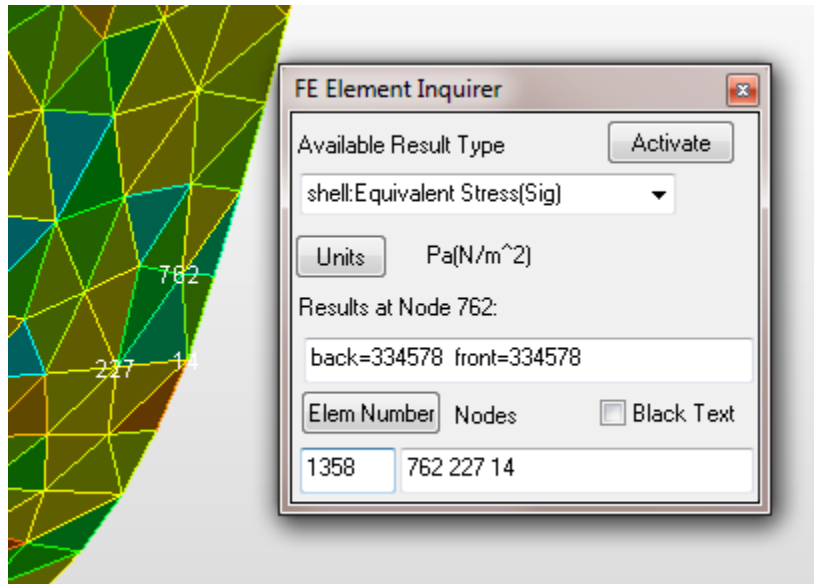


This can be used to verify that common nodes (between elements of different materials) have the same node number when the 'Glued' option is used to mesh the model.

To view a specific element by number, just enter the element number in the bottom left field and click the **Elem Number** (or press Enter).

Shell Elements

Shell elements generate element based results. If the Element Inquirer cursor is above a shell element, then the results for the back and front side of the shell are displayed:



H-7) Results>(Inquire) Plot

The **Plot** button in the Inquire section of the Result page can be used to bring up the plotting dialog (see below) to create various types of plots of nodal values. Three types of plots are available. The first two types are designed for multi-step (dynamic or non-linear) models that have multiple output steps. The last method plots the values of multiple nodes as a function of distance along a specified direction at the current time/step.

The selection toolbar should be used to select the desired geometry that contains the desired nodes before clicking the **Plot** button on the Plotting dialog:



Plot Multiple Node Results

Done Sync Sim/Part

Plot Selected Nodes

Equivalent Stress(Sig) Y-Axis

Units psi

☒ Results vs Time/Step

☐ Plot Sum

☐ Result vs Result (1 node, all steps)

Displacement X X-Axis

Units inch

☐ Plot along Direction (current step)

