

AMPView

User Reference Manual

Version 3.2, February, 2006

Copyright © AMPS Technologies Company 2003-2006

Table of Contents

AMPView	1
Welcome.....	1
Introduction	1
File Menu	3
The File Menu	3
Analyze Model	3
Read AMPSol Result	4
System Storage Files.....	4
Selection Menu	5
The Selection Menu.....	5
Selection Geometric FE Select.....	5
Selection FE Select.....	7
Selection Geometry Entity	7
Geometric Entities conversion to FE geometry	8
Analytical Geometry.....	8
Geomtry Line	8
Geometry Plane.....	9
Selection Filter	10
The Model Menu	10
The Model Menu.....	10
Model/System Menu	10
The Model/System/Controls Menu.....	10
The Model/System/Solution Iteration Control	14
The Model/System/Inertia Effects	16
The Model/System/Reference Frame Controls	17
The Model/System/Adaptive Controls	19
The Model/System/Unit Scaling Controls	21
Time/Step Dependent Data Control.....	22
Model/Define Menu.....	23
The Model Define Properties Menu	23
The Model Define Element Behavior - Stress Menu.....	25
The Model Define Element Behavior - Thermal/FLow Menu	28
The Model Define Element Behavior - Electro/Magneto Menu.....	30
Anisotropic/Orthotropic Material Definition	31
Creep Material Data and Controls Menu	33
Piezoelectrical Material Data	35
Nonlinear Property Multiplier.....	35
Beam/Rod Section Properties Menu.....	36
User Defined Constutive Material Model	38
User Defined Initial Stress and Strain.....	39
User Defined Plasticity Material	39
Material model and data	40
Model/Conditions Menu	40
The Boundary Condition Menu	40
The Model Condition Menu.....	41
The Model Condition Geometry Menu	42
The Model Condition Time Menu.....	44
The Model Condition Type Menu.....	46
Additional Arc-Length Controls	49
Nodal Local Coordinate System Controls.....	50
Basic nodal DOF choices.....	52
The Model Condition Values Menu.....	52
The Model/Condition Thermal Convective/Radiative Page	55

Table of Contents

The Model/Condition Surface Contact Condition.....	57
The Model/Condition Tie/Glue Condition.....	59
The Constraint Equation Condition.....	60
User Defined Load/Flux/RHS/LHS Condition.....	62
The OverSet Condition.....	63
Model/Material Library Operations.....	64
The Model Define Material Library Menu.....	64
Result Menu.....	65
The Results Menu.....	65
Contour Display.....	65
Result Contour Controls.....	65
Shell/Element Contour.....	67
Beam/Rod Contour.....	68
Vector Display.....	69
Vector Display.....	69
Vector Display by Component.....	70
Deformation.....	70
The Results Deformation Display Menu.....	70
Deformation Animation.....	70
User Result Type.....	72
User Results.....	72
Customized User Result Type.....	72
Normal/Shear Stress by direction.....	74
Deleting Generated User Result.....	74
FE Inquirer.....	74
FE Inquirer by Node.....	74
FE Inquire by Element.....	75
Inquirer Time Plot.....	76
FE Inquirer by Integral.....	76
Finite Element Menu.....	77
FE Group/Display.....	77
Create/Modify/Inquire Nodes.....	79
Create/Modify/Inquire Element.....	79
Coat Shell Elem on the Selected Elem Faces.....	79
Coat Beam/Rod element on the Slected Element Sides.....	80
Finite Element Graphic Display.....	80
Optimize Mesh.....	81
Import External Model.....	81
View Menu.....	81
The View Menu.....	82
Display Menu.....	82
The Display Menu.....	82
Cutting Plane.....	83
Cutting Plane Menu.....	83
Options Menu.....	84
The Options Menu.....	84
Option Visibility.....	84
Option Render Mode.....	85
Option Color.....	86
Option Size.....	86
Option Text.....	87
Window Control Menu.....	88
The Window Control Menu.....	88
File Toolbar.....	89
View Toolbar.....	89
Display Toolbar.....	89


Select Geometry FE Toolbar	89
Result Display Toolbar	90
Cut Plane Toolbar	90
Help Menu.....	91
The Help Menu	91
Finite Element Features of the Solvers.....	91
Element side or surface Boundary Condition Types.....	91
Tutorials	92
AMPSol Tutorial.....	92
Index	93

AMPView

Welcome

AMPView is a Finite Element pre and post processor with interactive graphic interface. It is planned to be the link between the meshes that we generate from solid models in AMPSolid and the full range of data required for a Finite Element Analysis. The user may call our AMPSol solvers directly from AMPView or generate data for another F.E.Solver. This manual is organized with an overview in the introduction section. Then each of the top menu items is fleshed out in individual chapters. This is followed by several tutorial examples. Throughout the manual we will be using a quarter gear solid generated in AMPSolid and stored as gear_st.dat. The _st denoting that it is a file for the stress and thermal analysis of the quarter gear.

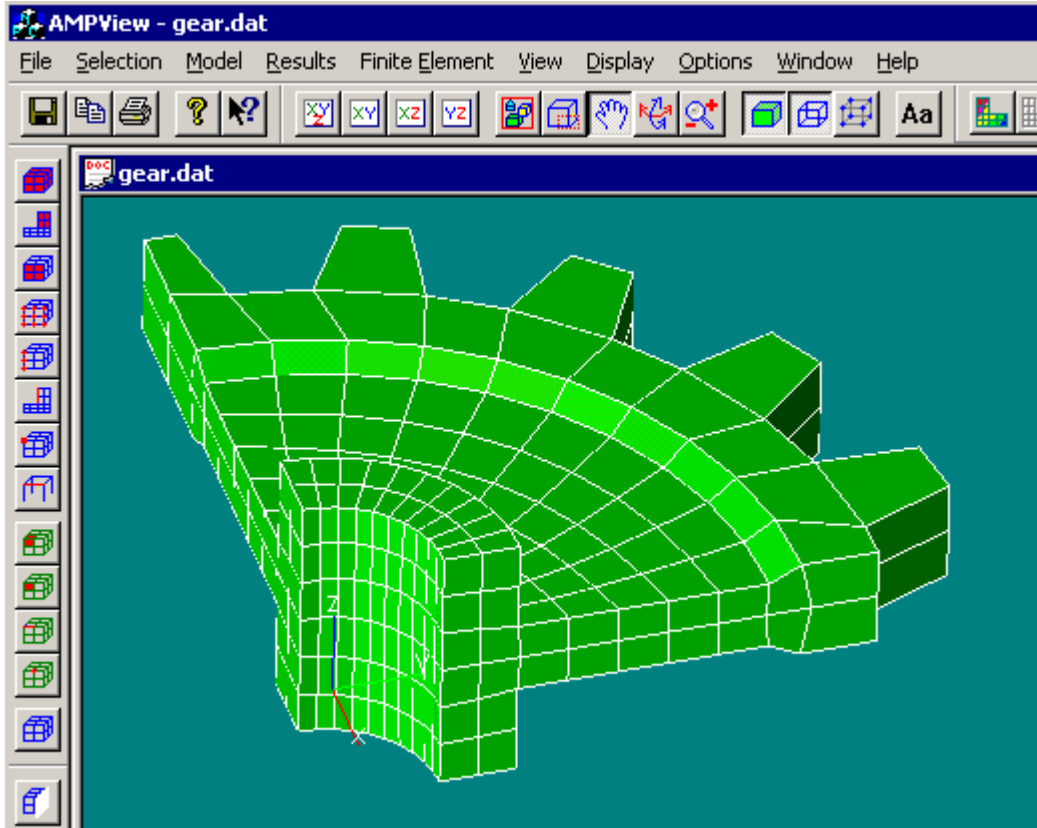
In the description of the menu options we will make use of several prefix symbols. We will use the ">" symbol to denote a sub-menu. The "||" and "&&" symbols for "or" and "and" will also be used where appropriate.

The online reference manual is provided in a context-sensitive manner such that if you use the  icon to place on the menu or item, it will provide the most relative help associated with the subject. You can also hit the function key F1 to get the immediate help content or go to the "Help" to browse the complete reference manual.

Introduction

The AMPView program is unique in the sense that it allows the user to select mesh entities in two ways. The first method is to allow selection by using the geometric entities of the solid model used to create the mesh in AMPSolid. The second method uses the finite element entities to define the model for analysis. The first method is the preferred method because it allows a full analysis model to be defined without tying it down directly to nodes and elements. As a result, the mesh for a particular solid model can be changed without requiring changes to the AMPView data file (.dat).

The AMPView features can be explained by looking at the top menu in Fig. 1.1.



Moving from left to right:

1. File. The file menu controls the import and/or export of data as well as the usual opening and printing of data. The AMPSol analyzers are run from this menu.
2. Selection. Implements the actual selection of the Geometric and/or F. E. data. The user may choose to access elements and nodes in both groups as well as individual items.
3. AMPSol. Allows the user to create data for the control of the F.E. Analysis, specify boundary conditions under cursor control, define material behavior through graphic selection and finally to create and update a material library. The data definition follows a consistent pattern. A geometric entity is first defined (selected) graphically. This is followed by appropriate selection of the type of condition that is to be attached to it. Finally the nonlinear version is controlled by increments in time. Static incremental loading is controlled in pseudo time steps. Previously defined data can be edited and reviewed graphically.
4. AMPSol RESULT. This controls the post-processing of data from a AMPSol Analysis. The user selects the type of data to be plotted. The display can be in deformed or undeformed mode including animation. Finally the user may slice the solid to view internal elements. This menu also includes an FE inquirer that is a feature for inquiring and plotting values and/or time history at a particular node.
5. Finite Element. This is the detailed control of display/grouping/create/delete and import/renumbering of finite element and nodes.
6. View. This menu controls the nitty gritty of graphic display such as zooming, background colors, lighting and etc.
7. Display. This also controls the details of the view. The type of projection and the handedness of the axis.
8. Options. This controls the F.E. details of the display. The user may specify nodes or element faces, edges and other similar details.

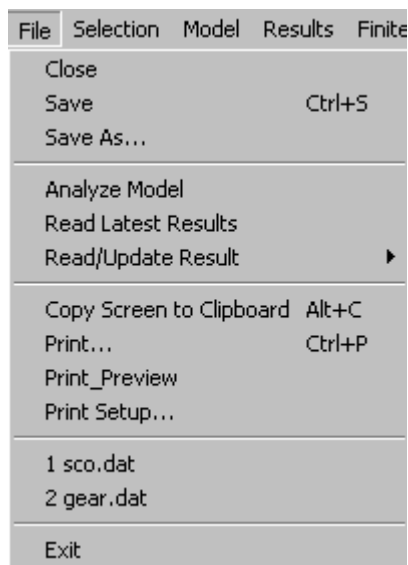
9. Windows. This gives the usual Windows controls as well as the display of toolbars and status bars.
10. Help. This provides the user with this document online as well as context sensitive help that is menu dependent.

Some of the menu items have been implemented as icons. Their purpose and function are explained below. As usual all clicking is done with the left mouse cursor. Also, as the standard of the Windows application, these icons can be dragged and placed to any user-desired position. This is usually necessary if you have a very small screen, or working on a notebook with limited screen size. To move these icon base, just drag the frame containing the icon to the desired position. If you drag these icon bars away from AMPView's border position, they become floating icon bar and can be placed on any screen position to increase the working window size. To move these icon bar back, simply drag them back to any border and "dock" them

File Menu

The File Menu

The file menu is shown below.



The current version of AMPView is designed to handle a single model, but can have multiple views for model display. This is due to the internal animation control design. If the user desires to view a different model, the current model will need to close and the new model opened, or to activate another instance of AMPView and examine the new model there.

File>Close||Save||Save As

These sub-menus follow the usual Windows functions. The mesh data is read in by opening the appropriate .dat file generated from the solid model in AMPSolid.

Analyze Model

File>Analyze Model

This starts the AMPSol F. E. analysis of the current model. Any edited files that will be used here should be saved. It is suggested after the AMPSol F.E. analysis is finish, the user examines the analysis log to check for possible analysis warning or error. It is also suggested to close the AMPSol analysis widow to conserve memory usage.

Read AMPSol Result

File>Read Latest Result

This instructs the program to read in the AMPSol results from an analysis in preparation for post processing of the AMPSol Results. For an analysis contains several time steps, it is not necessary to wait for the analysis to finish. Instead, as soon as a step result is finished from the AMPSol analysis windows, clicking this button will read the existing finished step results. If the analysis is of eigen analysis types such as modal or buckling analysis, then a separate result file can be read to for eigen mode examination.

File>Read AMPSol Result...

This menu is for precise reading of the result for either the standard output file, or to alternatively read in the modal, buckling, or eigen analysis result.

File>Copy to Clipboard

This copies the current scene image into the Windows' clipboard area as a standard clipboard meta file. The user can then insert this into other application if they accept the standard clipboard format.

File>Print||Print Preview||Print Setup

These also follow the usual Windows functions and need not be defined here.

File>Previously accessed files

The last ten files used are listed here.

File>Exit

This terminates the use of AMPView. The user is prompted to save all files created or changed.

System Storage Files

In order to achieve a seamless integration between our three programs, we have implemented a common database that is shared among all our programs. The storage of model information has however been split up into several functional files to give the user maximum flexibility in re-using previously defined data as well as minimum work in creating similar models. The following files are used.

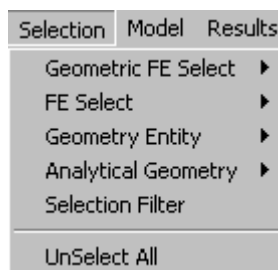
1. The *.sat file. This file stores the solid model data in ACIS format. It is used and optionally created in AMPSolid.
2. The *.flx file. This file stores the flex cell data that represents the flex cells that were created at any stage of the mesh generation. When the mesh generation process is complete in AMPSolid, the flex cell keeps the final mesh data.
3. The *.fe file. This file contains the raw mesh data. Its format is dictated by the VSMesh program, which can be tailored by each user to fit a FE program of choice.
4. The *.geo file. This file stores the solid model entity mapping to the FE mesh. This file changes with each mesh. The user is advised to give each mesh generated a unique name.
5. The *.dat file. This file is the model file. It contains the model data created in AMPView. The user is also advised to create a different file for each mesh. The previously defined data can be copied by saving the original file in the new name.

6. The *.out file. This stores the result of the FE analysis with the AMPSol Solvers. It is created in the solvers but also used in AMPView for post-processing. The results are stored at the nodes.
7. The *.rs file. This file is created for restart analysis. At the end of an analysis, the program stores the state of the program for later restart analysis.

Selection Menu

The Selection Menu

The selection menu gives two ways to pick the finite element entities. The first selects the FE entities by choosing solid model entities. The second selects FE entities directly. The first method gives the user flexibility in altering the mesh without having to change the other model data. All selections are indicated by a change in color of the entities changed. The menu is shown below.

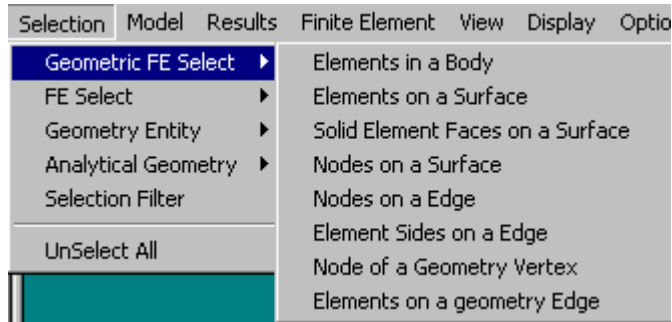


All selection implementation inside AMPView follows the Windows' selection guideline as commonly used in most Windows application. To add an item to the selection list, hold the "SHIFT" key while clicking the left mouse button on the object. To toggle the selected item in the current selection list, hold the "CTRL" (Control) key while clicking the left mouse button. In this toggle mode, an item in the selection list will be removed if it has already been selected, or it will be added if it is not in the selection list. If you click the left mouse button along, the selection buffer is first cleared, then the selected item is then added to the selection buffer.

The selection process as described is a very typical operation in all CAD/CAE programs. It is much easier to think of the selection as a "state", and there is a selection buffer associated with each state. The selection is usually stored in the database index such as material index or boundary condition geometry index. When you revisit the setting by selecting the geometry or material index, the selected items stored are immediately recalled into the current selection buffer and highlighted. You can then toggle the selection item to add/remove from the current selection buffer, and then save the modified selection. The ability to recall/store/modify the selection items is very powerful, especially when there are items in a complex model that are hidden, and you can not see it or select it at one time.

When lots of geometry faces/sides are hidden, and you want to make a very complex selection for boundary condition geometry items or material definition, you want to build up the selection items in sequence. You first select/save the first item and store the selection in the geometry/material index desired. After that, unhide/hide till the subsequence face or geometry face is shown, then go back to the stored index that you want to add to it. The instance that you select the index that you want to modify, the previous existing selection items will be recalled into the current selection buffer and highlighted. You can then add the desired selection to the selection buffer then save/store the selection list.

Selection Geometric FE Select



The Geometric FE Select is based on selecting from the original solid model entities that were used in creating the FE mesh in AMPSolid. This is the most flexible method of selection, because it associates specific solid model entities with the finite element mesh entities. When the mesh is changed either due to adaptive meshing or requested re-meshing, AMPSolid automatically associates the new finite element entities with the solid model entities. The result is that FE models created with Geometric FE selections need not be changed when the mesh changes. Using their corresponding icons as shown below can also choose the menu options



Multiple entities are selected by depressing the Shift key (for addition to the selection) or the Ctrl key (for toggle selection).

Selection>Geometric FE Select>Elements in a body
This selects all finite elements contained in a solid block.

Selection>Geometric FE Select>Elements on a Surface
This selects 2D elements.

Selection>Geometric FE Select> Solid Element Faces on a Surface.
This selects all the element faces that are contained in a solid block face. In conditions where nodal quantities are implied (such as fixed displacements); the element faces are in turn converted to all the nodes contained in the solid block face.

Selection>Geometric FE Select> Nodes on a surface
This selects all the nodes on a solid block surface.

Selection>Geometric FE Select> Nodes on a Edge
This selects all finite element nodes on a solid block edge.

Selection>Geometric FE Select> Element Sides on a Edge
This selects element sides in a 2D mesh.

Selection>Geometric FE Select> Node of a geometry Vertex
This selects a single Vertex of a solid block.

Selection>Geometric FE Select> Elements on a Geometry Edge
This selects all elements on a geometry edge. This is usually for beam/rod elements only.

Selection FE Select

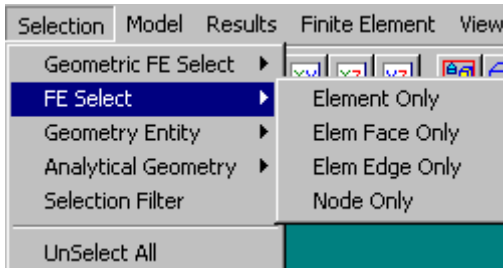


Fig 3.3 FE Select

This selection selects the Finite Element entities directly. Selection is made either with a cursor or by creation of an enclosing window. Multiple entities are selected by depressing the Shift key (for addition to the selection) or the Ctrl key (for toggle selection).

Selection>FE Select>Element Only || Elem Face Only || Elem Edge Only || Node Only

This allows the user to select according to one of the named F.E. entities. The options can also be selected from the icons shown below.



This defines the method of selection that will be used in the geometric entities such as text string, cutting plane, etc.

Finally, to remove the current selection, the user can just make a "empty" selection by clicking the mouse button on a window position away from the model, or just click the last selection toolbar icon "blue cube" (the cube without any highlight) to remove all selection items.

Selection Geometry Entity

This selection selects the associated geometry entities directly. Selection is made either with a cursor or by creation of an enclosing window. Multiple entities are selected by depressing the Shift key (for addition to the selection) or the Ctrl key (for toggle selection).

Selection>Geometry Entity>Geometry Entity Geometry Entity

This selects by using the cursor. If there is a selectable geometric entity under the cursor, it will be highlighted.

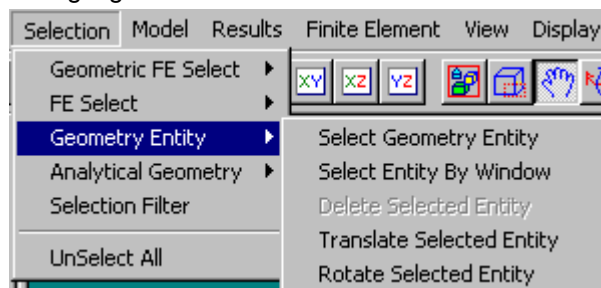


Fig. 3.4 Geometry Entity Selection mode

Selection>Geometry Entity>Select Entity by Window

The user drags a rectangle window to enclose the required entities on the screen. The selection window will show up as soon as the mouse left button is pressed and the mouse is dragged.

Selection>Geometry Entity>Delete Selected Entity

This will delete the selected geometric object.

Selection>Geometry Entity>Translate Selected Entity

This will translate the selected geometric object. The use of this function is basically to move a geometric entity such as annotation text from one position to another.

Selection>Geometry Entity>Rotate Selected Entity

This will rotate the selected geometric object about its local axes.

Selection>UnSelectAll

This unselects the current FE or geometric entities selected.

Geometric Entities conversion to FE geometry

The geometric Entities from the Solid Model can be used to select Element faces, sides and/or nodes. The FE quantities corresponding to these geometric entities (surfaces, edges and vertices) are then used to specify FE geometry for Boundary Conditions as well as material definition. The advantage of using geometric quantities is that the models can be built without regard to the finite element mesh size or distribution. That is, once the boundary conditions and material properties are defined, it is possible to re-apply a different mesh size and still use the same boundary/material definition that have been applied on the geometric entities.