Set Direction

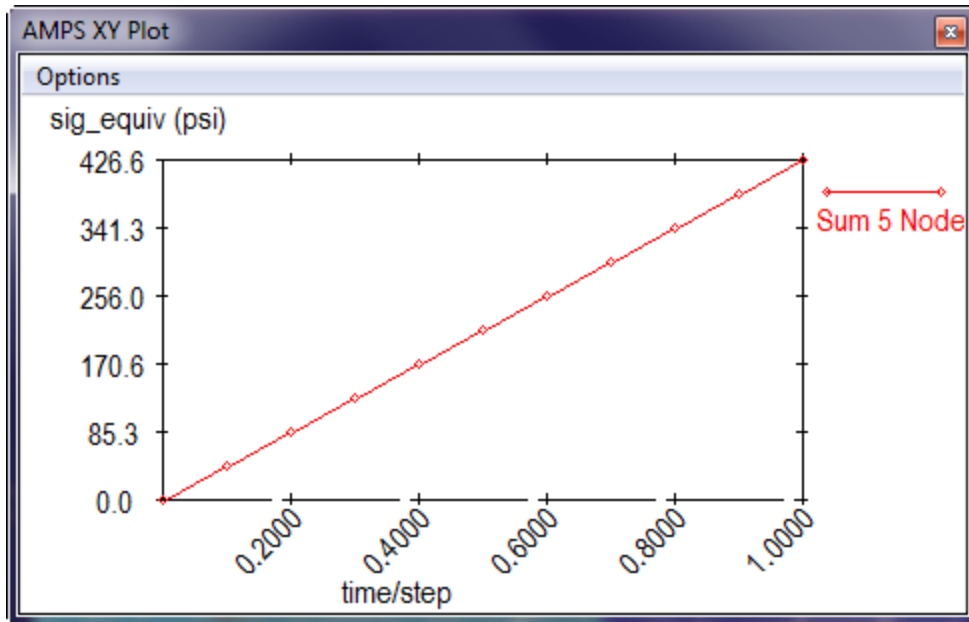
Direction
x= 1.0000 y= 0.0000 z= 0.0000

Plot

Results vs Time/Step

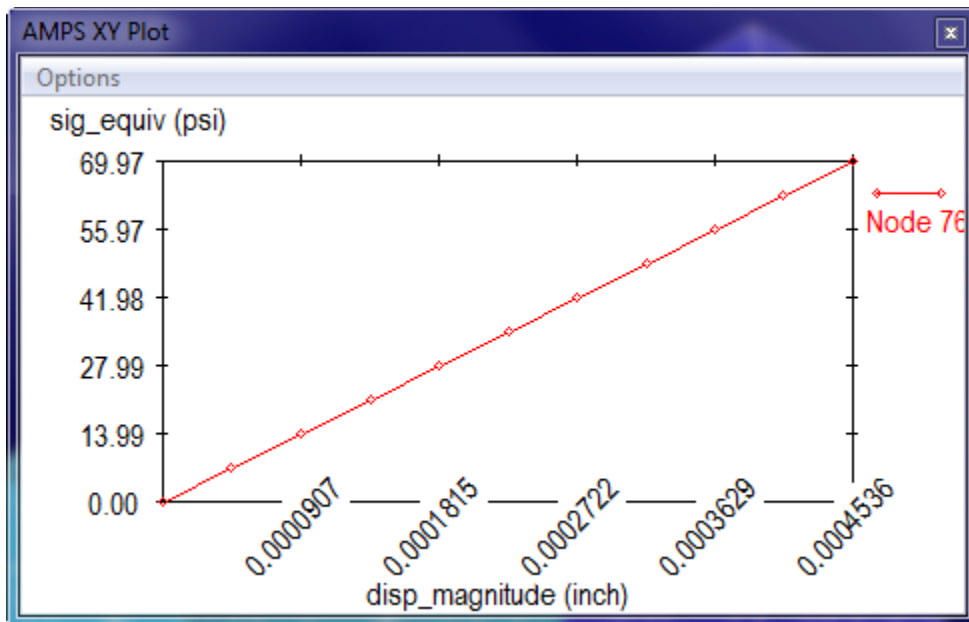
This option is the same as the **Plot Selected Nodes' History** shortcut button on the Inquire Node dialog. The pull-down allows the output parameter to be changed. If **Advanced Results** is checked, the pull-down displays all the output options generated by the processor.

The **Plot Sum** checkbox (not available from the Inquire Node plot option) allows the sum of the results for all the selected nodes to be plotted as a function of time:



Results vs Results (1 Node, all steps)

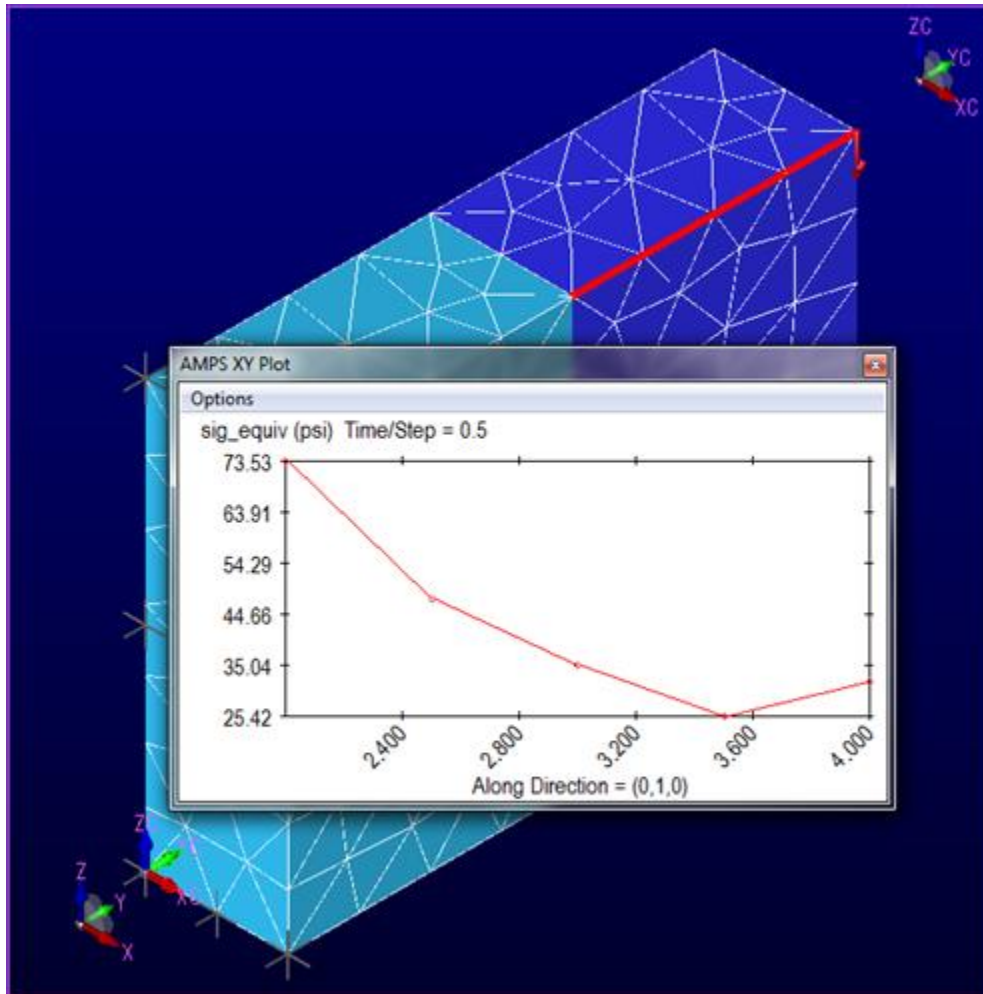
This option can be used to plot a curve of paired result values evaluated at one node at different time/steps. The vertical axis quantity is obtained from the top result pull-down and the horizontal axis quantity is specified by the second results pull-down. For example, a plot of sig_equiv vs disp_magnitude for node 76 for various time/steps is shown below:



Plot along Direction (current step)

If the Plot along Direction (current step) option is chosen, then the results specified in the top pull-down is displayed for all the selected nodes as a function of distance along the specified direction. The **Set Direction** button can be used to specify the direction manually, as an axis or based on an edge in the