This is especially important when adaptive meshing control is used since the mesh will either be refined during the analysis, or possibly refined by the user manually. In such case, the definition of the boundary condition and material properties will be applied only to the geometry face/edge/body, and not to a specific finite element mesh. This is also convenient when testing a model since a coarse mesh can be built and quickly tested. Once the model behavior is achieved, the user can just refine the finite element mesh size (as defined in the *.fe file) and still use the same model definition file (defined in *.dat file).

Analytical Geometry

Selection>Analytical Geometry

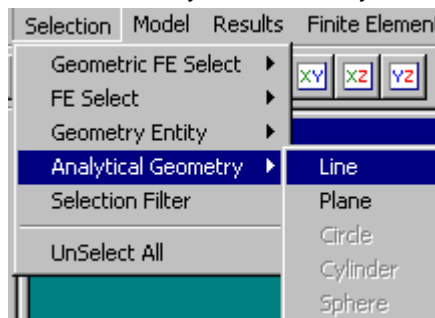


Fig. 3.5 Analytical Geometry Menu

During analysis, sometimes an analytical geometry is needed for geometrical boundary condition, e.g. contact. This session allow the user to define the analytical geometry for analysis use.

Geomtry Line

Selection>Analytical Geometry>Line

Fig. 3.6 Analytical Line Geometry

Analytical geometry line is defined by using a point and a direction vector. If the length of the line is left to be the default value zero, the line is a infinite long line. Otherwise, it is a finite line segment.

Geometry line can only be used in 2D contact analysis. When a line is used as the auxiliary geometry for contact analysis, the contacting side must be defined by either the left side or the right side of the line from the starting point of the line. The line can also move in the time/step analysis along this normal direction by using time movement scaling index. In such case, the line moves laterally, and the movement magnitude is controlled by the time scaling factor.

Geometry Plane

Selection>Analytical Geometry>Plane

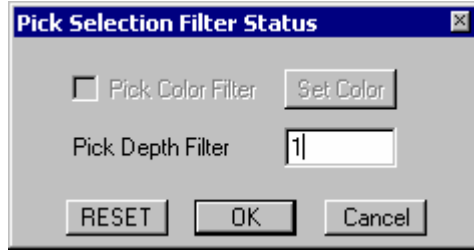
Fig. 3.6 Analytical Plane Geometry

Analytical geometry plane is defined by using a point and a plane normal vector. The plane passes through this point with a normal direction as specified.

When a plane is used as the auxiliary geometry for contact analysis, the contacting side must be defined by selecting either the side along the normal direction, or the reverse side. The plane can also move in the time/step analysis along this normal direction by using time movement scaling index. In such case, the plane moves laterally, and the movement magnitude is controlled by the time scaling factor.

Selection Filter

Selection>Selection Filter



The selection filter is usually used to select the face that is hidden under another face or faces. It is usually useful turn on the Display/UseTransparency when this option is activated to confirm the internal face selection.

The Model Menu

The Model Menu

This menu allows the user to specify all the system controls, material properties and boundary conditions for a F.E. Analysis. The top sub-menus are shown in Fig. 4.1. The AMPSol solvers share the AMPView database so their input are defined seamlessly by the AMPView menus. In developing the AMPSolid and AMPView GUIs the developers had an objective of allowing the user to perform visual analysis where an FE Analysis is carried out interactively and with no reference to element and nodal details. It is for this reason too that the user will not find a User's manual for the AMPSol Solvers. The use of the program is in fact embedded in the documentation of the Interactions in AMPView. The Appendix On AMPSol Capabilities will summarize the features available in the AMPSol Solvers.

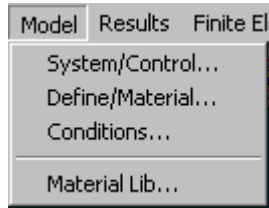


Fig. 4.1 The Model Menu

The user should note the convention for creating the model data here. The options below the top menu are called sheets. Thus we have the System Sheet, the Define Sheet and the Conditions Sheet. The options below the Sheets are called Pages. If there are finite element entities to be selected, we usually start the definition by selecting a geometric F.E. entity from the Geometry Page. AMPView turns this entity into a list of elements (elements, faces or edges) or nodes. The appropriate condition or material is then selected from a Property page and attached to the geometric entity. Both the geometric selection and the specified conditions can be reviewed and/or edited.

Model/System Menu

The Model/System/Controls Menu

This menu enables the overall F.E. Analysis Controls to be set.

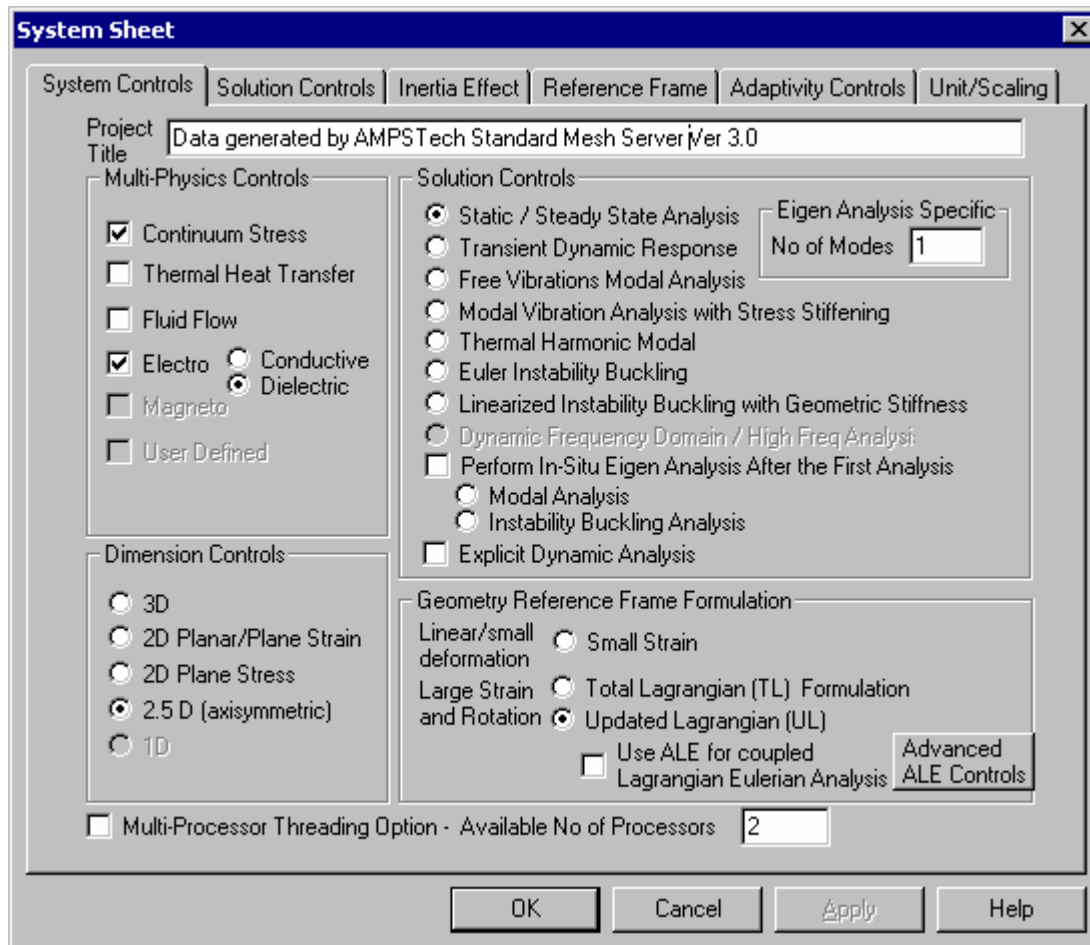


Fig. 4.2 The Model>System >System Control Page.

Model>System>Controls>Title

The user starts by giving a title for the analysis.

Model>System>Controls>Multi-Physics>Continuum Stress||Thermal Temperature||Fluid Flow||Electro||Magneto||User Defined

The checked boxes here allow a combination of the selected Physics. The grayed check boxes are due to either unavailable license or unreleased options. If the electro-field is activated, it is necessary to specify whether the analysis is based on a conductive potential-current analysis or a dielectric potential-electric field Intensity analysis.

Model>System>Controls>Solution Controls

The user is allowed to select from one of the options in this sub-menu:

Static/Steady state

This specifies a static or a steady state solution is desired.

Transient Dynamic Response

This specifies an analysis by step-by-step integration in time.

Followings are the available eigen analysis types:

Free Vibrations Modal Analysis

This performs a standard modal analysis to extract the desired modes of vibrations. The generalized eigen problem is $\mathbf{K} - \lambda \mathbf{M} = 0$, where \mathbf{K} is the system stiffness and \mathbf{M} is the consistent mass matrices. The natural frequency ω is computed from solution $\omega = \text{sqrt}(\lambda)$, and will be recorded in the solution file (with filename extension ".eig").

Modal Vibrations Analysis with 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.

Thermal Harmonic Modal

This calculates the thermal harmonic modes of temperature patterns. The generalized eigen problem is $\mathbf{K}_c - \lambda \mathbf{M}_t = 0$, where \mathbf{K}_c is the system conductivity matrix and \mathbf{M}_t is the consistent thermal mass matrix formed by scaling the consistent thermal mass matrix by the thermal heat capacity

Euler Instability Buckling

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

Linearized Instability Buckling with Geometric Stiffness

This is a linearized predicting of the nonlinear buckling load calculation. The initial geometric stiffness \mathbf{K}_g is computed based on the given loading, then the generalized eigen problem of $(\mathbf{K} + \mathbf{K}_g) - \lambda (\mathbf{K}_s + \mathbf{K}_s') = 0$ is solved. \mathbf{K}_s' is the 2nd order stress stiffness matrix based on the geometric deformation.

In-Situ Eigen Analysis

After either a successful static or dynamic analysis, the user can specify to perform either a modal or an instability buckling analysis in the same analysis. Although it is possible to restart the problem and then specify an eigen analysis, this is a convenient method of performing "in-situ" eigen analysis. In such way, the "in-situ" stiffness and mass matrices (or the stiffness and the stress stiffness matrices) represent the state at the end of the analysis, and the eigen solution give the "in-situ" prediction of the eigen problem.

For the in-situ modal analysis, the generalized eigen problem is $\mathbf{K}_t - \lambda \mathbf{M} = 0$, where \mathbf{K}_t is the system tangent stiffness matrix. For the in-situ instability buckling analysis, the generalized eigen problem is $\mathbf{K}_t - \lambda \mathbf{K}_s = 0$, where \mathbf{K}_s is the stress stiffness matrix based on the finished state.

Dynamic Frequency Response/High Frequency Analysis

This specifies a frequency domain steady state solution is desired.

Explicit Dynamic Analysis

Use Explicit Central Difference Method as the transient dynamic analysis method rather than the default implicit method.

Model>System>Controls>Solution Controls>Eigenvalue Analysis Specific/No of Modes.

The user specifies the number of eigen modes to be computed. All eigen analysis will have their eigen modes and eigen values result recorded in the eigen solution file (with filename extension .eig).

Model>System>Controls>Dimension>3D || 2D Plane Strain || 2D plane Stress || 2 ½ D axisymmetric.

This specifies the geometric dimension of the analysis. Note that for 2D analysis, only the first two coordinates of the nodal position data will be used. Effectively, the model should only be construct on XY plane if 2D analysis is desired. In the axisymmetric analysis, the first coordinate is the radial dimension.

Model>System>Controls>Geometry Reference Frame Formulation>Small Strain || Total Lagrangian Formulation || Updated Lagrangian Formulation

If small strain is used, the program will use only the small deformation linear strains in the stress calculations. For large deformation and rotation, either Total Lagrangian (TL) or Updated Lagrangian (UL) can be used. The difference between these two large strain calculation modes is simply the difference of the reference frame. The TL always uses the original state for the frame of reference, and the UL uses the most recent state of position 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, but the converged results will be almost similar since the final result is expressed in the engineering Cauchy stresses and strains.

Model>System>Controls>use ALE for coupled Lagrangian Eulerian Analysis

When the fluid flow region is used along with the Lagrangian solid element, If small strain formulation is used, the coupling between the fluid governing Navier-Stokes equations and the solid continuum equations are automatically coupled. However, if large deformation analysis option is activated, the Lagrangian solid region will penetrate into the Eulerian flow region, especially the solid elements deform very large. In such case, it is necessary to activate the Arbitrary Lagrangian-Eulerian formulation for the fluid region. For the time being, only the updated Lagrangian formulation is supported when ALE method is activated. The processor will automatically relocate the Eulerian fluid node based on the Lagrangian solid elements' nodal movement, and will try to smooth out the distortion as much as possible. In ALE analysis, the Eulerian variables' results are spatially mapped into the newly relocated node position, and all associated nodal and element variables are carried forward for the subsequent analysis step. Optional controls for the ALE method are the step interval for the ALE processing, and the meshing adjustment and smoothing controls.

Model>System>Controls>Enable Multi-CPU Treading

This will take advantage of multiple CPU's if they are present. AMPSol FE processor is capable of initiating multithread processing whenever possible during the analysis, and the use of the threading will take advantage of the SMP feature of operation systems NT and Windows 2000 that support multiple CPU's.

Model>System>Controls>Available No of Processors

For multiple-CPU machine, this will show the current count of CPU's. By specifying a number other than the default value, the user can limit the no of CPU's to be used in the multi-thread tasks.

This section gathers the possible execution options that affect the whole analysis.

Execution Options	Function
Plane Stress	Specify plane stress geometry
Axisymmetric	Specify axisymmetric geometry
Multithread	N threads. This will control the processor usage.
Acceleration	Globally applied acceleration field, subject to time scale if it is specified.
Solver	Wave Frontal, JPCG Iterative, direct in-core
Strain Formulation	Small Strain, Total Lagrangian, updated Lagrangian

The Model/System/Solution Iteration Control

This specifies the iteration procedure to be adopted.

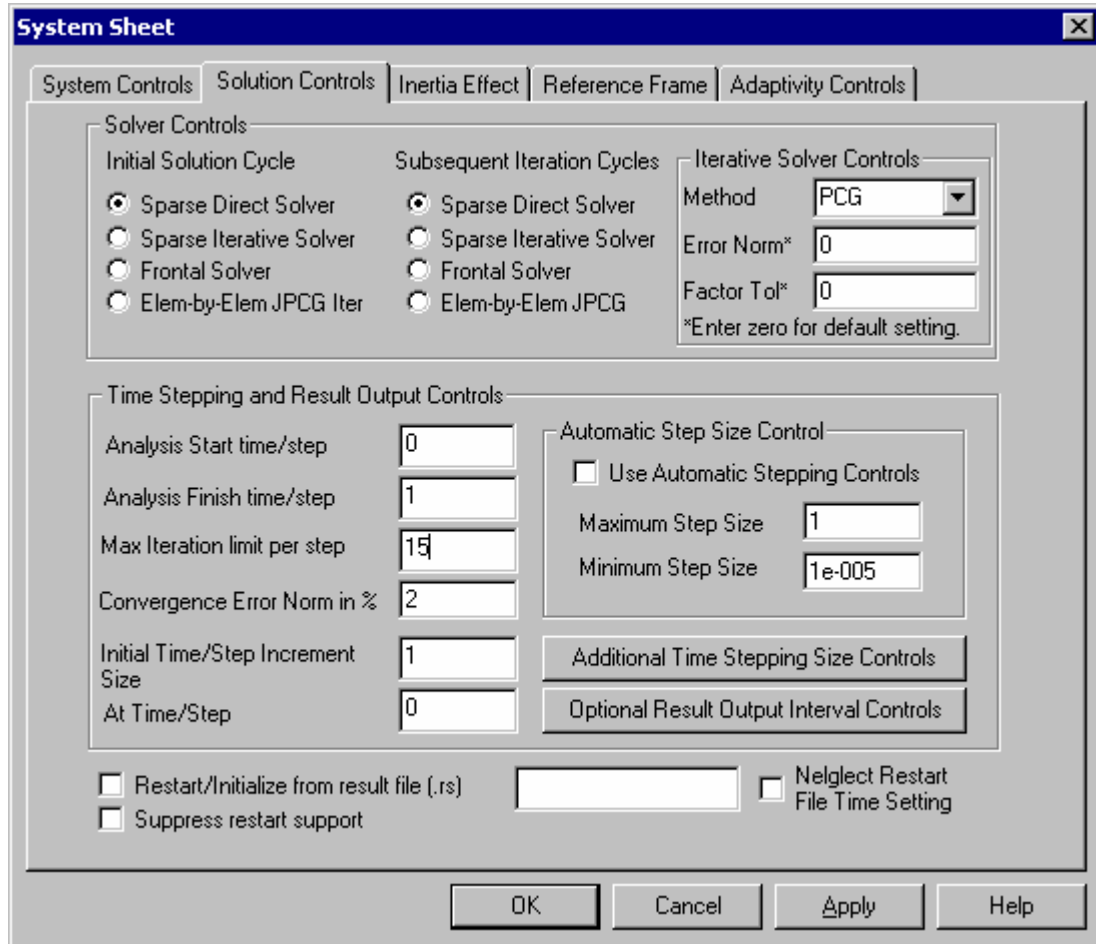


Fig. 4.4 The Model>System>Iteration Controls page

Model>System>Iteration Control>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 is the default selection as it 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 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.

Model>System>Iteration Control>Iterative Solver Controls.

Currently, following methods are available for sparse iterative solver:

PCG: Preconditioned Conjugate Gradient method

BiCGSTAB: BiConjugate Gradients Stabilized method

MLBiCGSTAB: a BiCGSTAB variant based on Multiple Lanczos Starting Vectors

TFQMR: Transpose-Free QMR (Quasi-Minimal Residual)

BGMRES: Block General Minimal Residual

These sparse iterative methods use an incomplete LU factorization matrix as the preconditioner for the iteration method. Since all iterative solver techniques are sensitive to the condition of the equation being solved, it should always be used with small enough error control to ensure the solution is correct. The default error_norm for the iterative solver is 1.e-6, and the incomplete LU factoring skip tolerance is automatically determined based on the ratio of the minimum to the maximum diagonal of the equations. This setting is found to be a reliable technique from our empirical testing, and this default setting handles well for most problems including shells, high order p element and general problems. In general, this error norm can be lowered to around 1.e-4 for most problems to speed up the solution time. For most problems, this factor_tol is generally found to in the range of 1e-2 to 1e-8. If necessary, the user can overwrite the automatic setting so the tolerance will be computed based on the value of factor_tol times the maximum diagonal term of the equations. Relaxing these tolerances may speed up the solution, but the iterative method may not converge and could cause the program to stop after reaching its maximum iteration limits.

Generally, PCG method performs well in most cases, but when it has erratic convergence behavior either due to poor preconditioning treatment or ill-conditioned equations due to the multi-physics properties, other methods have proven to be more stable. More detailed reference about these methods can be found in general numerical methods literature.

Model>System>Solution Control>Time Stepping/Pseudo Time Controls

For all analyses, the user specifies a time step size, the analysis start time and the finish time to control the solution. In the steady state analysis or static analysis, these time controls are pseudo time steps in the sense that they are used to control how the specific boundary conditions or loads are applied. For dynamic or transient analysis, the time controls should be the real analysis start/stop/increment that control the analysis. The program, by default, assumes the analysis is static or steady-state and fills in the default values for the user at the beginning.

Model>System>Solution Control>Time Stepping/Pseudo Time Controls>Analysis Start/Finish Time

The user specifies the analysis start and finish time here. For restart analysis, the start time will come from the restart file, and the start time control will automatically change to the values specified in the restart file.

Model>System>Solution Control>Time Stepping/Pseudo Time Controls>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 under the specified values

Model>System>Solution Control>Time Stepping/Pseudo Time Controls>Initial Time Step, At Time, Optional Step Size Controls

The solution starts from the analysis start time, and advances by the specified time step size as specified using this step size control. Other than the Initial Step Size, 3 additional pairs of Step Size n/At Time n can be use to specify the step size during the analysis. The actual step size used in the analysis is then interpolated from these controls. Note that the during the analysis, results are available at time/steps corresponding to the step size control here too.

Model>System>Iteration Control>Time Stepping/Pseudo Time Controls>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 till a successful analysis can be established. The program then advanced till these refined time steps reach the end of original failed step. At that time, the next time step will start again and continue. The user can control the maximum/minimum automatic step size that this automatic stepping control is allowed to take.

Model>System>Iteration Control>Initial Time/step Increment Size

This is the initial time/step size control. When the analysis starts, it will use the specified increment size as the initial size. Additional stepping size after the initial time can be set by pressing the "Additional Time Stepping Size Controls" button.

Model>System>Iteration Control>Optional Result Output Interval Controls

Normally, the processor will write a result data set for each successful time/step increment for AMPView to display the results. Sometimes, when the time increment is very small as in the case of transient dynamic analysis, it is very desirable not to perform the result output per step increment, but only to do so after couple more steps. This button provides a way to specify the output interval controls. In such case, the processor will only write out the results for AMPView only after the time/step increment reaches the specified count.

Model>System>Solution Control>Restart/Initialize from result file / Suppress restart support/Neglect Restart File Time Setting

By checking this box, the user can specify a restart analysis from a previous finished analysis, if the restart file *.rs is available. The user can also suppress the writing of the restart information if the available disk space is limited, or the restart ability is not desired. If the "Neglect Restart File Time Setting" is set, when the restart file is read, the processor will ignore the analysis time stored in the restart file.

The Model/System/Inertia Effects

The Inertia effects are specified in the following Menu

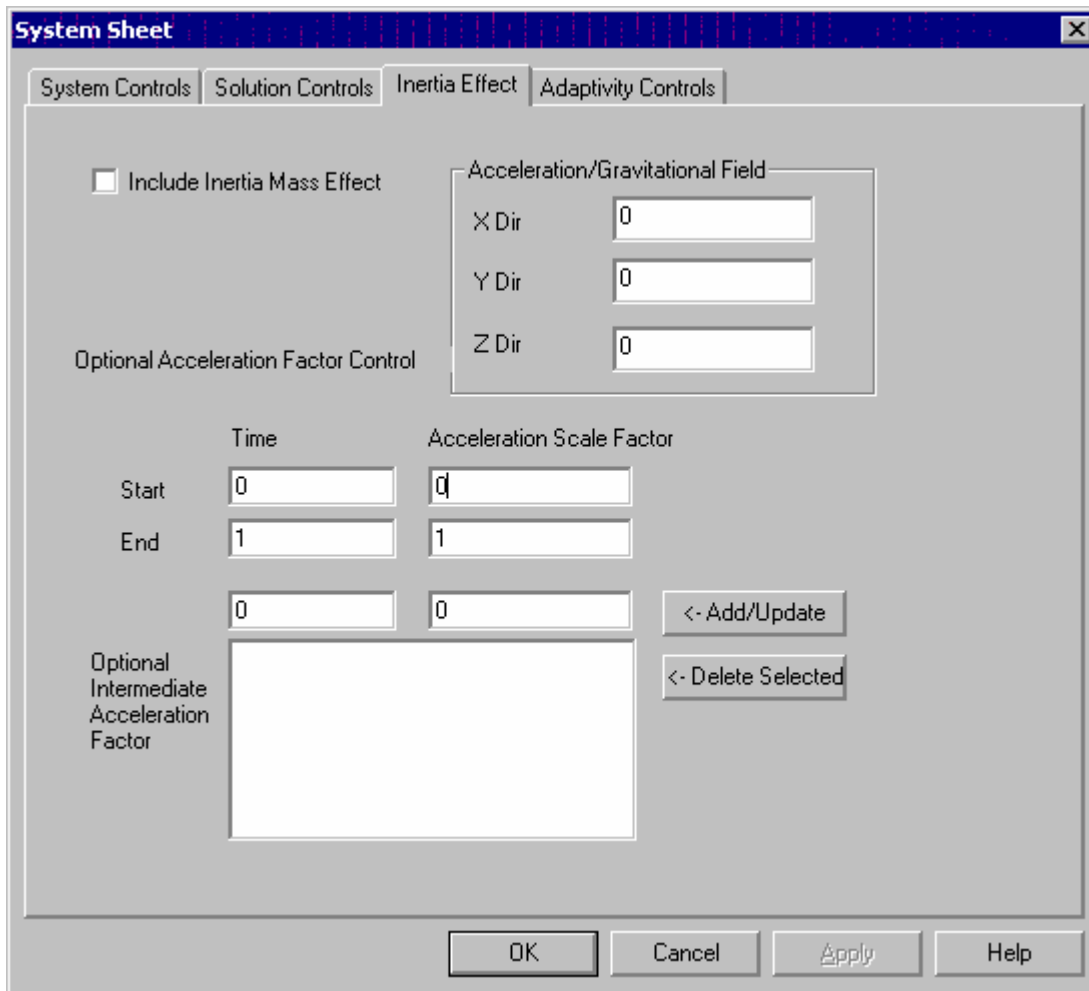


Fig. 4.5 The Model>System>Inertia Effect page.

The user specifies the inclusion of the Mass Effects by means of a checked box.

If the inertia effect is included, there is the provision to add an Acceleration/Gravitational Field. Defining the starting and ending times of such a scaling may scale the applied accelerations. Provisions are made for additional time/step to scale the accelerations, and there is no limit to the number of the intermediate scaling factor sets.

The Model/System/Reference Frame Controls

The Reference Frame Controls in the following Menu

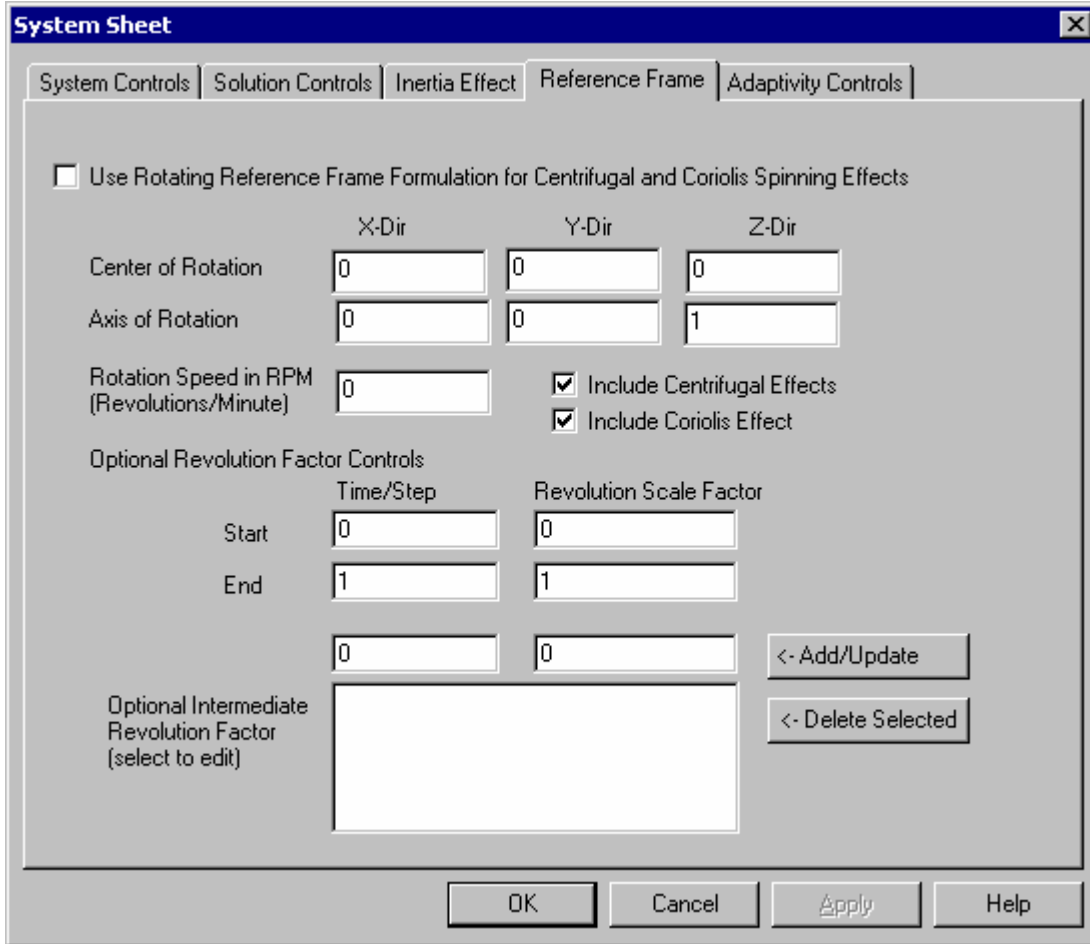


Fig. 4.5 The Model>System>Reference Frame.

When the model is spinning about a certain axis, and the interest of the analysis is in the behavior relative to this rotational frame, it is easier to solve the problem by attach the frame of reference to the model. The Rotation Reference method is one of the most commonly used methods for this type of spinning motion analysis.

When the Rotational Reference Frame is used, the boundary conditions and the computed results are referred to the spinning reference frame with the specified rotational speed. In the rotational reference frame formulation, the acceleration of the continuum/fluid is:

$$\hat{\partial}v / \hat{\partial}t + 2\Omega \times v + \Omega \times (\Omega \times x) + d\Omega / dt \times x$$

The symbols x and v are the position and velocity vectors with respect to this spinning reference frame, and Ω is the rotation axis vector. Note that when large deformation effect is included, the position vector refers to the updated position. The term $2\Omega \times v$ is the Coriolis acceleration effect, and $\Omega \times (\Omega \times x)$ is the centrifugal acceleration effect. Since the reframe of rotation is fixed with a given rotation frequency, the last term vanishes in the steady state solution.

The data needed for this Rotation Reference Frame formulation are:

Center of Rotation: this is the origin of the rotation frame.

Axis of Rotation: this refers to the vector about which the reference frame is spinning.

Rotation Speed in RPM: this is the scalar number of the rotation speeds in revolutions per minute.

Include Centrifugal Effect: this is the check box for the additional control to include/exclude the centrifugal effect.

Include Coriolis Effect: this is the check box for the additional control to include/exclude the Coriolis effect.

Finally, a scaling factor is allowed to defining the starting and ending times of the rotation speed effect. Provisions are made for additional time/step to scale the accelerations, and there is no limit to the number of the intermediate scaling factor sets.

The Model/System/Adaptive Controls

The adaptive controls are specified in the following spreadsheet page

System Sheet

System Controls | Solution Controls | Inertia Effect | **Adaptivity Controls**

Use h-Adaptive

Automatic h_Division Based on Error Estimate

Uniform h_Division Based on the Highest Error

Uniform h_Division as Specified

Use p-Adaptive

Use Specified p_Order p_Order/Increment

Increase p_Order as Specified

Use Highest Automatic p_Order

Refinement Target Error Norm

Maximum Mesh Refinement Allowed

Step Interval for New Refinement

Restart Analysis After Refinement (single step or linear problem)

Optimize Mesh After Refinement

The Model>System>Adaptive Controls>Use h-Adaptive, Use p-Adaptive
Active the h-Adaptive and/or p-Adaptive refinement. The default setting will be used.

Possible setting for h_Adaptive are:

The Model>System>Adaptive Controls>Automatic h_Division Based on Error Estimate

When this default h-Adaptive option is selected, each element will be divided according to the error estimated computed, could be different in each spatial direction. Currently, up to 5 division in the each direction is supported. If an element is split, the neighboring element will also be divided to match the mesh conforming criterion. Note that if there is a triangular face, each side can only split into 2 division.

The Model>System>Adaptive Controls>Uniform h_Division Based on the Highest Error

When this h-Adaptive option is selected, all elements are divided based on the highest element error. This will generate large amount of elements.

The Model>System>Adaptive Controls>Uniform h_Division as Specified

When this h-Adaptive option is selected, all elements are divided based on the specified division (in each element direction). For example, specifying 2 will split a 2D QUAD4 element into 4 QUAD4 elements.

Possible setting for p_Adaptive are:

The Model>System>Adaptive Controls>Use Specified p_Order

When this default p-Adaptive option is selected, each element will be transform upward or downward into a 2nd order Lagrangian family elements of QUAD9, HEXA27, TRI6, SHELL6 accordingly. This default setting is used since they produce the fastest error convergence with the least amount of problem size based on empirical tests. Currently, the highest p_order supported is three since beyond that further mesh refinement is usually more desirable rather than polynomial order increase. If triangular-faced elements are detected, the maximum p_Order can be used for such face is two on such elements.

The Model>System>Adaptive Controls>Increase p_Order as Specified

When this p-Adaptive option is selected, the element family is increased by the p_Order amount specified. For example, when p_Order is one, QUAD4 will turn into QUAD9, and HEXA8 will become HEXA27.

The Model>System>Adaptive Controls> Highest Automatic p_Order

When this h-Adaptive option is selected, all elements are transformed into the highest polynomial order (three for the current version, unless triangular-face elements are detected. In such case, the highest possible order is two).

The Model>System>Adaptive Controls> Refinement Target Error Norm

The mesh error of the analysis is based on the assumption that the error norm can be computed from the approximation

$$\|e\| \propto h^{**p}$$

Where $\|e\|$ is the computed error_norm from AMPSol based on the energy error estimate, with h being the mesh size, and p being the element's polynomial order. The target h and p can be computed based on the specified error_norm $\|e\|$ desired.

The Model>System>Adaptive Controls> Step Interval for New Refinement

This specifies the step interval when a refinement should take place for a time stepping analysis. In such case, the current step result is mapped into the new mesh, and the analysis starts from the refined mesh based on the interpolated results generated from the current step. Since mesh refinement could generate large amount of elements and nodes, care should be

taken to limit the maximum amount of mesh refinement step allowed (see Maximum Mesh Refinement Allowed).

During such time/step analysis, new models will be generated when a refinement interval is encountered. If the original model name is "old_model", and the count of the refinement is N, the new model generated will be stored as "old_model_hpN" (for both h-/p-Adaptive), or "old_model_hN" (for h-Adaptive only), or "old_model_pN" (for p-Adaptive only).

The Model>System>Adaptive Controls> Maximum Mesh Refinement Allowed

This specifies the maximum mesh refinement process can take place during an analysis. After reaching this limit number, the mesh refinement process will be skipped.

The Model>System>Adaptive Controls>Restart Analysis after Refinement (single step or linear problem)

Instruct AMPSol to restart the analysis using the refined mesh. This should only be used for a single step linear problem. Also, this option can not be used in conjunction with a restart analysis (see Model>System>Solution Control) since the older result file will not contain the refined mesh result.

The Model>System>Adaptive Controls> Optimize Mesh After Refinement

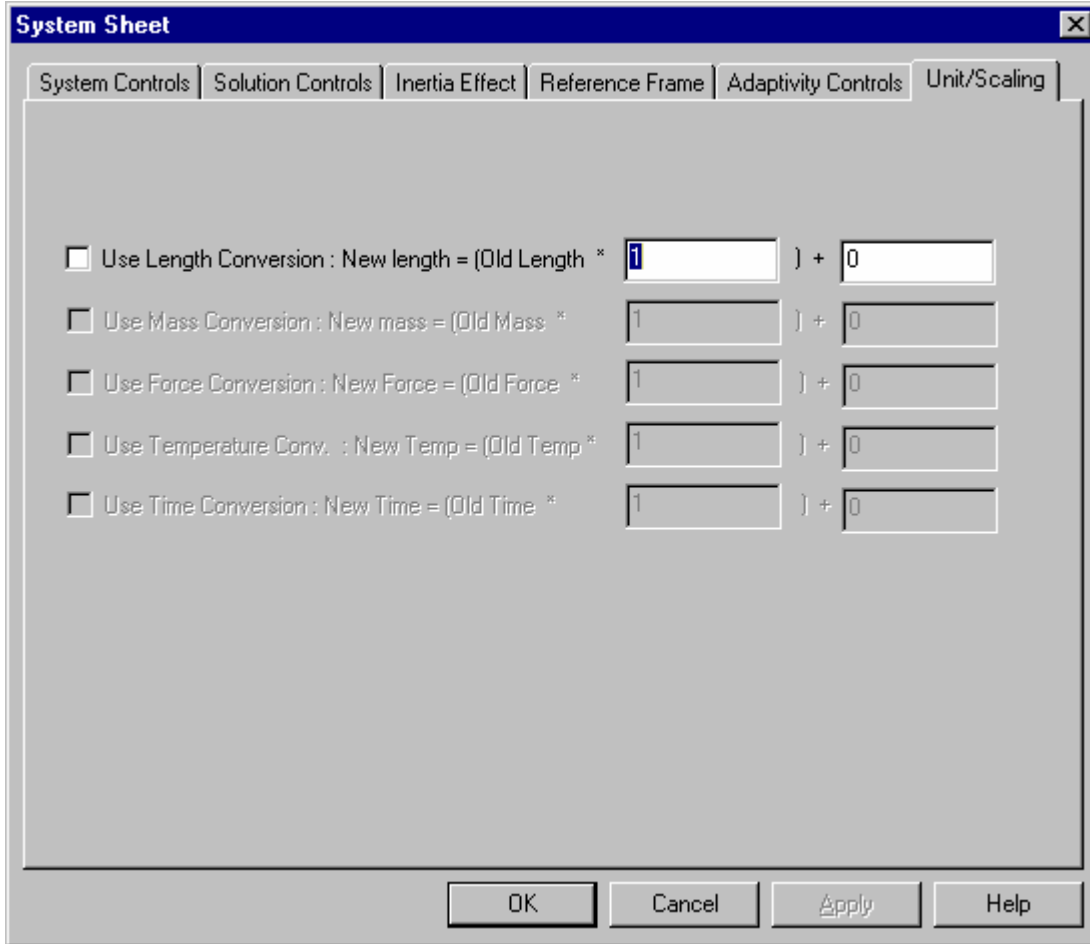
This control specifies a mesh optimization should be immediately performed on the refined mesh to minimize the runtime memory requirement and speed.

The Model>System>Adaptive Controls> Refine Mesh Base on Current Error Norm Now

If an analysis result is available, a new refined mesh can be generated based on the analysis error term. The new mesh will be created based on the current adaptive controls, and a new model will be generated. If the original model name is "old_model", the new model generated will be stored as "old_model_hp0" (for both h-/p-Adaptive), or "old_model_h0" (for h-Adaptive only), or "old_model_p0" (for p-Adaptive only).

The Model/System/Unit Scaling Controls

The adaptive controls are specified in the following spreadsheet page



The Model>System>Unit/Scaling

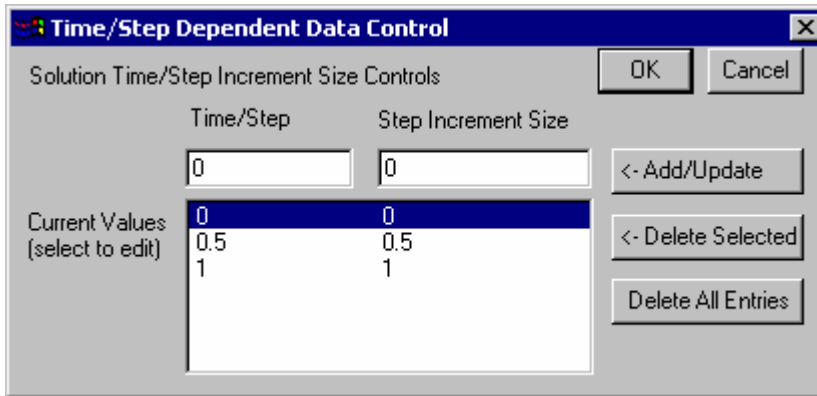
This is the optional menu for unit scaling. Since AMPS product is designed to allow the user to choose their own units specification, there is no additional unit conversion used in all calculations. However, to facilitate the ease of using data from different CAD/CAE system, the unit scaling options are provided to convert the input data into the appropriate unit in computing. The conversion is process as:

$$\text{New data} = (\text{input data} * \text{scaling}) + \text{offset}$$

Currently, only length conversion is active as of AMPS version 3.0.

Time/Step Dependent Data Control

This is a general data entry table for time/step dependent data control. By entering the specific time/step value with the desired data value and pressing the "Add/Update" button, the data set will be recorded and sorted in an increasing order. To modify the data, move the mouse cursor into the data display area and click on the desired data to modify. The entry's values will be retrieved into the current editing entry area for modifications. To delete an undesirable item, simple choose the item and press the "Delete Selected" button. The "Delete All Entries" button will clear all data entries.



Model/Define Menu

The Model Define Properties Menu

The Model> Define>Element Properties Menu is shown below

To use this define operation, the user starts by selecting the geometry to be defined, then create/change the appropriate material index and name, then assign the element behaviors by selecting the proper "Constitutive Behavior" check boxes. Additional pages will appear when new Constitutive Behavior is used for entering the actual material data.

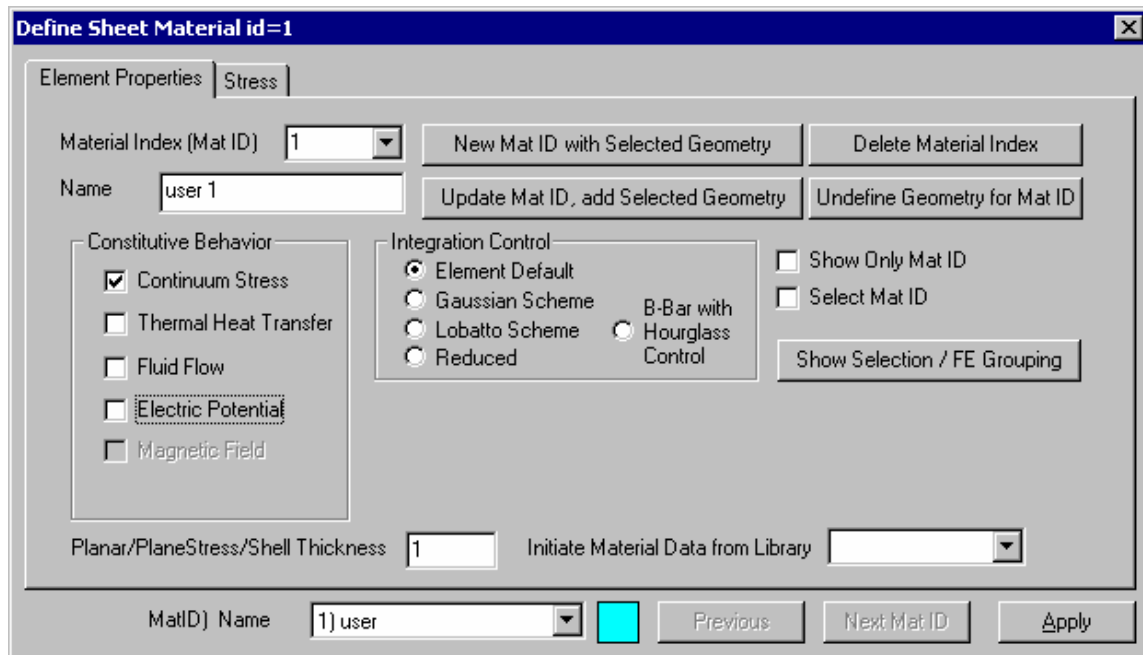


Fig. 4.6 The Model>Define>Element Properties page

When a material is assigned to a selected geometry region, the region changes its color to indicate that there is a material property assigned to the elements associated with the geometry. At the bottom of this Define Sheet, there is a Mat ID/Name pull-down list for convenient direct access to the already defined material properties with the associated material region color. There are also "Previous" and "Next Mat ID" buttons to quickly navigate through different material indexes.