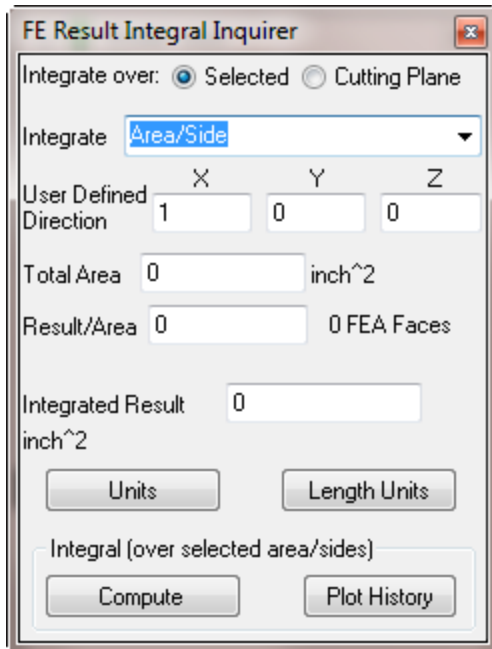
model. In the following plot, the sig_equiv values of the selected nodes are plotted as a function of the position of the nodes in the Y direction:



H-8) Result>(Inquire) Integrate

The **Integrate** button in Inquire section of the Result page is used to bring up the Fe Result Integral Inquirer dialog. This dialog allows output quantities to be integrated over selected surfaces or over a single cutting plane through the model.

Even if no results are available, the Integration dialog can be used to calculate the surface area of a selected FEA surface or the area of the FEA model that intersects a cutting plane. Note that all integration in this dialog is based on the FEA model and not the original CAD model.



Units Pull-down

The Units button on the Integral dialog allows different output units to be used for the next integral. Updated output units are shared between the various results options (Contour, Node Inquire and Plot) and are saved as part of the simulation when the 'Save Simulation' button is used.

If results are available, the Integrate pull-down includes not only the standard output from the processor but the following special quantities:

Area/Side: This can be calculate the Area (3D) of the selected faces (or cutting plane) or edges. It is available even before Results are calculated to assist in setting the force/Area in boundary conditions when you know the total force. For 2D Plane Strain and 2D Plane stress models, this option can be used to calculate the length of a side.

Heat Flow (Surface Normal): The heat flow calculation integrates dot product of the heat flux and the surface normal over the specified area/side. The result is the total heat flow across the surface/side.

Heat Flow (UserDir): The UserDir heat flow integral uses the user-specified vector instead of the surface normal. The resultant integral is the heat toward the direction of the user normal. The UserDir vector is normalized (length set to 1) and redisplayed for the calculation.

Electric Current: This is available for Electro:Conductive Analysis is the integral of the electrical current normal to the surface and also represents the total current passing through the surface or cutting plane.

Electric Charge: This is available for Electro:Dielectric Analysis is the integral of the displacement field normal to the surface. If the surface represents a conductor (so the displacement field is zero in the interior), the integral represents the total charge on the surface of the conductor. It probably does not make sense to use this with a cutting plane.



Coulomb Force (UserDir): This is available for Electro:Dielectric Analysis. It is the integral of the displacement field normal to the surface times the component of the electric field in the UserDir divided by 2. If the surface represents a conductor (so the displacement field is zero in the interior), the integral represents the total force (in a normalized UserDir) due to the induced charge on the surface of the conductor. It probably does not make sense to use this with a cutting plane.

Normal Force (SurfaceNormal): This is the integral of the force on the surface in the normal direction.

Shear Force (UserDir): This is the integral of the shear force on the surface. The UserDir vector is projected to the surface and then normalized for the calculation.

These are always available on the menu but display a warning if the associated physics is NOT included in the analysis.

Integrals over Selected Surfaces

The **Sel Face**, , or the **Sel FE Face**, , selection method should be used to select the faces for integration. The **Integrate** pull-down is used to indicate the quantity that is to be integrated. If **(UserDir)** is displayed on the **Integrate** pull-down name, then the **User Defined Direction X, Y and Z** is used to calculate the integral.

Integrals over a cutting plane

***** Cutting Planes have not yet been implemented in IronCAD *****

The FEA cutting plane icon can be used to generate a cutting plane through the model. If Cutting Plane (upper right of FE Result Integral Inquirer) is selected and a single cutting plane exists in the model when **Compute** is clicked, then the integration is performed over the intersection of the model and the cutting plane. If bodies are hidden using the model window right-click pop-up menu, then those bodies are not included in the integration.

Units

The default length unit used for integration is the model length unit from the Model Page of the FEA tree. The units displayed under the Integrated Results are correct but may not be reduced to the simplest form. For example, if the model length is feet, and a constant sig_equiv value of 2.5 psi is integrated over an area of 10 square feet, then the results are:

25 psi*foot^2

To convert this to pounds, you need to convert (foot/inch)² to (12 inch/inch)² or 144. So the result expressed in pounds is 25 * 144 pounds or 3600 pounds.

To avoid this calculation, you can use the Length Units pull-down to change the result units to inches. The results would then be 3600 psi * in² which converts to 3600 pounds.

Plot History

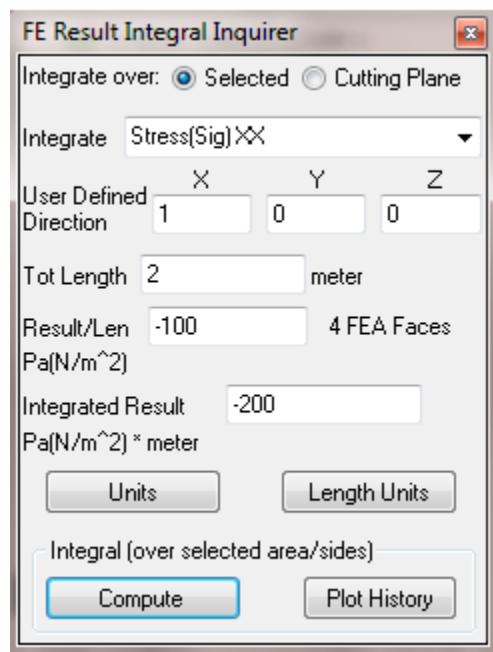
If the simulation is a multi-step simulation, the **Plot History** button can be used to generate a plot of the integrated quantity as a function of Time/Step.