When a geometry region has no material association, a grey color is assigned to the geometry to indicate that it needs to be defined. The region with material properties already defined will show up with color matching the current Mat ID to easy identification.

Model>Define>Element Properties>Material Index (Mat ID)

This index is the number assigned to the material defined under Element Behavior. This index is assigned when the user use the "New Mat ID with Selected Geometry" button, or can be changed by directly entering an index number, or by pull down the dropdown list, if available.

Model>Define>Element Properties>Name

This is the user defined material name for the selected geometry region. By default, it is assigned as "User n" where n corresponds to the Mat ID, and the user can change this name into a more meaningful way to identify the material region.

Model>Define>Element Properties>New Mat ID with Selected Geometry

Use this button to create a new Mat ID and apply the new material index/name to the selected geometry regions. The user will generally select the geometry (usually the BodyElement or the FaceElement geometry) and click this button. If the selection buffer is empty, a new material is created with no associated geometry.

Model>Define>Element Properties>Update Mat ID, add Selected Geometry

Use this button to add the selected geometry to the current Mat ID. If the Mat ID specified does not exist, it will be created with the Name as specified.

Model>Define>Element Properties>Delete Material Index

This button can be used to delete unused material type. If there are regions using this material Index, their material index association will become undefined.

Model>Define>Element Properties>Undefine Geometry for Mat ID

Press this button to make all existing geometry items associated with the current material index Mat ID to become undefined in material definition. Their default material color will become a grey undefined color.

Model>Define>Element Properties>Show Only Mat ID / Select Mat ID

These two check boxes are mainly for material association review. If "Show Only Mat ID" is selected, only the geometry associated with the current Mat ID will be displayed. The "Select Mat ID", when selected, will automatically select the geometry associated with the current Mat ID so it will be shown with a highlight color. If the global "Display/Zoom to Selection" function is active, the selected geometry will be properly placed in a zoom mode that fits all elements in the material region.

Model>Define>Element Properties> Show Selection / FE Grouping

The will bring up the "FE Group/Display" dialog box so the user can examine the selected elements, geometry items, or groups. This is usually for confirming the visual geometry selection items with the underlying element or geometry numerical information. For detailed information, please refer to the Finite Element/FE Group/Display control.

Model>Define>Element Properties>Constitutive Behavior>Cointinuum Stress || Thermal Heat Transfer || Fluid Flow || Electric Potential || Magnetic Field

These check boxes are for selecting the type of constitutive behaviors. Multiple element behaviors can be selected if the elements are to be used for multi-physic simulations. The grayed out items are either the existing license does not allow it, or not implemented. When a material physics is activated, the spreadsheet will add more pages to allow the input of the material data.

Model>Define>Element Properties>Integration Control>Element Default || Gaussian Scheme || Lobatto Scheme || Reduced Integration || B-Bar with Hourglass Control

This selects the numerical integration scheme to be used. Using the default scheme is recommended since the material group could contain different elements. The available methods are the Gaussian, the Lobatto, the Reduced, and the B-Bar with hourglass control schemes. These different integration schemes generally produce the same results when the element meshes are refined. When the mesh is coarse or when extremely nonlinear geometry/material properties exist, some schemes could improve the accuracy. For different type of elements, the default integration is selected for best performance, e.g. B-Bar for low order HEX8 or QUAD4 elements, Gaussian for TETRA4, TETRA10 TRI3, and TRI6 elements.

Model>Define>Element Properties>Planar/Plane Stress/Shell Thickness

Specify the planar type model, plane stress or shell element thickness. If not specified and the planar, plane stress or shell model is used, it is defaulted to be a unit thickness, the same as the default thickness values of 2D problems.

Model>Define> Element Properties >Initiate Material Data from Library

If the material data had previously been defined in the material Lib. Simply selecting from the drop down list may specify the material. In such case, the field associated with the material library data will be initialized. Note that the material library may not contain all information needed to initialize the needed data for analysis.

Model>Define> Element Properties >Apply

If the material data are changed, this button will become active. Pressing it will save the changed data back to system memory. To permanently save the changed model, the user still need to use File/Save command to save it back to the computer storage.

The Model Define Element Behavior - Stress Menu

This menu spreadsheet is shown below. This dialog page is mainly for assign the associated material properties to the material index type.

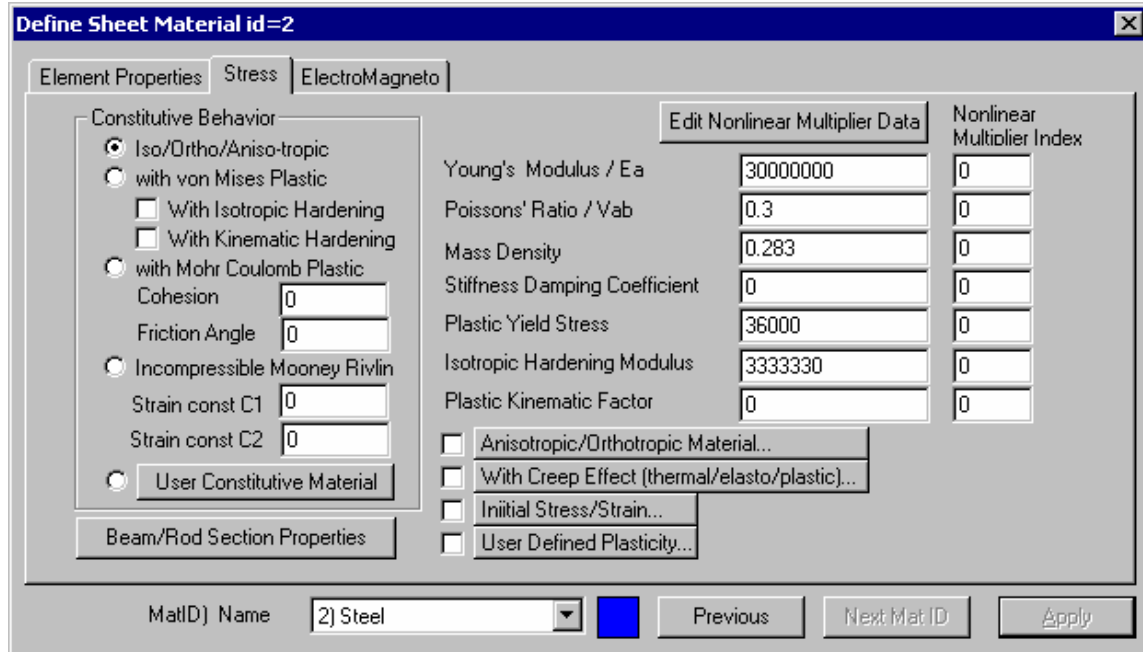


Fig. 4.7 The Model>Define>Element Behavior page

Model>Define>Element Behavior>Material Property Index

This index is the number assigned to the material index. This index can be changed by directly entering an index number, or by pull down the dropdown list. If available, and select one from the list.

Model>Define>Element Behavior>Add/Apply/Update/Delete Material Index

The desired material index is selected from the dropdown list or by entering it at the editable index space. After entering all associated information, clicking the Apply/Update button will update the material information of the material index selected. If "add" option is used, a new index is created by adding one to the highest existing material index, and then the data entered are then saved. The "delete" button can be used to delete unused material index type, and the associated material data.

Model>Define>Element Behavior>Constitutive Behavior>Isotropic Elastic

This specifies an isotropic elastic property.

Model>Define>Element Behavior>Constitutive Behavior>von Mises Plastic

This specifies the metal Von Mises plastic behavior. The isotropic hardening and/or kinematic hardening options can be used together with this plastic material model.

Model>Define>Element Behavior>Constitutive Behavior>Platic Mohr Coulomb

This specifies the cohesive material with cohesion and the friction angle. The friction angle should be between 0 and 90 degrees.

Model>Define>Element Behavior>Constitutive Behavior>Incompressible Mooney Rivlin

This specifies the hyperelastic Mooney Rivlin incompressible material model. The C1 and C2 coefficients define the strain energy equation

$$W = C1 (I_1-3) + C2 (I_2-3)$$

Where I_1 is the first principal strain invariant, and I_2 is the second principal strain invariant. When $C2=0$, the model degenerated into a neo-Hookean material. More information about this material model can be found in "Finite Element in Nonlinear Continua", J. T. Oden, or other popular references.

Model>Define>Element Behavior>Constitutive Behavior>User Constitutive Material Model

This specifies the material model is defined in the user-supplied material model routine. Additional material property data can be stored and passed into the user-defined routine by clicking the button "User Constitutive Material". When this option is used, the user is responsible to carefully define the constitutive material model following the model development guideline. Details of the user-defined material model can be found in the AMPS User Application Interface Reference Manual.

Model>Define>ElementBehavior>Young's Module, Poisson's Ratio, Linear Expansion Coeff., Mass density.

Specify the isotropic material's stress property, the mass property is necessary only if inertia effect is included, or a modal or dynamic response is active.

Model>Define>ElementBehavior>Mass Damping Coefficient

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.

Model>Define>ElementBehavior>Plastic Yield Stress

Specify the initial yield stress

Model>Define>ElementBehavior>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

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

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

Model>Define>ElementBehavior>Plastic Hardening Factor

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

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

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.

Model>Define>Element Behavior>Nonlinear Multiplier Index

This index allows the control of the material data to be dependent on the available system variables such as time or stress or strain, etc. For further information see the nonlinear property multiplier dialog session.

Model>Define>ElementBehavior>Beam/Rod Section Properties

This button will show a control menu to set the cross section property specification menu. The data are only necessary if the material properties are dealing with either beam or rod/truss elements.

Model>Define>ElementBehavior>Anisotropic/Orthotropic Material

Specify the anisotropic material's property is active. To specify the material's anisotropy, click the "Setting" button to define the material data.

Model>Define>ElementBehavior>With Creep Effect (thermal/elasto/plastic)

When this option is checked, the thermal-elasto-plastic creep strain is computed based on the power creep law as specified in the creep material data. To specify the power creep law data, click the "Setting" button.

Model>Define>ElementBehavior>Initial Stress/Strain

When this option is checked, the default initial stress and strain can be set for each element. This is usually used for problems that require initial pre-stress or need the strain history for specific material model. For details, please refer to the User Defined Initial Stress and Strain section.

Model>Define>ElementBehavior>User Defined Plasticity

The USRAPP user interface routine can be used to defined any plasticity material model if the provided material library is not applicable. In such case, by checking this box, the user can enter additional material parameters to be sent to the USRAPP API interface. For detailed information about user interface functions, please refer to the "AMPS User Application Interface Reference Manual."

The Model Define Element Behavior - Thermal/FLow Menu

Property	Value	Nonlinear Multiplier Index
Thermal Conductivity	8.159e-006	0
Specific Heat Capacity	0.999	0
Volumetric Heat Flux Density	0	0
Linear Expansion Coeff	3.83e-005	0
Exp Reference Temperature	0	0
Mass Density	0.0361	0
Viscosity	5.61085e-005	0

MatID Name: 1) Water

Model>Define>ElementBehavior>Thermal Conductivity, Specific Heat Capacity
Specify the isotropic material's thermal property, the specific heat capacity is necessary for transient response analysis.

Model>Define>ElementBehavior>Volume Heat Flux Density
Specify a volume distributed heat flux associated with the material index. This is usually used to simulate a heat sink or a heat source situation. Note that the same effect could also be achieved using the nodal-based heat load (but it then require the user to specify the integrated nodal value. So, it is not as convenient as the volume based heat flux density).

Model>Define>ElementBehavior>Linear Expansion Coeff, Expansion Reference Temperature
This option allows the user to specify the thermal expansion coefficient and the reference temperature. It is mainly applicable for thermal-stress coupling, or when buoyancy effect in the fluid is expected. Note that when in thermal fluid application, the buoyancy effect is computed based on the Boussinesq assumption - i.e. the thermal expansion term only affects the density variation in the gravity calculation, and not in the momentum terms.

Model>Define>ElementBehavior>Mass density.
For fluid flow analysis, the mass property is always necessary. For a pure thermal analysis, the mass density is only necessary when the transient response is desired.

Model>Define>Element Behavior>Nonlinear Multiplier Index
This index allows the control of the material data to be dependent on the available system variables such as time or stress or strain, etc. For further information see the nonlinear property multiplier dialog session.

Model>Define>Element Behavior>Anisotropic/Orthotropic Material

Specify the anisotropic material's property is active. To specify the material's anisotropy, click the "Setting" button to define the material data. Please note that anisotropic thermal material property only apply to thermal continuum, not to the fluid material properties.

The Model Define Element Behavior - Electro/Magneto Menu

The screenshot shows the 'Define Sheet Material id=1' dialog box with the 'ElectroMagneto' tab selected. The dialog contains the following fields and controls:

- Edit Nonlinear Multiplier Data** button
- Nonlinear Multiplier Index** input field (value: 0)
- Electric Conductivity/Dielectricity** input field (value: 1)
- Volumetric Charge Density (Dielectric Analysis only)** input field (value: 0)
- Piezoelectrical Material..**
- Anisotropic/Orthotropic Electric Material..**
- Electric Joule Heating to Thermal Energy Conversion Factor (Conductive Analysis Only)** input field (value: 0)
- Magnetic Permeability** input field (value: 0)
- MatID Name** dropdown menu (value: 1)AIN)
- Previous** button
- Next Mat ID** button
- Apply** button

Model>Define>ElementBehavior>Electric Conductivity/Dielectricity

Depending on the system control setting, this specifies either the electrical conductivity (for conductive analysis) or the absolute dielectric property (for the dielectricity analysis).

Model>Define>ElementBehavior>Volume Charge Density

For a dielectric analysis, the user can specify a volume-distributed charge associated with the material index. Note that the same effect could also be achieved using the nodal-based charge load (but it then requires the user to specify the integrated nodal value. So, it is not as convenient as the volume based heat flux density).

Model>Define>ElementBehavior>Magnetic Permeability

This specifies either the magnetic permeability property.

Model>Define>Element Behavior>Nonlinear Multiplier Index

This index allows the control of the material data to be dependent on the available system variables such as time or stress or strain, etc. For further information see the nonlinear property multiplier dialog session.

Model>Define>Element Behavior>Electrical Joule Heating to Thermal Energy Conversion Factor

Since the energy unit conversion among different forms of energy (such as electrical, thermal, or mechanical) is dependent on the unit used in the model, it is necessary to use a energy conversion factor to convert the electrical Joulian heat to the equivalent thermal energy. The electrical joule energy is computed from the heat generation $E=IV$ (per unit volume sense), and

the conversion factor is used to convert this electrical Joulian energy into the thermal energy unit. For example, the energy unit Joule to BTU conversion factor is 0.00094845166 since one Joule equals 0.00094845166 BTU.

Anisotropic/Orthotropic Material Definition

Model>Define>Element Behavior>Anisotropic/Orthotropic Material/Setting

This dialog is for general anisotropic or orthotropic material data input. It could be one of the follow types.

Anisotropic/Orthotropic Material Definition							
Material Orthogonal a, b, c axes direction							OK
							Cancel
direction a vector	x	y	z				
	1	0	0				
direction b vector	0	1	0				
Young's modulus	Ea	Eb	Ec	Nonlinear Multiplier Index			
	30000000	30000000	30000000	0			
Poisson's Ratio	Vba	Vca	Ecb				
	0.3	0.3	0.3	0			
Shear Modulus	Gab	Gbc	Gca				
	11538461.538	11538461.538	11538461.538	0			

Thermal Anisotropic/Orthotropic Material Definition							
Thermal Material Orthogonal a, b, c axes							OK
							Cancel
direction a vector	x	y	z				
	1	0	0				
direction b vector	0	1	0				
Thermal Conductivity	Ka	Kb	Kc	Nonlinear Multiplier Index			
	0.0007612	0.0007612	0.0007612	0			
Thermal Linear Expansion Coefficient	Ba	Bb	Bc				
	8e-006	8e-006	8e-006	0			

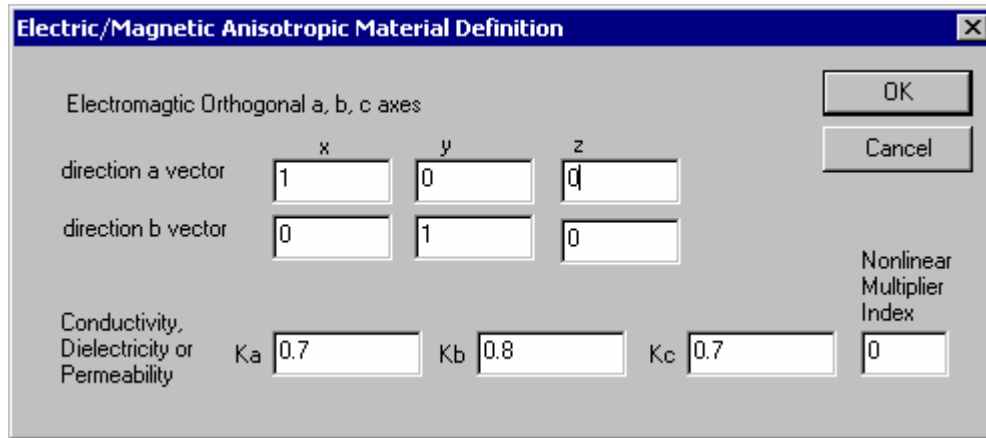


Fig. 4.8 The anisotropic/orthotropic material dialog

Model>Define>Stress>Setting>Directoin a/b vector

General anisotropic material with material axes "a", "b", and "c" data can be entered here. The orientation of the main material axis "a" vector direction must be entered followed by the material axis "b" vector direction. The "a" and "b" axis vectors now define the material "ab" plane formed by this two vectors. The third material axis direction "c" is indicated by the normal direction of this plane "ab". If vector "b" is not perfectly orthogonal to the "a" axis, it will be adjusted during the AMPSol solution calculations, so it will be normal to both axis "a" and the internally computed normal "c" axis.

The 2D orthotropic material following exactly the same definition as the 3D anisotropic material axis orientation, and the user has to enter a meaningful vector "a" and "b" direction in the global XY plane.

For shell material in the 3-dimensional space, the vector "a" direction as entered will be projected to the shell surface to indicate the shell orthotropic major axis "a" direction. If the "a" axis projection is zero (e.g., happen to be in the shell's normal direction), the 2nd orthogonal axis "b" is then used to project to the shell and to indicate the 2nd orthotropic direction.

Using this method, all 2d/3d/shell anisotropic material definition can be shortened into this simple consistent dialog for material definition .

Model>Define>Stress>Setting>Young' Modulus, Poisson's Ratio, Shear Modulus

Enter General anisotropic material's Young's modulus, Poisson's ratio, and the shear modulus in the material axes a, b, and c directions. Note that even if you are analyzing a 2D or shell material, all 3 directions material +data must be entered since they are necessary for the material property transformation calculations.

The entered material property must follow the following constitutive equations:

$$\boldsymbol{\sigma} = \mathbf{C} \boldsymbol{\varepsilon}$$

where

$\boldsymbol{\sigma} = \{\sigma_{aa}, \sigma_{bb}, \sigma_{cc}, \tau_{ab}, \tau_{bc}, \tau_{ca}\}^T$ and $\boldsymbol{\varepsilon} = \{\varepsilon_{aa}, \varepsilon_{bb}, \varepsilon_{cc}, \gamma_{aa}, \gamma_{bb}, \gamma_{cc}\}^T$ are the engineering stress and strain tensors. The Poisson's ratio is defined as $\nu_{ij} = -\varepsilon_j / \varepsilon_i$ as interpreted as the strain reduction in the J direction subjected to the uniaxial stress test in the I direction. And the constitutive matrix C is entered as:

$$C^{-1} = \begin{bmatrix} \frac{1}{E_a} & -\frac{\nu_{ba}}{E_b} & -\frac{\nu_{ca}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ab}}{E_a} & \frac{1}{E_b} & -\frac{\nu_{cb}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ac}}{E_a} & -\frac{\nu_{bc}}{E_b} & \frac{1}{E_c} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{bmatrix}$$

Note that the coefficient of these material constants must satisfy

$\nu_{ab}/E_a = \nu_{ba}/E_b$, $\nu_{ca}/E_c = \nu_{ac}/E_a$, and $\nu_{cb}/E_c = \nu_{bc}/E_b$.

Further more, $\nu_{ba} < \sqrt{E_b/E_a}$, $\nu_{ca} < \sqrt{E_c/E_a}$, and $\nu_{cb} < \sqrt{E_c/E_b}$.

For shell problem, since the local material axes "a" and "b" are always parallel to the shell mid-plane curvilinear surface, and the "c" axis is always the normal direction, it is common to use the shear shape factor reduction by applying $G_{bc}' = G_{bc}/1.2$ and $G_{ca}' = G_{ca}/1.2$.

Model>Define>Thermal/Fluid>Setting>Thermal Conductivity, Thermal Linear Expansion Coefficient

Enter the anisotropic thermal conductivity and the thermal expansion coefficient in the local material axes a, b, and c directions. If both thermal and stress anisotropic materials are defined, the local axes are shared in both definition.

Model>Define>Thermal/Fluid>Setting>Conductivity, Dielectricity, or Permeability

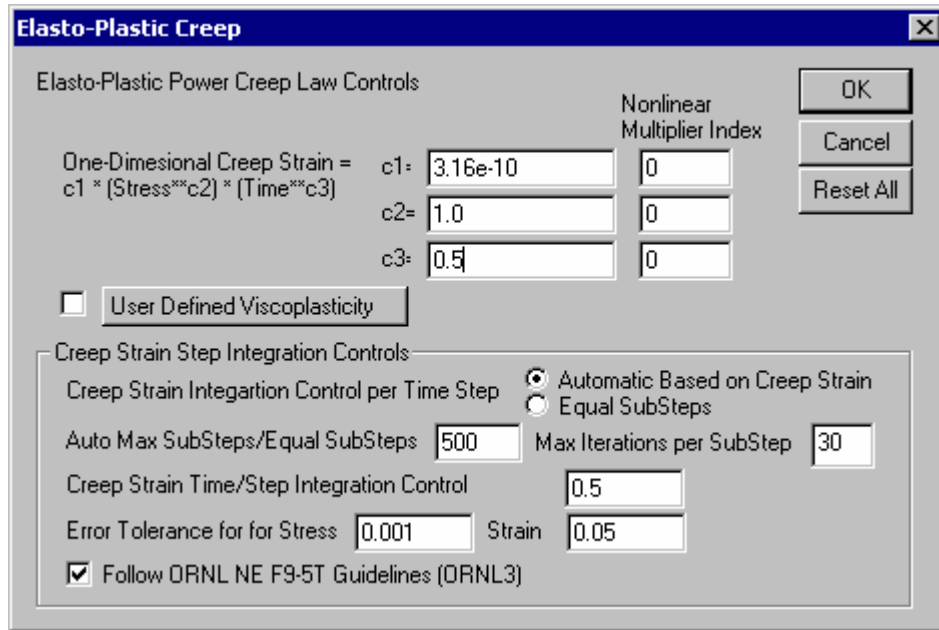
Enter the electrical conductivity or dielectricity, or magnetic permeability coefficient in the local material axes a, b, and c directions.

Model>Define>Stress>Setting>Nonlinear Multiplier Index

This index allows the control of the material data to be dependent on the available system variables such as time or stress or strain, etc. For further information see the nonlinear property multiplier dialog session.

Creep Material Data and Controls Menu

The creep effect uses a power creep law to compute the creep strain, or $\epsilon^{\text{creep}} = c_1 \sigma^{c_2} t^{c_3}$ in a one-dimensional uniaxial creep law, σ being the uniaxial stress, and t is the time scale. Usually, these coefficients c_1 , c_2 , and c_3 can be obtained from a creep test, or from a material data sheet.



Model>Define>Stress>Setting>c1, c2, c3

These are the uni-axial creep law constant to control the creep strain calculation. The constant c2 is the power constant applied to the effective uniaxial stress, and c3 is the power constant for the time dependent effect. The c1 constant controls the total scaling. If these constants are also dependent on the system run time variables, then additional Nonlinear Multiplier Index can be assigned to each constant. Note only non-zero multiplier index indicates the need for scaling, and the corresponding index must be entered in the Define property sheet.

Model>Define>Stress>Setting>Creep Strain Integration Control per Time Step: Automatic Based on Creep Strain/Equal SubSteps

Sometimes, the creep behavior is very nonlinear, especially when the stress or time dependency power constant are greater than one. In the conversion of the multi-axial stress state into the effective stress and the effective strain in order to compute the effective time, the process may not converge due to the inelastic behavior. As such, sub stepping must be used. The automatic sub stepping uses the incremental creep strain computed and automatically decreases or increases the step size to achieve the time step creep strain calculation based on the request accuracy. The equal sub stepping method uses the specified sub step size to compute the creep strain. Since the creep strain rate is not always equal in all time/steps, it is suggested to use the automatic sub stepping to control the sub step size automatically.

Model>Define>Stress>Setting>Auto Max SubSteps/Equal SubSteps

This is maximum sub steps that the automatic stepping algorithm is allowed to take, or the total sub steps that the manual equally sub stepping controls.

Model>Define>Stress>Setting>Max Iterations per SubStep

This is maximum iteration allowed for the nonlinear creep strain calculation.

Model>Define>Stress>Setting>Creep Strain Time/Step Integration Control

This is the Euler integration parameter. It's value must be between 0.0 and 1.0. When this parameter is 0.0, the elasto-plasto-thermo-strain is computed based on the previous step

value, i.e., an explicit approach. When it is 1.0, it becomes an implicit method assuming the full projected strain based on the previous step. The default setting is 0.5.

Model>Define>Stress>Setting>Error Tolerance for Stress and Strain

This is the error tolerance setting for each sub step stress and strain calculations.

Model>Define>Stress>Setting>Follow ORNL NE F9-5T Guidelines (ORNL3)

When this check box is active, the creep strain calculation procedure follows the most recent 1986 DOE Nuclear Standard NE F 9-5T "Guidelines and Procedures for Design of Class I Elevated Temperature Nuclear System Components" published by Oak Ridge National Laboratory recommends criteria for creep strain hardening and calculation procedure (the so called ORNL3). The computed results following these guidelines are most noticeable in cyclic loading situations since the strain hardening calculation guidelines require the tracking of the stress reversal and strain origin and uses that for the creep strain hardening calculations.

Piezoelectrical Material Data

The piezoelectrical material must be supplied by the user for the stress-electrical coupled analysis. There are two possible types of data format: piezo-strain or piezo-stress matrix formats. This is usually based on the manufacturer's supplied information. The piezo-strain matrix relates the electrical potential gradient to the S11, S12, S12, S22, S23, S33 strain for each gradient component, while the piezo-stress matrix relate the potential gradient directly to the stress tensor.

The dialog box titled "Piezoelectrical Material" has two radio buttons: "Piezo-Strain Matrix" (unselected) and "Piezo-Stress Matrix" (selected). It contains a table with columns for Voltage Gradient 1, Nonlinear Multiplier Index, Voltage Gradient 2, Nonlinear Multiplier Index, Voltage Gradient 3, and Nonlinear Multiplier Index. The rows represent strain components S11, S12, S13, S22, S23, and S33.

	Voltage Gradient 1	Nonlinear Multiplier Index	Voltage Gradient 2	Nonlinear Multiplier Index	Voltage Gradient 3	Nonlinear Multiplier Index
S11	0	0	-0.58	0	0	0
S12	0	0	1.55	0	0	0
S13	0	0	-0.58	0	0	0
S22	-0.48	0	0	0	0	0
S23	0	0	0	0	0	0
S33	0	0	0	0	-0.48	0

Nonlinear Property Multiplier

Model>Define>Element Behavior>Edit Nonlinear Multiplier Data.

Other than a specified value, each material property value may be multiplied with a nonlinear function of any computed variable. This nonlinear multiplier is defined by this option and assigned a nonlinear multiplier index to the associated material property. The following dialog will show up

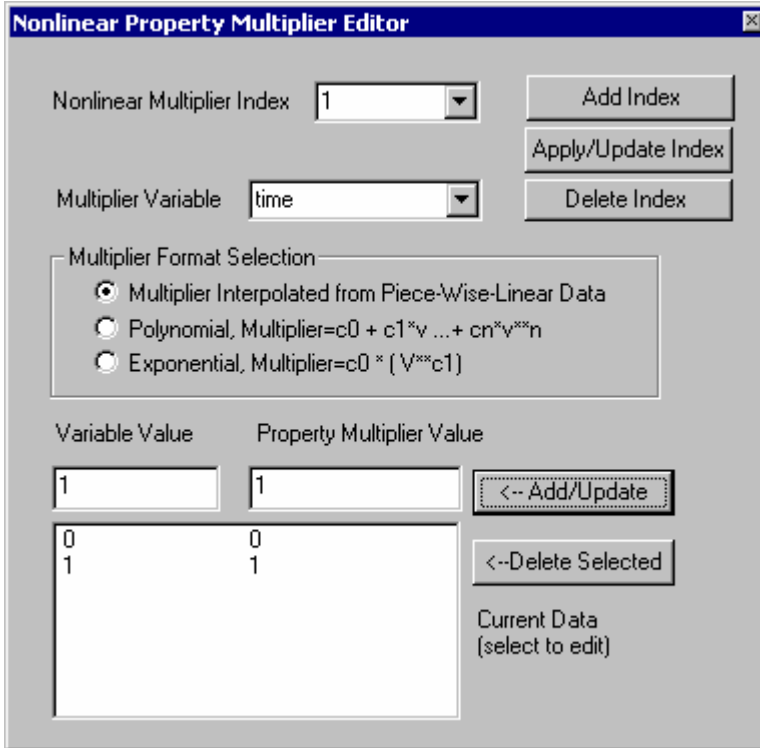


Fig. 4.9 The nonlinear property multiplier dialog

The Multiplier Index is the index to be used. Add/Apply/Update/delete options will add or update the index's data, or delete the index information. The multiplier's dependent variable is specified from the drop-down box "Multiplier Variable". The availability of the variable will depend on the type of the analysis performed, and the material models used. After that, there are three type of multiplier functions

1. Piece-wise linear interpolated
 Pairs of the dependent variable values and the multiple values must be entered in the multi-list box corresponding to the variable and multiplier.
2. Polynomial expansion
 This allow the multiplier to be computed as

$$M(v) = c_0 + c_1v + c_2 v^{\wedge}2 + c_3 v^{\wedge}3 \dots + c_n v^{\wedge}n$$
 Where v is the dependent variable and M(v) is the multiplier function's value. The coefficients are entered in the multi-list box.
3. Exponential expansion
 This allow the multiplier to be computed as

$$M(v) = c_0 v^{\wedge}c1$$
 Where v is the dependent variable and M(v) is the multiplier function's value. The coefficients are entered in the multi-list box.

Beam/Rod Section Properties Menu

The sectional property menu is designed to specify all beam/rod section properties and the optional member loading controls. The main difference between beam and rod is the rigidity of bending associated with the rotational degree of freedom in beam. Both beam and rod can be used in either 3D or 2D. The beam element has three rotational and translational DOF's in 3D, but only have two translational DOF's in 2D analysis, and one rotational DOF (rotation about

the Z-axis). The rod element has three translational DOF's in 3D, and two in 2D, both without any rotational DOF.

Model>Define>Beam/Rod Cross Section Type

There are total of four types of predefined beam/rod cross section types:

- 1) **User defined sectional properties:** enter the pre-computed data of area, torsional rigidity, and the moment of inertia about the local a and b axes.
- 2) **Rectangular section:** enter the width and the height of the cross section and the shear shape factor if shear stress correction for deep beam is desired.
- 3) **Circular section:** enter the diameter of the cross section and the shear shape factor if shear stress correction for deep beam is desired.
- 4) **Tubular section:** enter the outer and the inner diameters of the cross section and the shear shape factor if shear stress correction for deep beam is desired.

If the shear stress correction is desired, enter the shear shape factor or enter zero to bypass shear stress correction. The shape factor accounts for the discrepancy of actual parabolic shear stress distribution profile as compared with the simple uniform shear stress assumption. The shape factor is the ratio of the actual shear force over the uniform average shear force, and is dependent on the Possions' ratio. For rectangular cross section, it is $(12+11v)/(10(1+v))$, and for the circular cross section, it is $(7+6v)/(6(1+v))$, and for tube, $(4+3v)/(2(1+v))$, where v is Possions' ratio. More information of this shape factor can be found in the traditional elasticity theory.

For "User Defined Sectional Properties", these are the general terminology used:

Area: the cross section area.

Moment of Inertia M.I: This is the bending flexibility of the beam cross section. For a rectangular section, it is $M.I.=width*height^3/12$ about the major axis bending (local axis b), or $M.I.=height*width^3/12$ about the minor axis (axis 3).

Tensional Rigidity J: It is the torsional rigidity needed to compute the torsional rotation angle from the applied torque based on the formula $\theta = TL/GJ$, where **T** is the applied torque, **L** being the length of the beam, and **G** being the shear modulus of the beam. For a circular section it is $0.5*\text{吋}^4$ (and the Moment of Inertia for such section is $0.25*\text{吋}^4$) where r is the radius.

For a general section property, an elasticity textbook or AISC Design Handbook is useful in finding the section properties. Please also note that that the user-defined section properties can be used in the large deformation/rotation analysis, but can not be used in the elasto-plastic analysis since the exact integration point plastic strain history can not be tracked in the such a section where geometry shape is unknown. If plasticity is expected in the beam section, only the sections of know shapes shape can be used. An exception is the rod element since the cross section is either tensile or compressive, so the plastic state of user-defined input section can also be used in plasticity.

Model>Define>Beam/Rod Section Property>Sectional Property Orientation

The beam/rod local uses local axes a, b and c (sometimes are also referred to as the local axes 1,2,3 in some output nomenclature) system to identify the orientation of the section. The a-axis direction is along the beam/road axial direction. The user enters the local b-axis direction through the local b-vector component. An additional b-axis vector specification is used is the b-axis vector specified happens to be coincident with the beam/rod member axis.

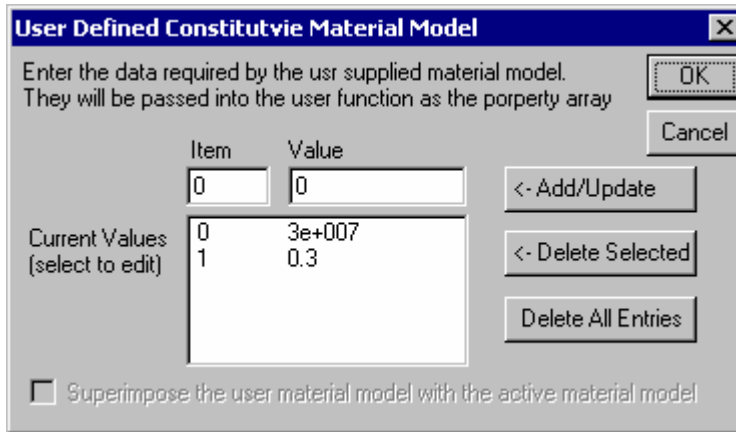
Model>Define>Beam/Rod Section Property>Member Loading

Other than the regular boundary condition control, the beam/rod element has the additional member loading ability.

- 1) **Uniformly Distributed Local Axis Loads:** this allows the user to specify a uniformly distributed load along the local a, b and c direction. The total force will be calculated from the integration of these uniformly distributed loads after axes transformation. It is necessary to enter the Time Function index to control the time multiplier. For more information about the Time Function, please refer to the Model/Condition section.
- 2) **Initial Tension and/or Tension/Compression Only:** this setting is only applicable to rod element. To specify an initial tension, enter a positive tensile stress, and enter a negative stress for initial compressive state. The rod can also function as a tension-only state (such as a rope) or a compression-only mode (such as a compression-only spring contact). In such modes, the rigidity of the rod is scaled by a factor of $1e-30$ till the rod element become active again, and the corresponding external loads/forces are redistributed during the calculation. For further flexibility, the user can enter a slack distance to specify the initial tensile/compressive length change before the tension/compression mode will engage. A typical application of this control is when trying to model a directional opening gap when a compressive rod must travel through the initial opening before the compression only mode engages.

User Defined Constitutive Material Model

When the user select the user-defined constitutive model, additional material property data can be stored in AMPView and then later pass to the user supplied routines. This dialog box is used to store such property array data in the model.



Enter the data required by the user supplied material model. They will be passed into the user function as the property array

Item	Value
0	0

Current Values (select to edit)

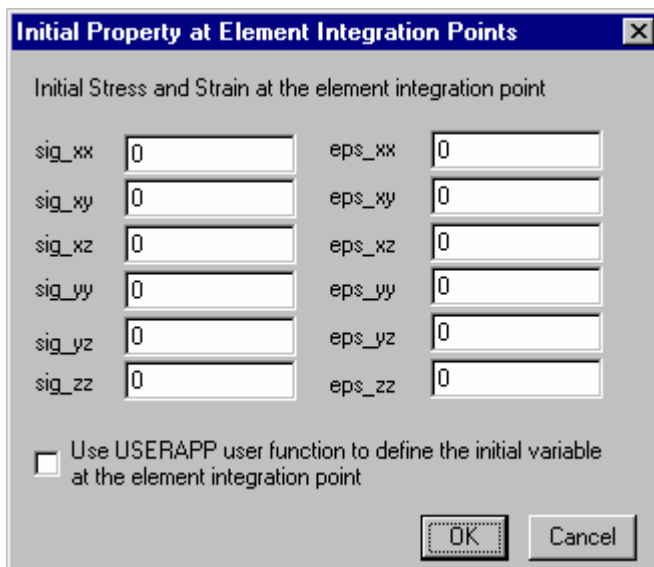
0	3e+007
1	0.3

Superimpose the user material model with the active material model

Buttons: OK, Cancel, <- Add/Update, <- Delete Selected, Delete All Entries

User Defined Initial Stress and Strain

When this option is checked, the default initial stress and strain can be initialized for each element at each integration point. This is usually used for problems that require initial pre-stress or need the strain history for specific material model.



Initial Stress and Strain at the element integration point

sig_xx	0	eps_xx	0
sig_xy	0	eps_xy	0
sig_xz	0	eps_xz	0
sig_yy	0	eps_yy	0
sig_yz	0	eps_yz	0
sig_zz	0	eps_zz	0

Use USERAPP user function to define the initial variable at the element integration point

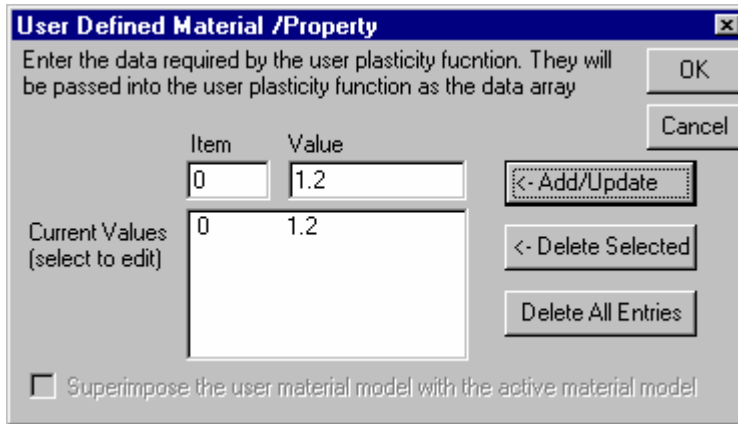
Buttons: OK, Cancel

Additionally, if these stress and strain initialization is not enough, and more other properties need to be initialized, the user can select the "Use USERAPP..." check box, and initialize the desired variable in the USERAPP interface routine. Detailed of this user interface application can be found in the "AMPS User Application Interface Reference Manual."

User Defined Plasticity Material

For general plasticity, AMPS provides a quick way for the user to define a user defined plasticity material model. Instead of writing a full constitutive material model, the user can just supply the plastic yield function and the flow rule (flow function). Usually, the yield and flow rule are the same for metal, and non-associated flow rules are used for brittle and geological materials. When the user selects the user-defined plasticity model, additional plasticity property data can be stored in AMPView and later pass to the user supplied routines during program execution.

This dialog box is used to store such property array data in the model. For more details, please refer to "AMPS User Application Interface Reference Manual."



Material model and data

The material type is defined in the input file (*.dat file) is specified by first defining the type index, field1,field2 ...The field variables refer to the parameters associated with the field governing equations. Currently the continuum stress field and the thermal field are the only two types of property that are supported. After this declaration the user also has to specify the property type according to the following specifications.

Property	Data
Plane stress thickness	plane stress thickness
Integration points	Gauss/Lobotto/Reduce/Default
Density	Rho, the material density
Elastic Poisson's ratio	Nu, Poisson's rato
Elastic Young's modulus	E
Linear Expansion Coeff.	beta coefficient
Hyper Mooney Rivlin	C1,C2, hyperealstic type, Mooney Rivlin
Plastic kinematic hardening	Beta_ratio This controls the kinematic hardening ratio (0 to 1)
Plastic Mohr Coulomb	Cohesion, friction angle
Plastic Von Mises	Initial yield stress
Thermal capacity	C
Thermal conductivity	K

Model/Conditions Menu

The Boundary Condition Menu

The Conditions Sheet allows the boundary conditions to be defined interactively. The user first selects a F.E. geometric entity or multiple entities by graphical selections, and then the type of

boundary condition to be applied is then defined. After that, the specific values associated with the boundary condition type are assigned. Then the user specifies the time range in which the boundary condition is to be applied. Each of the above activity, referred as "geometry", "type", "value", or "time", is assigned an index or a name. The boundary condition is actually applied by the selection of the combinations of the four type and then specified as the final "boundary condition" index. Using this approach, the boundary conditions can be specified by selecting the combinations of different geometry, type, value, and time indexes.

It is helpful for the user to turn both the node and the element side display on in order to review the applied boundary conditions.

The Model Condition Menu

The boundary condition is defined by a combination of Geometry, Time, Type and the associated additional data. Once these boundary condition data are set, the Condition page is used to define a complete set of boundary condition. When the user enter the Condition page the first time, most of the menu are disabled till the Geometry, Time and Type pages are specified, then a New Condition button will be enabled to create a new boundary condition.

Once a boundary condition set has been defined, this is the best place to quickly review what the boundary conditions of a model, especially when there are many condition sets. If you switch the combination set, the geometry set associated with the boundary condition will be automatically brought up and shown in highlight, and the type/value/time page will be automatically recalled ready for review.