Integrate for 2D Analysis

There are three types of 2D analysis Plane Strain (Uniform Z), Plane Stress (Z thickness) and Axisymmetric (RZ). For each of these methods the user selects an edge for integration instead of a face.

For Axisymmetric integration, the edge represents a surface that is obtained by rotating the edge 360 degrees about the Y axis. The units are the same as for 3D surfaces.

For Plane Strain and Plane Stress, an edge represents a surface of undetermined length in the Z direction. Consequently, the user must multiply by Z length to obtain the same units as the 3D case. The following is the Integration dialog for the Plane Strain or Plane Stress case:



The image shows a software dialog box titled "FE Result Integral Inquirer". It contains the following fields and controls:

- Integrate over:** Two radio buttons, "Selected" (which is selected) and "Cutting Plane".
- Integrate:** A dropdown menu showing "Stress(Sig)XX".
- User Defined Direction:** Three input fields for X, Y, and Z. X is 1, Y is 0, and Z is 0.
- Tot Length:** An input field with the value 2, followed by the unit "meter".
- Result/Len:** An input field with the value -100, followed by the text "4 FEA Faces".
- Pa(N/m^2):** A label for the units of the result/length.
- Integrated Result:** An input field with the value -200.
- Pa(N/m^2) * meter:** A label for the units of the integrated result.
- Buttons:** "Units", "Length Units", "Compute", and "Plot History".
- Footer:** A label "Integral (over selected area/sides)".

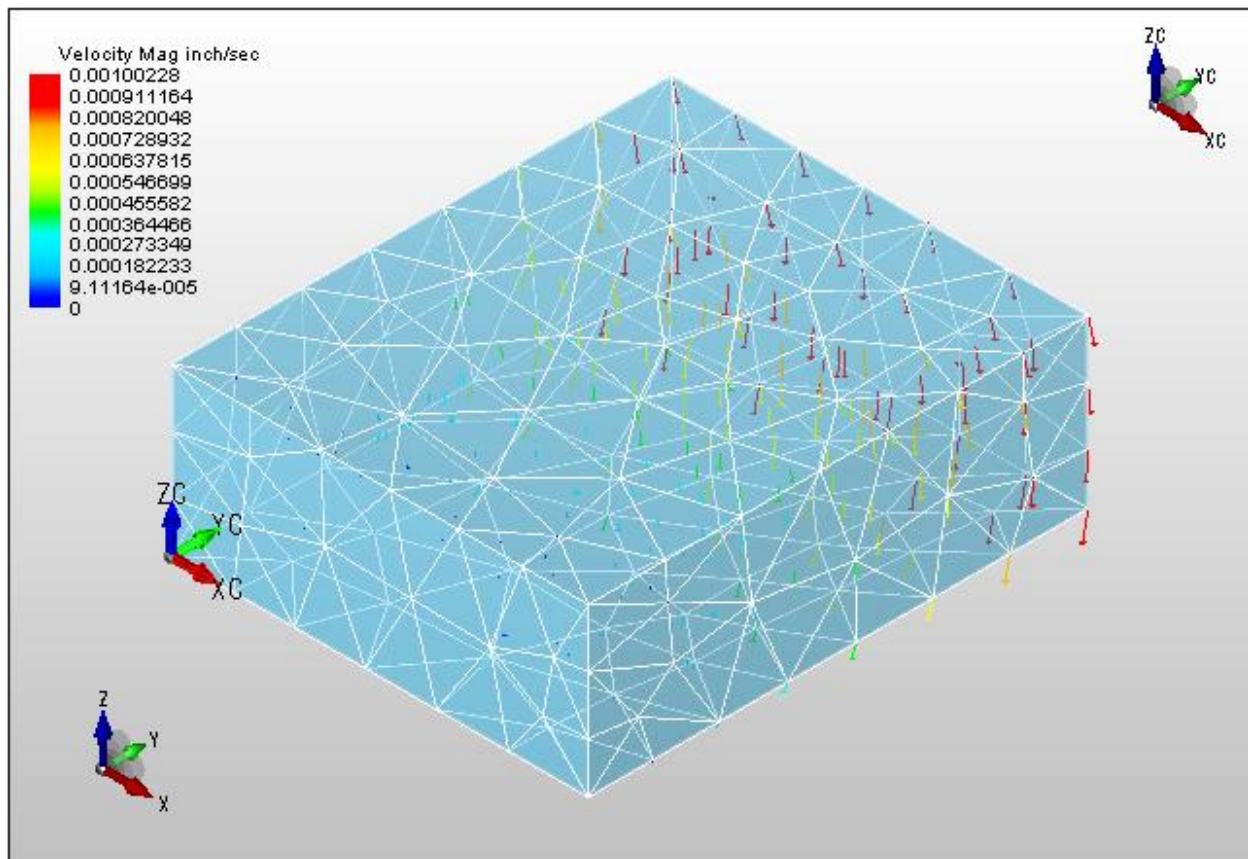
The Integrated Results is -200 (N/m²)*m or 200 N/m. For a plane stress model with a Z thickness of 0.1 meters, the force would be 200 N/m * 0.1 m or 20 N. Similarly for a plane strain model, the force along a 2 meter Z surface formed by extruding the edge 2 meters in the Z direction would be 200N/m * 2m or 400N.

The average Sig XX applied to the surface is -100 N/m². Note that this is also the average value of Sig XX along the selected edge.