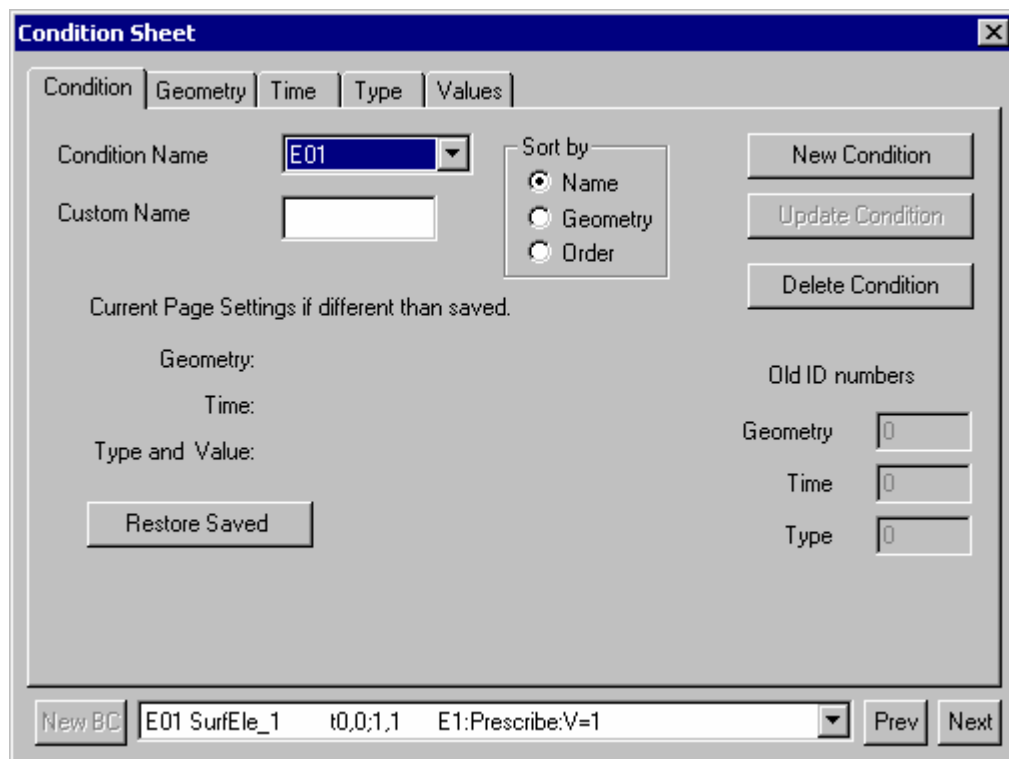


Fig. 4.17 Condition Combination Page

Model>Condition>Condition>Condition Name

This name is generated automatically by the "New Condition" button. The user can also enter a Custom Name to better indicate the Condition Name. It is also used as a pull-down list to select one from the existing available selections, if any.

The Model>Condition>Combination>Custom Name

This is an optional name that the user can enter to specify a meaningful name to a Condition set.

Model>Condition>Condition>New Condition

This will generate a new Condition Name based on the already define Geometry, Time, Type data as listed in the bottom pull-down list. name is generated automatically by the "New Condition" button. The user can also enter a Custom Name to better indicate the Condition Name. It is also used as a pull-down list to select one from the existing available selections, if any.

Model>Condition>Combination>Update Condition

If there are any changes to the combination of the Condition, this Update Condition button will be enabled to allow the saving of the changes.

Model>Condition>Combination>Delete Condition

This will delete the selected Condition set.

Model>Condition>Combination>Sort by Name/Geometry/Order

When there are many boundary condition set, this is a convenient way to sort the pull down list of the condition either by their name, geometry name, or order it was entered.

Model>Condition>Combination>Restore Saved

If there is any change to the Condition set, the different Geometry, Time or Type will be shown. If the user wants to revert to the originally setting, the "Restore Saved" button will retrieve the saved data and restore the Condition setting.

Model>Condition>Combination>Old ID Names

These areas are mainly for users who are familiar the previous versions (earlier than 3.03 version) that use index rather than the condition name for geometry/time/type indications. They are displayed here mainly for informational purpose. It is also used to indicate the generation sequence index of the internal Geometry, Time, and Type names for information purpose.

Model>Condition>Combination>New BC, Prev, Next

These are the general Condition Sheet control button. The New BC is the same as the New Condition button, except it can be accessed from any page. The Prev and Next are two useful buttons to page throw the defined boundary condition sets. The pull-down load at the bottom of the Condition Sheet is for the quick identification of the condition set, and it tries to use the abbreviation so the user can quickly capture the nature of the geometry, time, and type values. This pull-down list of the Condition set at the bottom of this Condition page is accessible from any Condition page for convenience.

The Model Condition Geometry Menu

This menu allows geometric F.E. entities to be created

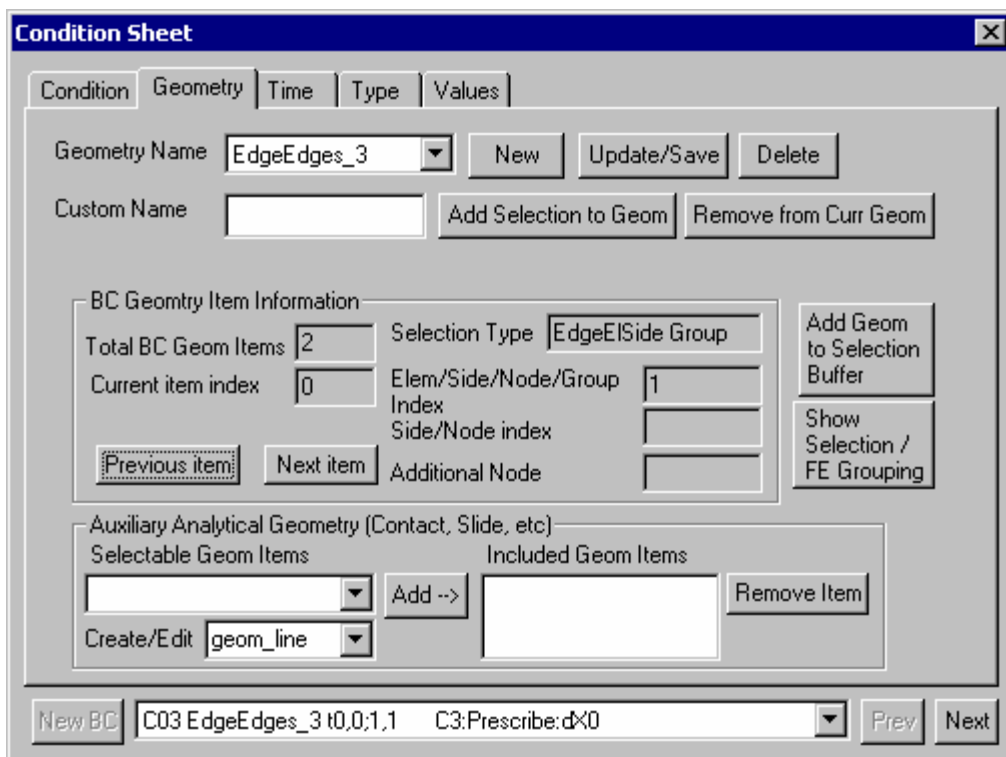


Fig. 4.11 Geometric Condition

Model>Condition>Geometry>Geometry Name

This name is generated automatically by the "New" button. The user can also enter a Custom Name to better indicate the Condition Name. It is also used as a pull-down list to select one from the existing available selections, if any.

The Model>Condition>Geometry>Custom Name

This is an optional name that the user can enter to specify a meaningful name to a Geometry set

Model>Condition>Geometry>New

This will add the current Graphic selection(s) to the next available Geometry Condition Index and create a new Geometry Name. The geometry condition is usually created by first specifying the geometric selection using any of the graphic selection tools, and then click on this button to store the selected geometric items into the geometry condition selection.

Model>Condition>Geometry>Update/Save || Delete

The update/save button is used to update the graphic selection associated with the Geometry Name. The delete button will delete the current Geometry Name and the associated data.

Model>Condition>Geometry>Add Selection to Geom

This will add the current graphic selection(s) to the current geometry name index. This is usually used to add more geometry items to the existing geometry name due to some situation that not all geometry can be easily selected in one time, e.g., multiple internal faces selection for contact/glue boundary condition since they are originally invisible unless some elements are temporarily hidden for internal faces selection.

Model>Condition>Geometry>Remove from Curr Geom

This will remove the current graphic selection(s) from the current geometry name index.

Model>Condition>Geometry>BC Geometry Item Information

This area describes the details of the geometric entity in the geometry set. The details of each item will appear in the "Selected Type" area with details of the geometry.

Model>Condition>Geometry>Previous Item/Next Item

If there are multiple geometry items, the current item may be advanced or reversed by the Previous item button or next item button. The information will include the type of the geometry such as the basic FE Element, Side, Node, or the grouped geometric selection BodyElement, FaceElement, EdgeEISide, etc. It also contains information about its index. If a face or a side of an element is selected, it will also show the associated face count within the element, or the node numbers associated with the side.

Model>Condition>Geometry>Add Geometry to Selection Buffer

This will add the current graphic selection(s) to the global selection buffer. This is usually used to automatically set the global selection buffer from the stored the geometry index so they can be used for subsequent selection operations. existing Geometry Condition Index.

Model>Condition>Geometry>Show Selection / FE Grouping

The will bring up the "FE Group/Display" dialog box so the user can examine the selected elements, geometry items, or groups. This is usually for confirming the visual geometry selection items with the underlying element or geometry numerical information. For detailed information, please refer to the Finite Element/FE Group/Display control.

Model>Condition>Auxiliary Analytical Geometry

This session defines the analytical geometry that can be used in the analysis. Current, only contact/slide definition uses analytical geometry. The pull-down list will list the available geometry for selection, and the "Create New" button can used to add new analytical geometry. Clicking the "Add->" button will include the selected geometry into the "Included Geometry Items" list, and the "Remove Item" button will remove the selected item in the list.

** Note: Sometimes it is necessary to select the geometry entities that are very close together, or some of the items are hidden inside the body. In such case, it is usually easier to select the visible items first, and store the selections into the geometry index. After that, use the FiniteElement/FE Group/Display display grouping and hiding control to hide the undesirable portion of the model so the desired element/node/geometry become visible. After that, use the "Add Selection to Geom" button to add the new selection to the previous stored the geometry selection. For more information about finite element graphic selection, refer to the Selection Menu section.

The Model Condition Time Menu

The time range over which the condition applies is specified in this menu.

Fig. 4.16 The Condition Time Page

Model>Condition>Time>Time Name

This name is generated automatically by the "New" button. The user can also enter a Custom Name to better indicate the Time condition name. It is also used as a pull-down list to select one from the existing available selections, if any.

The Model>Condition>Time>Custom Name

This is an optional name that the user can enter to specify a meaningful name to a Time data set

The Model>Condition>Time>New

This will create a new Time Name based on the current time definitions entered.

Model>Condition>Time>Update/save || delete

This button updates and save the time data associated with the Time Name. The delete button will delete the current Time Name and all associated data.

Model>Condition>Time>Initial Boundary Condition

This options specifies the boundary condition is only for the initial condition only, and will not be applied in the subsequence analysis steps. This is usually used to specify the initial deformation, temperature, velocity or other nodal based variables' initial values.

Model>Condition>Time>Time, Load Factors

The index is associated with a time range and a load scaling factor at the start and one at the end. The load factor is interpolated in between. The user may optionally specify any intermediate breakpoints.

The Model Condition Type Menu

This menu specifies the type of the boundary condition desired. Additional values needed for the type specified can then be entered in the next page immediately following this page. The Type selection can be classified into the following categories: Continuum, Thermal, Fluid, Electro/Magnetic, and General. To avoid unnecessary choices, some of the boundary condition types will be disabled from selection. Once the user chooses the Type category, the available boundary condition Type will be active for selection.

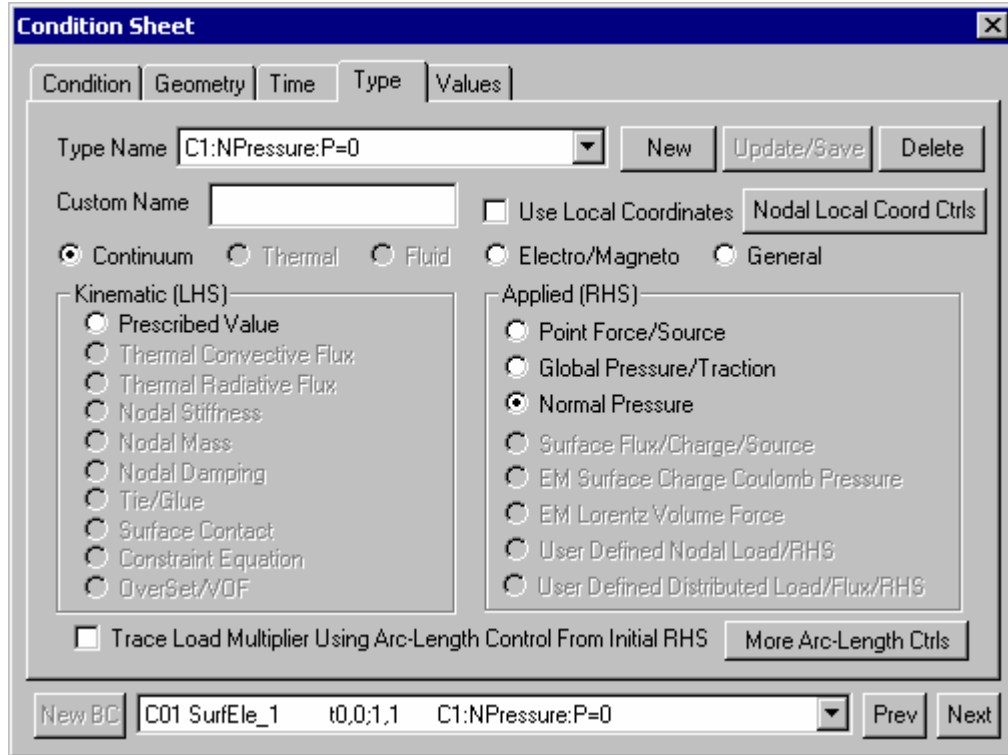


Fig. 4.12 The Condition Type menu

The Model>Condition>Type>Type Name

This name is generated automatically by the "New" button. The user can also enter a Custom Name to better indicate the Condition Name. It is also used as a pull-down list to select one from the existing available selections, if any.

The Model>Condition>Type>Custom Name

This is an optional name that the user can enter to specify a meaningful name to the Type set

The Model>Condition>Type>New

This button creates a new Type Name and stored the selected condition Type under the index. The user can also enter a custom Type name to better indicate the type of the condition.

The Model>Condition>Type>Update/Save || Delete

The update/save button is used to update the Type selection associated with the Type Name. The delete button will delete the current Type Name and the associated data.

The Model>Condition>Type>Kinematic (LHS) || Applied (RHS)

The user may select one type from any of these radio buttons. The Applied (RHS) refers to the nature that these conditions generally contribute to the right-hand-side of the system equation,

and the kinematic conditions usually restrict the left-hand-side solution unknown to be restrained to certain specified values.

Model>Condition>Type>Prescribed Values

This selection applies a LHS nodal unknown. This is normally used to specify nodal displacement, temperature, voltage, etc. Depending on the geometric dimension of analysis, either 3 (3D) or 2 (2D) magnitude values must be entered in the directional value page.

Model>Condition>Type>Thermal Convective || Radiative Flux

This applies a convective or radiative boundary condition in the thermal problem. The convective or radiative coefficient and the associated ambient condition must be entered in the corresponding value page.

Model>Condition>Type>Nodal Stiffness || Nodal Mass || Nodal Damping

This specifies the additional nodal values to a specific node or nodes for physics. Depending on the geometric dimension of analysis, either 3 (3D) or 2 (2D) magnitude values must be entered in the directional value page

Model>Condition>Type>Tie/Glue

This specifies a tie/glue boundary condition to be enforced on the geometry selected. Tie/Glue can only be specified on the boundary of the model. In 2D analysis, the allowed geometry can be geometry side, element side, or nodes and element. In 3D, this also includes element face. In general, at least two geometric selections are required to construct a tie/glue (a master and one of multiple slave geometry). The nodes on the slave geometry are then kinematically constrained to the master surface/side based on the relative initial geometry, and will be enforced throughout the analysis unless a time-controlled tie/glue is specified. Details controls can be found in the Tie/Glue setup page.

Model>Condition>Type>Surface Contact

This specifies a contact boundary condition based on the geometry selected. Contact can only be specified on the boundary of the model. 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. Possible contact parameters for stress and thermal setting must be entered in the surface contact value page.

Model>Condition>Type>Constraint Equation

This specifies a constraint equation condition among the selected nodal geometry. The constraint equation condition can only be applied to finite element nodes or geometry vertices. Details controls can be found in the Constraint Equation setup page.

Model>Condition>Type>OverSet/VOF

This specifies an advanced boundary condition when a finite element group (OverSet Entity) is moving across another finite element group (OverSet Domain). The user must specify the OverSet type and select the proper geometry association. It can also be used as the traditional VOF (volume of the fluid) method specification for applications such as bubbling in water. In such case, the air bubble is the OverSet Entity, and the water is the OverSet Domain.

Model>Condition>Type>Point Force/Source

This selection is to apply a RHS nodal value. This is normally used to apply nodal forces, nodal heat flow, current etc. Depending on the geometric dimension of analysis, either 3 (3D) or 2 (2D) magnitude values must be entered in the directional value page.

Model>Condition>Type>Global Pressure/Traction

A global pressure/traction applies uniform traction (general pressure) to the surface (3D) or edge (2D) to the model. The traction magnitude is specified by the components in the global direction in the value page. Depending on the geometric dimension of analysis, either 3 (3D) or 2 (2D) magnitude values must be entered in the directional value page.

Model>Condition>Type>Normal Pressure

A normal pressure/traction applies uniform pressure normal to the surface (3D) or edge (2D) to the model. The positive value follows the stress convention point away from the model.

Global Pressure/Traction. A single scalar value is expected in the directional value page.

Model>Condition>Type>Surface Flux/Charge/Source

A distributed thermal heat flux or dielectric surface charge, or surface distributed source term to the surface (3D) or edge (2D) of the model. A single scalar value is expected in the directional value page.

Model>Condition>Type>EM Surface Charge Coulomb Pressure

This selection is used only when a dielectric electrical analysis is active and when electrical-stress coupled analysis is desired. In such case, on the surface of the conductor, the user can specify the electrical Coulomb force of the surface charge in the electrical field to be computed. The integrated Coulomb force (based on the Maxwell stress tensor and the electrical field intensity) is then applied as a pressure load to the stress continuum equation of the conductor. The associated BC geometry must be either a side (2D) or a surface (3D) for this Coulomb pressure to be meaningful.

Model>Condition>Type>EM Lorentz Volume Force

This selection is used only when a electromagnetic analysis is active and when stress-coupled analysis is desired. In such case, on the specified volume, the Lorentz force is computed and integrated into the stress equations for analysis.

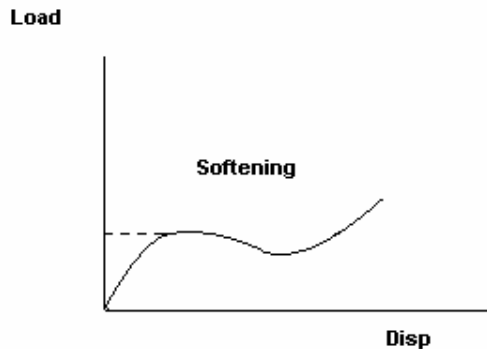
Model>Condition>Type>User Defined Nodal Load/RHS/User Defined Distributed Load/Flux/RHS

This specifies the nodal loads or the distributed loads are defined in the user-supplied nodal load routine. Additional load property data can be stored and passed into the user-defined routine in the subsequent spreadsheet. The distributed load can be used to calculate distributed pressure/flux/LHS/RHS for either a surface (3D) or a side (2D). When this option is used, the user is responsible to carefully define the nodal load following the user interface development guideline. Details of the user-defined material model can be found in the AMPS User Application Interface Reference Manual.

Model>Condition>Type>BC Components Refer to Nodal Local/Axis Directions (if defined)

If this check box is active, then the boundary condition values as specified are referring to the local coordinate system, if it exist for the selected geometry. This is usually useful when trying to specify a directional load/deformation/velocity magnitude on non-global oriented direction. In such case, the user can first define a local coordinate system (see Nodal Local Coordinate System Controls), then check this box to inform the program that the magnitude refers to the local axis directions

Model>Condition>Type>Trace Load Multiplier Using Arc-Length Control From Initial RHS



This specifies an arc-length control loading scheme for the Applied RHS terms. When this is activated, the time/step function no longer controls the RHS applied load. In stead, the program will automatic increases or decreases the load multiplier as is required for the time/step solution. This is usually necessary for models that contain softening point such that a fixed load will generate no solution, or cause the program to diverge since the system can not withstand the applied load. The result from the analysis will be given in the output along with the load multiplier as the result instead of the time/step value. Additional controls of the arc-length method can be set in the Arc-Length dialog entries. For details refer to the "Additional Arc- Length Controls" section.

Additional Arc-Length Controls

Normally, specifying arc length control for a loading type and use the default setting will perform well enough. However, in a very nonlinear problem, additional settings can be achieved to control of the arc length marching method:

Arc-Length Method: The default is the Rik's method. Alternatively, the Crisfield's method is also available. In the Crisfield arc-length, dual solution path is possible, and a forward marching algorithm is used. For details about both methods, see "A Fast Incremental/Iterative Solution Procedure That Handles Snap Through", M. A. Crisfield, IJNME, 1981, pp.55-62.