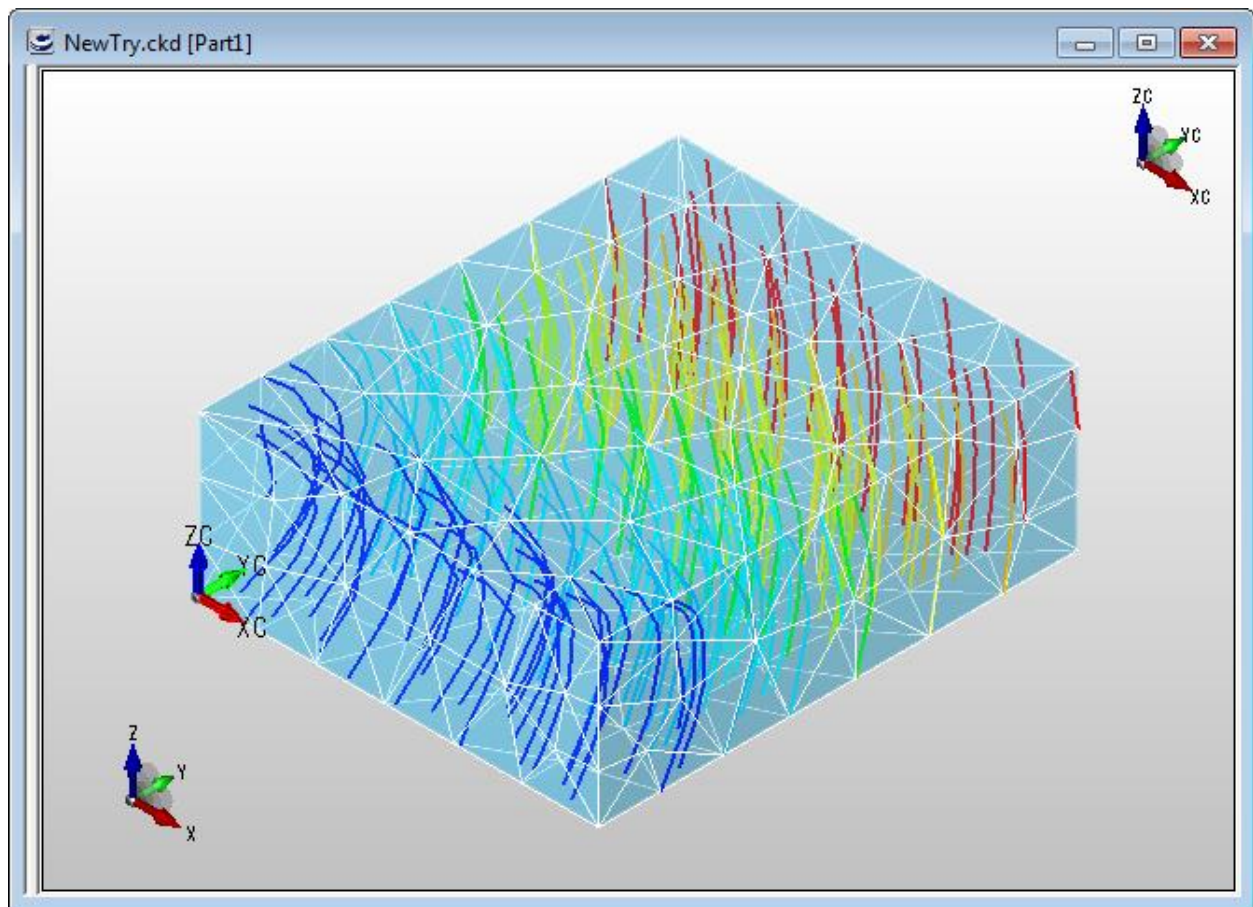
Plane Stress (Z thickness), if multiple thicknesses are used, the integral should be performed only on edges of the same thickness so that, when multiplied by the thickness, the correct value is displayed. The program does not do this automatically because a single edge can be shared between elements of different thickness.

H-9) Result>Vector

The vector section of the results page provides ways of viewing results as vectors. The Setup button can be used to setup the vector to be used. The **show** checkbox is used to display colored vectors at each node. A legend is generated and the color of the displayed vector corresponds to the magnitude color on the legend. The following shows the velocity vectors for a dynamic model. The 'Transparent Mesh Faces' option is used to make the inner vectors visible:



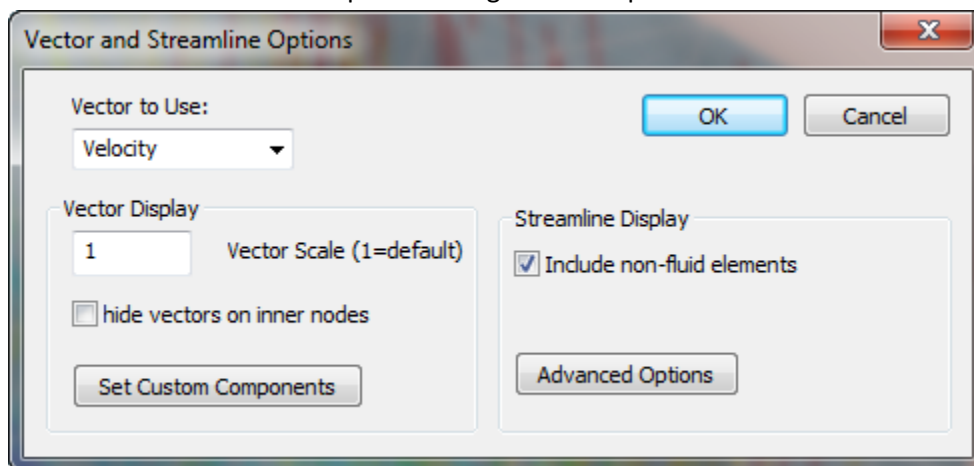
The Streamline checkbox can be used to toggle streamlines on and off. Both vectors and streamlines use the vector components specified by the Vector>Setup dialog. Streamline represent the paths of particles follow the direction of the local vectors. The following shows streamlines for the velocity field shown above:



Note that for dynamic models, the streamlines are calculated by using only the vectors at the current time.

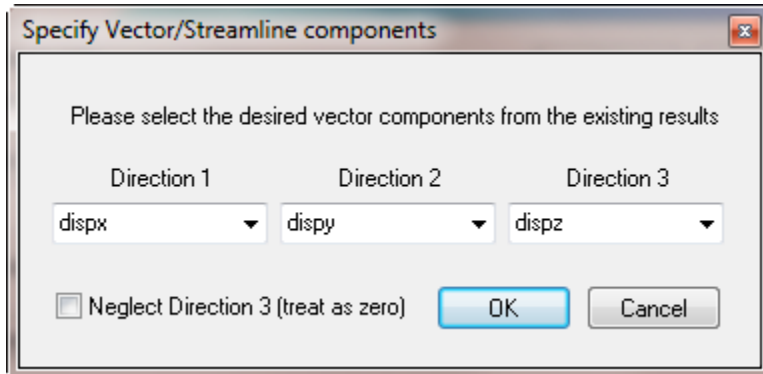
Vector>Setup dialog

The Vector and Streamline Options dialog includes options for vector and streamline display.



The **Vector to Use:** pull-down provides options for the various vectors that can be used based on the current analysis. The last options in the **Vector to Use:** pull-down is User Component. If this option is

chosen, the user can independently specify the vector x, y and z components to use based on the output from the processor. The **Set Custom Components** dialog is used to set the components:



Each Direction pull-down in this dialog can be independently set to one of the processor output quantities. The units for the legend are assumed to be the units of the first component.

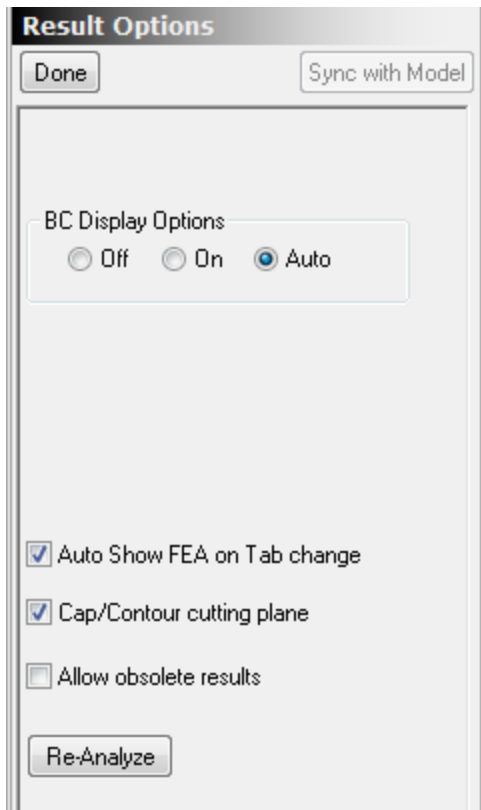
The **Vector Scale** option can be used to increase the length of the vectors in the display.

If **hide vectors on inner nodes** is specified, then vectors are not drawn on inner nodes.

The **Streamline Display>Include non-fluid elements** option should be un-checked if you are doing a fluid problem and don't want non-fluid elements to be included in the streamline.

H-10) Result> Options

The Options button on the Result page brings up the Result Option page:



Re-Analyze

The **Re-Analyze** button can be used to analyze a model again when the Analyze button on the Result page is not active.

BC Display Options

The symbols used to display boundary conditions on the CAD model can also be displayed when the results are displayed. Since these symbols identify the CAD faces and edges, they are not displaced when the FEA mesh is displayed with a non-zero scale factor. Sometimes it is useful to display these symbols along with the FEA results. There are three options for the visibility of the BC symbols. **Off** never displays the symbols. **On** always displays the symbols (but never displaced) and **Auto** (the default) displays the symbols when the displacement scale factor is 0.

Auto Show FEA on Tab change

This is useful if you have multiple simulation tabs and want to compare the results of different simulations. If **Auto Show FEA on Tab change** is NOT checked (the default), the FEA mesh is never automatically displayed when the Sim Tab is changed. Even if the Result page is being displayed and valid results are available. In this case, you can click on the Contour On/Off to restore the Contour and displacement scaling or you can click on the Scale On/Off to just display the scaling.

Because the FEA mesh can take a long time to display for models with a large number of FEA elements, the default option allows you to change Sim tabs quickly even when your model has lots of elements.

For models with fewer FEA elements, the FEA mesh can be displayed quickly when the Sim tab is changed and it is often useful to restore the mesh, scale and contour when the Sim tab is changed and the new Sim has the Results page active. Turning on **Auto Show FEA on Tab change** allows this to happen.

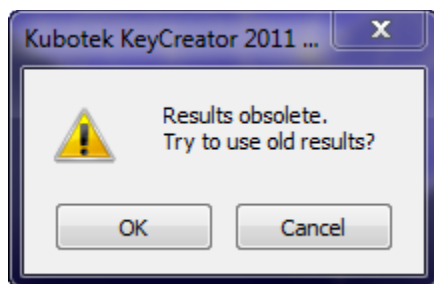
Cap/Contour cutting plane

If this option is checked, then the intersection of the cutting plane and the model is contoured when contouring is on. Uncheck this option if you want to be able to use the cutting plane to look inside a model. Make sure **Show Interior Elements** on the model window right-click menu is not checked if you want the model to appear hollow except for material boundaries.