Maximum Load Increase Factor After a Successful Arc Marching Step (default=3.0): After a successful arc-length load increase and iteration is successful, the next load increment is taken computed based on the ratio of the iteration error norm and the desired error norm. The next arc-length load increment size is computed from this estimated load multiplier based on the specified initial load magnitude. This maximum increase load factor will limit the amount

load to be applied in the new arc-length step. The recommended 3.0 (300% increase) factor is based on several average nonlinear problems we have tested, and usually is a recommended setting. However, if the problem softens or stiffens a lot, i.e. extremely nonlinear, then a smaller setting such as value close to 1.0 is recommended. For a extremely nonlinear softening/stiffening, if the load marching increment is too big, the resultant load-displacement curve will have erratic jump or oscillation, and that's also a good indication of smaller increment is necessary.

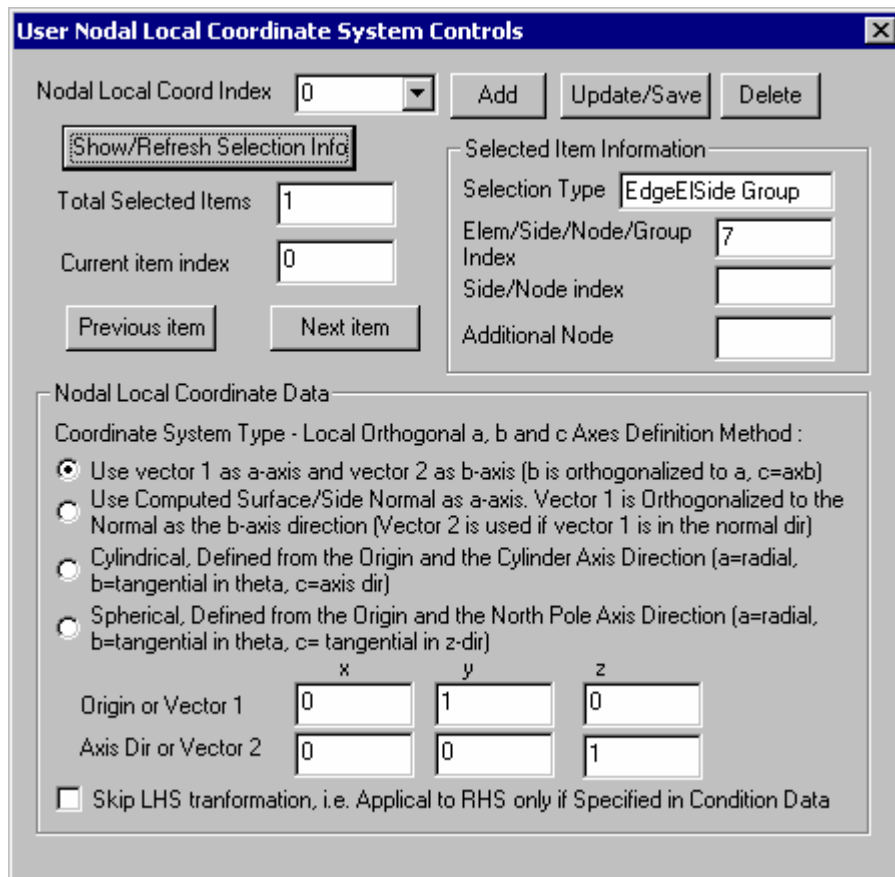
Minimum Load Decrease Factor After an UnSuccessful Arc Step (default=0.001): similar to the above setting, this specifies the minimum load factor will be used when the arc-length loading step size is cutting back during a very nonlinear analysis.

Desired Deformation Increment: The user can directly input the desired incremental deformation size. In such case, the deformation corresponding to the specified incremental load will be scaled so the incremental arc-length size will be roughly equal to the desired deformation size.

Maximum Deformation Increment Size: When this is entered, the arc-length deformation will not march larger than the specified size. If not specified, the program will initially limit the size to no more than 5 times the initial increment.

Minimum Deformation Increment Size: specify the minimum size of the arc-length deformation magnitude to take when the load step is cutting back.

Nodal Local Coordinate System Controls



Model>Condition>Type>Nodal Local Coordinate>Nodal Local Coord Index

The assigns an index number by entering a value or select one from the dropdown list if there are existing indexes. Note that at the first opening of this page, if there are Geometric Condition Indexes that have been previously defined, the program will automatically switch to

the highest index and highlight geometric entities associated with that index for user's information.

Model>Condition>Type>Nodal Local Coordinate>Add

This is a short-cut button. Its purpose is to add the current Graphic selection(s) to the next available Nodal Local Coordinate Index. This is the recommended way of adding a new Nodal Local Coordinate index to avoid overwriting the existing ones. The Nodal Local Coordinate index is usually created by first specifying the geometric selection using any of the graphic selection tool, then store the selected geometric items into the database.

Model>Condition>Type>Nodal Local Coordinate>Update/Save || Delete

The appropriate action is selected to link and update the graphic selection with the Nodal Local Coordinate index. The delete button will delete the Nodal Local Coordinate index.

Model>Condition>Type>Nodal Local Coordinate>Show/Refresh Selection Info

Press this button will refresh the 'Selection Item Information' for reference purpose. It is also automatically updated whenever the graphic index is changed. However, if the selection is changed later, it is necessary to press this button to get the most recent update.

Model>Condition>Type>Nodal Local Coordinate>Selected Item Information/Total Selected Items/Current Item Index

This provides the user with details of the type of geometric entity selected. The details of the selected item will appear on the 'Selected Item Information' area.

Model>Condition>Type>Nodal Local Coordinate>Previous Item/Next Item

If there are multiple selections, the current item may be advanced or reversed by the Previous item button or next item button. The information will include the type of the geometry such as the basic FE Element, Side, Node, or the grouped geometric selection BodyElement, FaceElement, EdgeElSide, etc. It also contains information about its index. If a face or a side of an element is selected, it will also show the associated face count within the element, or the node numbers associated with the side.

Model>Condition>Type>Nodal Local Coordinate>Nodal Local Coordinate Data

There are total of four different types of user defined nodal coordinate system options. The local coordinate system refers to the local a, b and c orthogonal axes.

(1) User specified vector 1 as the a-axis and vector 2 as the b-axis. Later in the processor calculations, the b direction vector is orthogonalized to vector a, and third local axis c is calculated from the cross product of axes a and b,

(2) Use Computed Surface/Side Normal as the local a-axis. Vector 1 is orthogonalized to the normal as the b-axis direction. Vector 2 is used if vector 1 is in the normal direction.

(3) Cylindrical coordinate system. Defined from the Origin and the Cylinder Axis Direction. In such system the local a-axis is along the radial direction, and the b-axis is tangent to the radial in the rotational theta direction. The local c-axis is always along the specified z-axis direction.

(4) Spherical coordinate system. Defined from the Origin and the North Pole Axis Direction. The local a-axis is along the radial direction, and the b-axis is tangent to the radial in the rotational theta direction. The third tangential direction is the projection of the z-axis vector tangent to the local a-axis (radial) direction.

Model>Condition>Type>Nodal Local Coordinate>Skip LHS Transformation

This options specifies that the nodal coordinate specification should not be applied to the LHS transformation, and only applicable to the RHS (applied) data if local components information is intended, as specified in the Condition/Type specification.

Basic nodal DOF choices

The following table specifies the nodal DOF as well as the state of the material properties. These properties are all stored at the nodes. The stresses are calculated from the element integration locations and extrapolated to the nodal position.

Field quantity	Definitions
Displacement	Dx, Dy, Dz are included in the nodal dof freedom. These are the dependent unknown variables in the FE continuum problem.
Velocity	Vx, Vy, Vz these are sometimes included in the unknown dof. At other time they are implicitly calculated from the incremental dof.
Temperature	The temperature field is included in the nodal dof. These are solved as part of a combined mechanics, thermal diffusion and convection problem.
Thermal Flux	Nodal property computed from the thermal gradient
Stress tensor	Nodal property, retrieved from solution
Yield function	State property of material, used for plasticity.
Isotropic hardening	State property defining an isotropic work hardening material
Kinematic Hardening	State property defining an alpha scalar that determines the state of a kinematic hardening elastic plastic material
Elastic strain tensor	State property describing elastic strains
Plastic strain tensor	State property, describes plastic strains
Total strain tensor	State property defining total strains
Velocity/Vorticity	Fluid flow Eulerian formulation velocity and vorticity. At the interface between the fluid and continuum, the velocity value become consistent between the Eulerian and the Lagrangian measurement.
Fluid Pressure	Fluid pressure scalar value

The Model Condition Values Menu

The values menu is dependent on the type of condition chosen. In the diagram we show the menu obtained by specifying the first option in the Kinematic (LHS) Prescribed Values.

Condition Sheet [X]

Condition | Geometry | Time | Type | Values

Name: C4:Prescribe:dX0dY0 [New Type+Value] [Update]

Continuum: Prescribed Value

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

Geometry Follower

[New BC] test EdgeEdges_4 t0,0;1,1 C4:Prescribe:dX0dY0 [Prev] [Next]

There are other types of Values menu page will show up:

Condition Sheet [X]

Condition | Geometry | Time | Type | Values

Name: C4:GPressure:X0Y0 [New Type+Value] [Update]

Continuum: Point Force/Source

Dir	Use	Name	Value
X	<input checked="" type="checkbox"/>	Point Force	0
Y	<input checked="" type="checkbox"/>	Point Force	0
Z	<input checked="" type="checkbox"/>	Point Force	0

Geometry Follower

[New BC] test EdgeEdges_4 t0,0;1,1 C4:GPressure:X0Y0 [Prev] [Next]

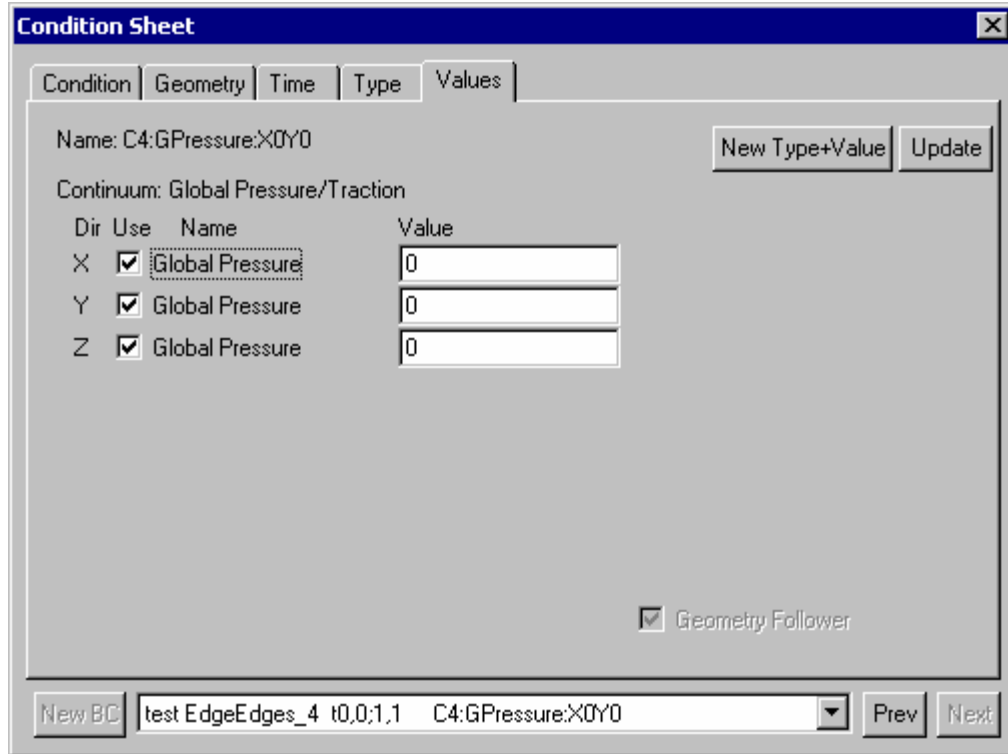


Fig. 4.13 The Directional Values Condition.

This menu allows the user to enter vector data or scalar data, pending on the BC Type that is specified in the Condition Type page. The user is required to check the appropriate variable to associate the entered value to the proper physical variables.

The Model>Condition>Directional Value>Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition>Directional Value>New Type+Value

This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition>Directional Value>Update

This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

Model>Condition>Directional Values>Displacement, Rotation, Velocity, Vorticity, Voltage, Temperature, Point Force, Rotational Load, Current/Charge, Global Pressure, Normal Pressure, Heat Flux, Source, Current/Charge Density, Nodal Stiffness/Mass/Damping, Surface Charge

These are the list of possible items need to be entered pending on the type of condition Type page. These fields will only show up for user data input, and pending on the problem dimension size, the data length could vary. For data requiring angle measurement, they must be in "radian" unit. The rotational DOF's (either rotational deformation or applied moment) can

only apply to model with shell or beam continuum element with such degree of freedom. Due to the expanded usage of the code, there are possible new items will appear as the analysis capability of the program expands.

Model>Condition>Directional Value>Shell Pressure/Source Position

This option is specifically for normal pressure or flux boundary conditions that involve shell element or for the 3D-line pressure/flux when a unique normal direction can not be identified without user specification. This locator is a point located at the pressure/flux side of the shell/line so the pressure/flux integrator can determine the normal orientation from the locator position. The positive pressure/flux is interpreted as direction pointing away from the locator position (like a ray). For the 3D line, the computed normal vector will lie on the plane formed by this locator position and the end points of the line. If this option is not specified for shell normal pressure loading, it is defaulted to be at location (0,0,0). For 3D-line, it is always necessary to set this locator position. This option is not necessary for shell or line with global pressure loading since in such case, the pressure components (Px,Py,Pz) are supplied by the user.

Model>Condition>Directional Value>Geometry Follower

By default, all integrated condition types such as pressure/flux are follower types if nonlinear geometric strains are used. Currently, all nodal values are not follower in nature that they are in the global direction sense. As a result, this check box is not necessary for nodal control, but is reserved for future use.

For all distributed properties such as pressure or flux, if large deformation (either Update Lagrangian or Total Lagrangian) is active, the follower nature of the property is automatically activated.

The Model/Condition Thermal Convective/Radiative Page

The thermal convective/radiative value page is specific for thermal boundary conditions.

The screenshot shows a software window titled "Condition Sheet" with a close button (X) in the top right corner. The window has several tabs: "Condition", "Geometry", "Time", "Type", and "Convection/Radiation". The "Convection/Radiation" tab is active. Inside the window, there is a "Name:" label followed by the text "T4:Convect:COT0". To the right of the name are two buttons: "New Type+Value" and "Update". Below this, there is a section titled "Thermal Convection Values". Under this section, there are two input fields: "Convection Coeff" with the value "1e-3" and "Ambient Temp" with the value "0". At the bottom of the window, there is a "New BC" button, a list box containing "test EdgeEdges_4 t(0,0;1,1 T4:Convect:COT0", a dropdown arrow, and "Prev" and "Next" buttons.

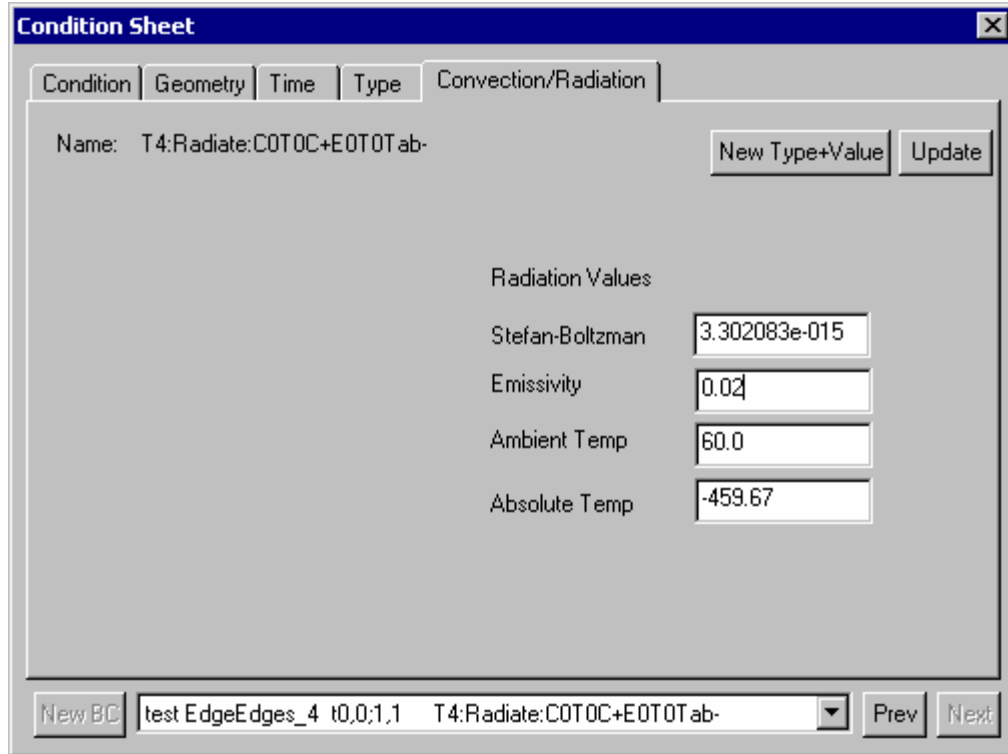


Fig. 4.14 The Thermal Convection/Radiation Page

The convective/radiative heat transfer is computed from the equation

$$k\nabla^2 T + g = \rho C_p (\dot{T} + v\nabla T) + h(T - T_c) + es(T^4 - Tr^4)$$

Where

k is the conductivity

g is the heat generation in the domain

ρ is the mass density

v is the convective velocity

h is the convective heat transfer coefficient

T_c is the convective ambient temperature

s is the Stefan-Boltzman constant

e is the emissivity coefficient

Tr is the radiative ambient temperature (relative to Kelvin absolute zero temperature).

The Model>Condition>Convection/Radiation>Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition>Convection/Radiation >New Type+Value

This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition>Convection/Radiation >Update

This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

The Model>Condition>Directional Value>Update/Save

This will updates the database of the current Thermal Convection/Radiation values page.

Model>Condition>Thermal Convection/Radiation>Thermal Convective/Radiative Values

These specify a convective or radiative boundary condition in the thermal problem. Depending on the type of the condition, only the convective or the radiative values are necessary.

The Model/Condition Surface Contact Condition

The surface contact value page is specific for contact boundary conditions.

The screenshot shows the 'Condition Sheet' dialog box with the 'Surface Contact' tab selected. The 'Name' field is 'T4:Radiate:COTOC+EOTOT ab-'. There are two buttons: 'New Type+Value' and 'Update'. The following options are checked:

- Selected geometry defines contact boundary, and
- Examine selections' associated nodes for contact
- Solid Continuum Contact
 - Friction Coefficient: 0
 - Penalty coeff. (0 for automatic): 0
- Thermal Contact
 - Friction to thermal energy conversion coeff.: 0
 - Penalty coeff. (0 for automatic): 0
- Electric Contact
 - Penalty coeff. (0 for automatic): 0
- Magnetic Contact
 - Penalty coeff. (0 for automatic): 0
- Apply movement to Analytical Geometry time scaled motion control, if movable. Time index is 0
- Skip other contact geometry set (faster check)

At the bottom, there is a 'New BC' button, a dropdown menu showing 'test EdgeEdges_4 t0,0;1,1 T4:Radiate:COTOC+EOTOT ab-', and 'Prev' and 'Next' buttons.

Fig. 4.15 The Surface Contact Page

The Model>Condition>Surface Contact >Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition>Surface Contact>New Type+Value

This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition>Surface Contact >Update

This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

The Model>Condition>Surface Contact>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 define 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.

The Model>Condition>Surface Contact>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.

The Model>Condition>Surface Contact>Solid Continuum Contact

When checked, solid contact detection is activated. The friction coefficient is used to calculate the friction force from the normal force between the contacting surfaces. Since the contact formulation is based on the numerical penalty method, the penalty value is automatically chosen if the Penalty Coefficient is left to be zero. In such case, the penalty value is automatically computed based on the solid's stiffness at the point of the contact. It is usually is a value about 3 magnitude higher than the maximum stiffness detected. However, if it becomes necessary to specify a more suitable weight penalty in numerical calculations, it should be entered here. If the penalty coefficient is a non-zero value, it refers to the relative magnitude to the default computed value. For example, enter a value of 0.1 will request the program to use only 1/10 of the computed penalty value. Usually, if a value is too low, some slight penetration could be observed. A higher value usually will have a better result, but at the cost of more numerical iterations, and lead to possible instability if the penalty value is too high.

The Model>Condition>Surface Contact>Thermal Contact

When checked, thermal contact checking is activated. The thermal continuity is enforced through the contacting face if solid deformation contact is detected. Optionally, 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 numerical penalty method, the penalty value is automatically chosen if the Penalty Coefficient is left to be zero. In such case, the penalty value is automatically computed based on the solid's conductivity, and it is about 5 magnitude higher than the maximum conductivity detected. However, it is sometimes necessary to specify a more suitable weight penalty in numerical calculation, and it should be entered here.

The Model>Condition> Surface Contact>Apply Movement to Geometry Time Scale Motion Control, if movable.

When the geometry condition specified is an auxiliary analytical geometry with possible movement control, this option will control the time scaled motion of the geometry using the time index specified. This is best used to specify contact situation such as a moving contact boundary such as squeezing/crushing of a solid in a mold pressing simulation. Note that the default time index control in the combined condition will control all other parameters, except the penalty coefficient.

The Model>Condition> Surface 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.

The Model/Condition Tie/Glue Condition

The Tie/Glue value page is specific for Tie/Glue boundary conditions.