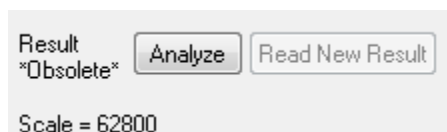
Allow Obsolete Results

Generally if any change to the FEA parameters has been applied, the results are assumed to be incorrect and can't be displayed. The Result>Analyze button is active to indicate that the model needs to be analyzed again. However, as long as the mesh hasn't changed, it is often possible to view the obsolete results. Turn on **Allow Obsolete Results** to have this option.

After turning on **Allow Obsolete Results**, if the Result>Analyze button is active, clicking on one of the results possibilities (Scale, Contour or Inquire) will display the following warning:



Click OK to use the obsolete results. To emphasize that obsolete results are being displayed, ***Obsolete*** is displayed under the word Result on the Result page:



I) Right Click Menus

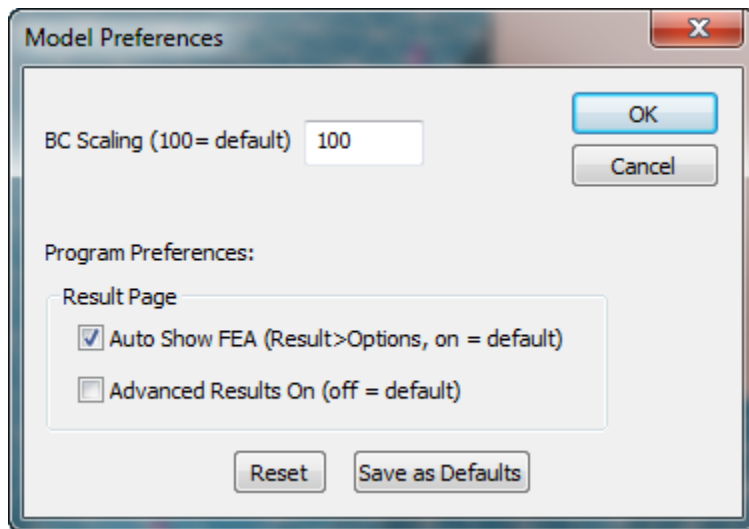
Many of the FEA tree leaves have Right-Click menus associated with them. The menu for the Analysis leaf contains options to delete the current Sim or to copy and optionally change the analysis type of the

current Sim. The BC leafs (Constraint and Load leaves) right menus contains options to disable/enable the BC as well as an option to delete the BC.

I-1) Analysis Right-Click Menu

Model Preference

The Model Preference dialog is displayed by selecting Model Preferences from the Analysis right-click menu. It is used to specify Model Defaults that can be saved between sessions.



Bc Scaling (100 = default)

This option controls the size of the BC symbols that are displayed on the model.

Auto Show FEA (Results, off=default)

This is the same as the **Auto Show FEA on Tab change** button on the Result>Options page. If checked, when a Sim tab is changed to a Sim that displays Results, the FEA mesh and any results are reloaded and displayed. Because loading the FEA mesh is slow for large modes, the default is off.

Advanced Results On

If checked, the 'Advanced Results' on the Results page defaults to checked. This is useful if you are frequently using some of the advanced result options for contours and node inquiry.

Reset

This resets the options on this page to their default values. These values are not saved as default values unless the Save as Defaults button is clicked.

Save as Defaults

This button saves the current settings as the default settings. The default settings will be in effect the next time the program is started.

Collapse Tree

This closes all the nested leaves in the tree except the upper level leaves.

Expand Tree

This expands all the nested leaves in the tree.

Copy Sim

This can be used to copy the current simulation and change the analysis type. It brings up the Select Analysis type and Dimension dialog. When the Sim is copied all material information is copied with it.

Delete Sim

This option is used to delete the current sim. After a sim is deleted it cannot be recovered. After clicking this, a verification dialog is displayed to verify that you actually want to delete the sim.

I-2) Material Right Click Menu

Disable/Enable

The enable/disable state of the material can be toggled with this menu. If the material is disabled, then none of the bodies associated with this material are included in the simulation.

I-3) Body

Disable/Enable

The enable/disable state of the body can be toggled with this menu. If the body is disabled it is not included in the simulation.

I-4) Constraint BC

Disable/Enable

The enable/disable state of the Constraint BC can be toggled with this menu. If the BC is disabled it is not included in the simulation. The title of the leaf starts with *D* if the leaf is disabled.

Increase Priority

This moves the Constraint BC down and essentially gives it a higher priority if a conflict occurs. If a conflict in BCs occurs, then the lowest BC is given priority. For example, if the top constraint constrained a surface to have a temperature of 125 degrees and the next constraint constrained a connected surface to have a temperature of 45 degrees, the common edge shared by both selected surfaces would use the lowest constraint and have the temperature constrained to 45 degrees.

Delete

This option can be used to delete a constraint BC. Once deleted, the constraint can't be recovered. Use Disable if you want to temporarily disable a BC and restore it for a later analysis.

I-5) Load-Source BC

Disable/Enable

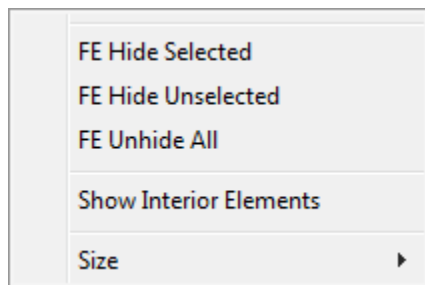
The enable/disable state of the BC can be toggled with this menu. If the BC is disabled it is not included in the simulation. The title of the leaf starts with *D* if the leaf is disabled.

Delete

This option can be used to delete a BC. Once deleted, the BC can't be recovered. Use Disable if you want to temporarily disable a BC and restore it for a later analysis.





I-6) FEA Mesh model Window

When the FEA mesh is displayed in the model window, a right click menu allows the user to hide/unhide FEA elements and control the display of edges, nodes and inner faces.



FE Hide Selected and FE Hide Unselected

NOTE: You should turn on **Show Interior Elements** BEFORE using FE Hide Selected or FE Hide Unselected if you are planning on hiding some of the elements on the surface of a solid. This insures that interior faces will be processed correctly.

Before using these options, you must select some FEA elements using the Sel Solid, , Sel Face, , Sel FE Solid, , or Sel FE Face, , selectors.

After selecting FEA elements or faces, **FE Hide Selected** is used to hide these elements. Or, after selecting FEA elements or faces, use **FE Hide Unselected** to hide all elements except those that are selected.

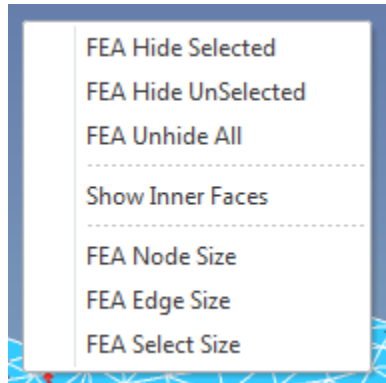
Show Interior Elements

This is usually off to speed up displaying the model as interior elements are not visible. It is important to turn this on if you are hiding surface FEA elements so that the interior faces will be visible. Note that this really refers to interior elements of the same material. The interior boundary between two different materials is visible and can be examined using the transparency option or cutting planes. The transparency option is the farthest right option on the display toolbar:



Size Options

The FEA xxx size menus allow the size of Nodes, Mesh Lines and Selected Nodes to be changed:



The default value for Node/Point is 0.24. The default value for Mesh Line Display Weight is 1. The default value for Selected Node Display Size is 0.3.

These values are not saved after IronCAD is closed.

J) Selection Hints

As selection is a common part of BC creation, here are some hints about entity selection that apply to the selection on all the BC pages.

1) When 'Update Entities' is active

When 'Update Entities' is active, the button name changes to 'Accept' and a 'Cancel Update' button appears for canceling the selection update operation.

2.1 **Clicking** on entities will **toggle** their selection. When 'Accept' is clicked, the currently selected set of entities becomes the new Entity list for this BC.

2.2 Holding **Alt and CTRL and SHIFT** allows you to select faces and edges that are hidden by other faces. If you don't hold down the SHIFT key, then all items are unselected and only the item from the Alt-Shift pop-up menu is selected.

2.3 Holding **CTRL** allows you to select an entire part. This is useful for specifying initial conditions. For example, you can set the initial temperature of the entire block to a fixed temperature. You can also use the IronCAD selection pull-down to change the type to 'Part'. Not all BCs can be applied to a full part. For example, pressure must be applied to a face.

3) Aborting 'Update Entities'

You can abort an 'Update Entities' operation by clicking on the 'Cancel Update' button. If you accidentally accept something you don't want, you still have the option of canceling ALL changes for the boundary condition by clicking the 'Cancel' instead of 'Apply' button in the middle of the FEA window.

4) When 'Update Entities' is not active

You can **double-click** on an entity name in the **Entities list**. This un-selects any preselected entities and displays the double clicked entity as selected. A dialog allows you to delete the selected entity from the list. Click 'OK' to delete. Click 'Cancel' if you used double-click to identify an entity.

The current selection changes along with all the other changes you made in the BC dialog will be discarded if you click on the page 'Cancel' button (just above the BC page) before clicking on the 'Apply' button.

5) Resolving visual selection ambiguities

On complicated models it is often difficult to determine exactly what is selected based on the BC symbols alone. Here are some techniques to help you identify items for a specific BC.

5.1) If you click on a BC leaf (e.g. Fix or Pressure) then only the BC symbols for that BC are displayed.

5.2) BC markers are displayed on edge lines of the entities associated with the BC. The number of markers on an edge line indicates the type of select entity (including end points): 4 for edge selection, 3 for each edge of a surface and one in the middle, 1 for each vertex of a solid and 1 for a single selected vertex. To avoid too many BC symbols, at most 20 selection symbols are put on each surface or body entity.

5.3) The text in the Entities list describes the types of entities selected. For parts, it will have the internal part number an underline and the part name (2_Part1). If the item is a Face, Edge or Vertex, then F, E or V and the index of that face, edge or vertex within the part will be added.

5.4) Clicking 'Update Entity' causes the entities for the current BC to be displayed as selected. This can sometimes resolve ambiguities. Press the [Cancel Update] to abort the 'Update Entities' operation without making any changes.

5.5) To be able to see BCs symbols that are within or on the back of the model, the 'Wire Frame' rendering option is useful. The CAD model is displayed as a wireframe, but the BCs are still visible.

6) Changing the BC symbol size

The default BC symbol size is based on a fraction of the model size. If you are zoomed into a small part of the model, the BC symbols may be large enough to hide smaller details of the model.). To change the BC symbol size, right click on the Analysis leaf and select Model Preferences from the menu. In the Model Preference dialog, make the BC Scaling number smaller (50 for half size, 25 for one forth the size etc.) and click OK. When you are done with selection you can change the BC Scaling back to a number that works well when the full model is displayed.

7) Hiding Bodies (IronCAD command)

Sometimes it is useful to hide some of the bodies in order to select interior faces. Also, multiple bodies are used in FEA models with multiple materials and it is sometimes necessary to select interior surfaces.

Right click on the parts to hide and select 'Hide Selected' from the right click menu can hide parts to hide the selected IronCAD parts. Right click anywhere and select 'Show All' to re-display all the parts.

If you hide or unhide IronCAD parts, IronCAD will assume the model has been changed and prompt you to save changes to the .ics file when you close the model.