Condition Sheet

Condition | Geometry | Time | Type | Tie/Glue

Name: G4:Tie:All New Type+Value Update

The first geometric selection is the master surface/side/node, and the rest of the selections are to be tied/glued to the master geometry.
 The master geometry is defined in geometry selection item (0 as first)

Tie all active DOF's (default)
 Solid Continuum Disp/Rotation or Fluid Flow Velocity/Vorticity/Pressure Tie/Glue
 Dir 1 Dir 2 Dir 3 Rot 1 Rot 2 Rot 3 Pressure
 Thermal Electro Magnetic

Use specified factor instead of those computed for geometric compatibility
 The ratio of the slave DOF to the master DOF is

Apply time/step factor to Tie/glue

Apply transformation to the slave geometry before processing tie constraint

	x	y	z	
Translation	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>	Rotation Angle (in degree, CCW about Rotational Axis is positive) <input type="text" value="0"/>
Rotation Origin	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>	
Rotation Axis	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="1"/>	

New BC G04 EdgeEdges_4 t0,0;1,1 G4:Tie:All Prev Next

Tie/Glue condition is often used to connect two separate geometric objects together. This is done by specifying a "master" geometric entity along with one or multiple "slave" geometry objects to be tied/glued. In 2D analysis, the tie geometry can be geometry side, element side, or node, and in 3D analysis, it can be geometry surface, element face/side, or nodes. 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 the geometric compatibility or an optional user tie factor. Note that if initially these objects are not in contact, the tie/glue condition will not be established and will be ignored. If there are any initial contact/penetration 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.

The Model>Condition> Tie/Glue >Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition> Tie/Glue >New Type+Value

This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition> Tie/Glue >Update

This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

The Model>Condition>Tie/Glue>Master Geometry Setting

The user must select at least two geometric entities for tie/glue definition. By default, the first selection is the master geometry, and the rest of the selections are tied to the master geometry. All nodes associated with the slave geometry (face/slide/node) are first extracted, and their relative geometric position with respect to the master geometry (could be either a face/side) are computed. If the slave node and the master geometry are initially in contact, the geometry compatibility is enforced using the initial geometry constraint through out the entire analysis with the optional time factor control.

The Model>Condition>Tie/Glue>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.

The Model>Condition>Tie/Glue>Ratio of the slave DOF to the Master DOF

This specifies the factor that will be applied on top of the geometric compatibility constraint, and it should not be used unless specific master/slave tie factor is desired. In the case that the master geometry is a node, the initial slave/master contact search is skipped, and this ratio (default to be 1.0) is applied to the slave/master tie relationship.

The Model>Condition>Tie/Glue>Apply time/step factor to Tie/Glue

This option will apply the time/step factor to the tie/glue relationship. Normally, it is only meaningful when the factor is either one (a fully geometric compatible tie/glue) or zero (a completely disengaged tie/glue). This could be used to control the release/engage of the tie/glue condition by assigning a Time/Step index with proper factor between 1.0 and 0.0.

The Model>Condition>Tie/Glue>Apply transformation to the salve geometry before processing tie constraint

This option will apply the specified translation/rotation to the slave geometry before the 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 to the master geometry region so a proper geometric tie constraint coefficients can be automatically computed.

The Constraint Equation Condition

The Constraint value page is specific for Constraint boundary conditions.

Condition Sheet

Condition | Geometry | Time | Type | **Constraint Equation**

Name: G4:Constraint:???

New Type+Value | Update

This condition casts a linear constraint among the specified nodal DOF's in the expression of $\text{Sum}(N_i D_i C_i) + h = 0$. C_i is the coefficient; N_i indicates the order count (zero as first) of the node in the geometry selections, and D_i is the DOF of the node. The constant h , if non-zero, is the nonhomogeneous terms of the constraint.

	N_i	D_i	C_i	h
Add/Update ->	0	dispy	-1	0

Current Values (select to edit)

0	dispx	1
0	dispy	-1

<- Delete Selected

The first DOF is the default dependent variable. The current setting is (zero as first) 0

Apply time/step factor to Constraint Equation (constraint vanishes when time_factor is zero)

New BC | G04 EdgeEdges_4 t0,0;1,1 G4:Constraint:???

Prev | Next

Constraint condition is used to construct an equational constraint among different nodal DOF's so a specific boundary condition can be enforced. For example, in the above constraint boundary condition, the node stored in the first geometry selection position is constrained by the equation

$$\text{Dispx} - \text{dispy} + 0 = 0.$$

Physically, this is equivalent to rotating the node by 45 degrees (using Local Coordinate controls), and specifies a zero normal deformation condition. The general formation of the constraint equation is

$$\text{Sum}(N_i D_i * C_i) + h = 0.$$

Where N_i refers to the selection count order in the geometric selection. The geometry selection items have to be finite element nodes or geometric vertices since the condition casts a constraint among the nodal DOF's. D_i is the nodal DOF specification, and can be selected from the pull-down list based on the available runtime-dependent solution variables. C_i is the coefficient of the DOF of the constraint equation. The constant term h , if non-zero, is the non-homogeneous terms of the constraint.

Internally, the constraint condition converts the equation by choosing a dependent variable and expresses it in terms of the rest of the DOF' variables and the constant term. By default, the first DOF is set as the dependent variable, and can be changed by selecting a different DOF as shown in the edit field.

Couple special notes about the constraint condition. If there are multiple constraints specified on a same dependent nodal DOF, the last one is always a dominating one. In addition, the independent DOF variable as specified can not be the dependent variable of another constraint equation. Finally, if a local coordinate system is assigned to a node, the constraint equation always refers to this local coordinate format

The Model>Condition>Constraint Equation>Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition>Constraint Equation >New Type+Value

This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition>Constraint Equation >Update

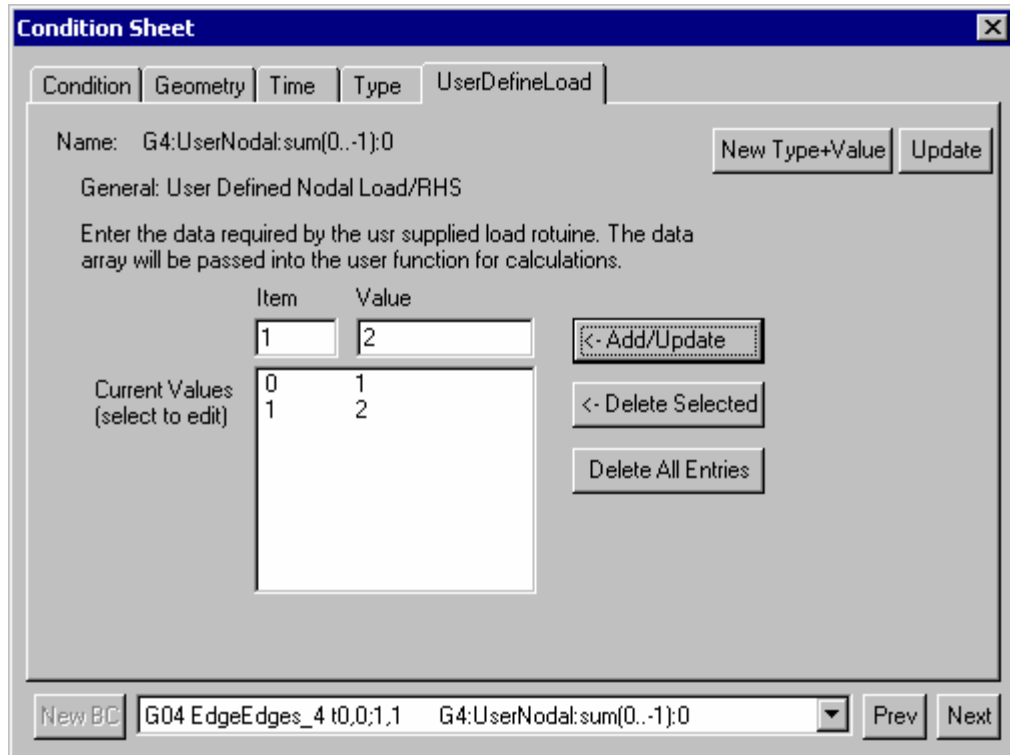
This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

The Model>Condition>Constraint Equation >Apply time/step factor to Constraint

This option will apply the time/step factor to the Constraint condition. Normally, it is only meaningful when the factor is either one or zero (a non-existent constraint). This could be used to control the release/engage of the constraint condition by assigning a Time/Step index with proper factor between 1.0 and 0.0.

User Defined Load/Flux/RHS/LHS Condition

The user defined load value page is specific designed to store any necessary load control data from the user so these data can be passed into the user routines during runtime for load calculation.



These user load data are stored in an array as defined in the list box. The array has the index range from zero to index n. If any one of the index data is missing, it is considered to be zero. In such case, the whole data array is expanded and stored in the model so during runtime, they are retrieved and passed to the user routines. For more details of the user defined load boundary condition, please refer to "AMPS User Application Interface Reference Manual."

The Model>Condition>UserDefinedLoad>Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition>UserDefinedLoad >New Type+Value

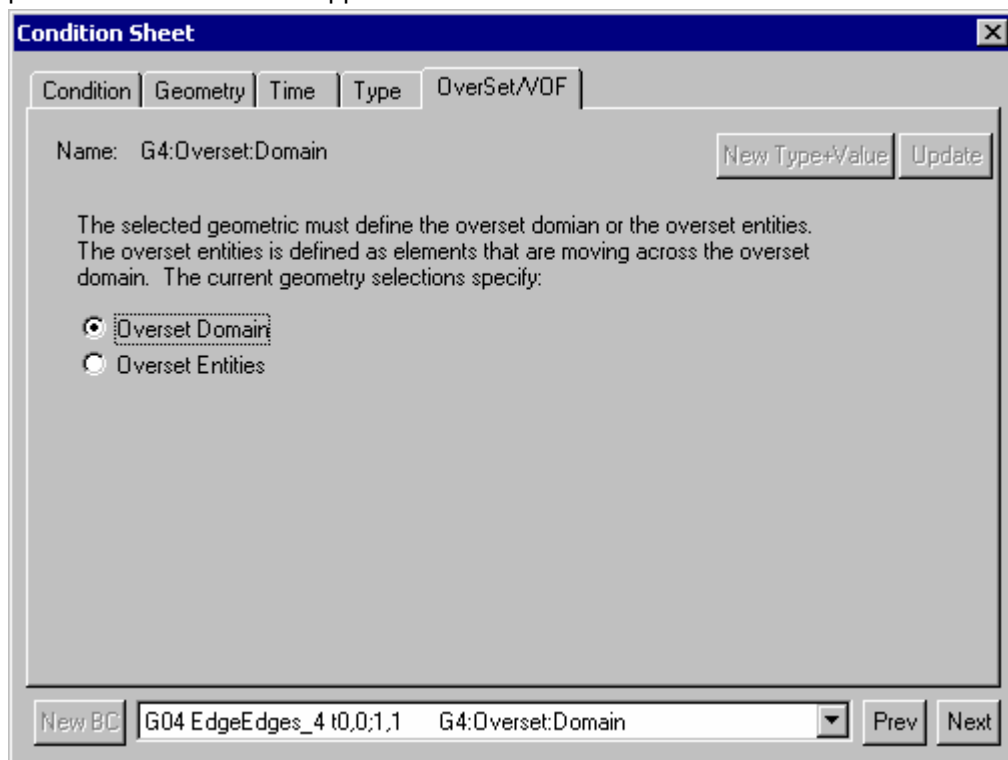
This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition>UserDefinedLoad >Update

This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

The OverSet Condition

The OverSet boundary condition is an advanced condition set that is mainly used for large deformation motion such as bullet penetrating through air or bubble rising in a pond type application. In such cases, a finite element group (OverSet Entity) is moving across another finite element group (OverSet Domain). The user must specify the OverSet type and select the proper geometry association. It can also be used as the traditional VOF (volume of the fluid) method specification for applications such as bubbling in water. In such case, the air bubble is the OverSet Entity, and the water is the OverSet Domain. The problem is extremely nonlinear due to the geometric constraint and fluid-solid interaction on the interface. To use this feature, please contact technical support for further assistance.



The Model>Condition>OverSet/VOF>Name

This name is the same as the one have been created in the condition Type page. It is displayed here for information purpose.

The Model>Condition>OverSet/VOF>New Type+Value

This will create a new Type Name and save the current data entered this page into the condition new Type Name condition set. This is a quick way of creating a new Type Name if the user has not created a new Type Name in the previous Type page.

The Model>Condition>OverSet/VOF>Update

This will save the entered data and updates the database of the current active Type Name as shown in the Name field.

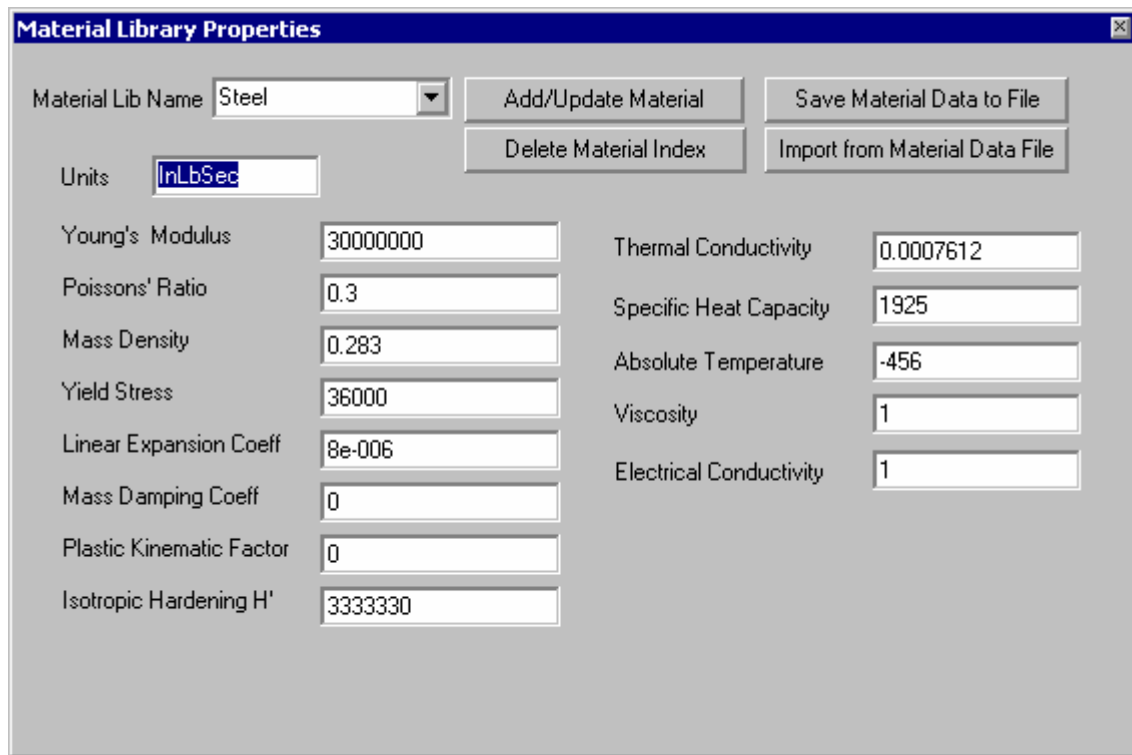
The Model>Condition>OverSet/VOF>Overset Domain || OverSet Entities

The user must specify whether the selected geometry is an OverSet Entity that moves across the OverSet Domain, or the OverSet Domain that form the region that the OverSet Entity will move through.

Model/Material Library Operations

The Model Define Material Library Menu

The Material library is created and maintained by this menu. SagreView has a built-in material library that is included in the original installation. By default, the name of the library is "material.lib". The user can use this material library editor to create/modify the library to tailor the material data.



Model>Define>Material Library>Material Library Name

The material data are entered here and must be given a name. The material database is stored by the name.

Model>Define>Material Library>Material Data

The user may specify any of the material items. Only data that will be used by a type of analysis needs to be specified here. Note that this data is used as a lookup table in the AMPSol/Define element behavior section.

Model>Define>Material Library>Apply/UpdateMaterial||Delete Material Index

These options allows the material data to be created /or edited.

Model>Define>Material Library>Save material data to file

This saves the up to date material library to the material.lib file. This is the file that is always used when AMPView look for a material information.

Model>Define>Material Library>Import from Material Data File

If the user has multiple material data file stored under different filename, this utility will read the information stored on the file, and then append them to the current material database. Note that it should be save back to the material database file so the future AMPView session will automatically use the newly merged material database.

Result Menu

The Results Menu

This performs the post-processing of the model results.

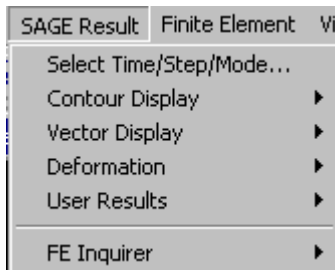
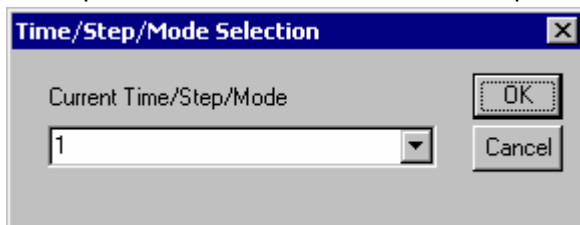


Fig. 5.1 The Model result Menu

Model Result>Contour Display>Select TimeStep...

This allows the user to select the time/step for result examination, inquiry, animation. A dialog box is presented with the available time/step selection as below.



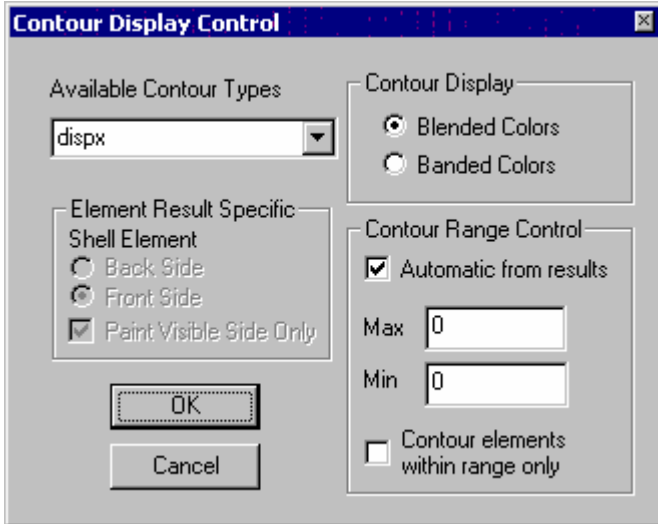
Note that this value shown could be either a time/step value, or a eigenvalue corresponding to a eigen analysis mode, or could be a load multiplier from an arc length analysis. It really depends on the nature of the output file being read in. Choosing this value will automatically update the corresponding nodal results to match the selected time/step/mode/multiplier.

Contour Display

Result Contour Controls

Model Result>Contour Display>Contour Type

This allows the user to select the quantity to be contoured from a dropdown list as show below.



The user may also elect to use fixed bands or interpolated colors for the contouring. The following types can be displayed in all their x, y, z components (note that additional items may be available based on different version of AMPSol processor):

Displacement	disp
Temperature	temp
Velocity	velx, vely, velz
Reactions/RHS summation	sum
Stress	sig
Strain	eps
Von Mises Equivalent Stress	sig_equiv
Tresca Equivalent Stress	Tresca
Principal Stresses	SigMax, SigMin, SigInt
Equivalent Plastic Strains	eps_plastic
Thermal Flux	tflux
Yield function	yield_func
Error Norm	error_norm
Nodal balance	sumX, sumY, sumZ

Note that **disp, vel, sum, tflux** are in the vector component format (x,y,z), **sig, eps** are in tensor format, and the rest are in scalar format. Specifically,

sig_equiv = $\sqrt{(3/2 \sigma_{ij}' \sigma_{ij}')}$ where σ_{ij}' is the deviatoric stress

SigMax, SigInt, SigMin these are the principal stresses. They are arranged in their value such that SigMax > SigInt > SigMin in value format (not in magnitude format)

Tresca = SigMax - Sigmin

eps_plastic is computed from the time step integral of $d\epsilon^P = \sqrt{(2/3 d\epsilon_{ij}^P d\epsilon_{ij}^P)}$

yield_func is the difference of the material yield criterion to the yield stress, e.g. in Von Mises, $yield_func = sig_equiv - \sigma_y$

error_norm this is an error distribution (square-root of sum of squares of energy error) based on a energy theory by comparing the finite element result to a estimated theoretical solution based on the equivalent-energy theory.

sumx, symy, symz these are the results from the equilibrium check. They represent the total net RHS at the end of the analysis. In stress problem, these are the total net forces on the nodes, and for diffusive heat transfer problem, the net flux flow at the node. For node without any boundary condition, the value should be close the error tolerance as specified, and normally is very near zero. For the nodes with the boundary condition, this reflects the net RHS value of the boundary node, or reaction from the boundary condition.

If the check box "Automatic from result" is specified, the contour range will be automatically selected based on the specified result type. Otherwise, select the desired minimum and the maximum range to be color contoured.

Model Result>Contour Display>Contour Elements Within Range Only

In situation that only a certain range of contour values are of interest, the user can check "Contour Element Within Range Only", and the contour shading will show only on the elements that contains nodal results that are within the range. Those out of range will be hidden. To show all elements again, please use the "Finite Element/FEGroup/Display/UnHideAll" option to reset the hidden elements (since they are internally temporarily marked as hidden element without any contour).

When examining shell element's result, all properties show are referring to the shell's mid-plane result, or can be considered as approximately the average of the result across the shell's depth. To examine the shell elements' top or bottom surface's results, use the optional "Shell/Element Contour" menu.

Model Result>Contour Display>Show Contour

This initiates the contour plot.

Model Result>Contour Display>Show Model Only

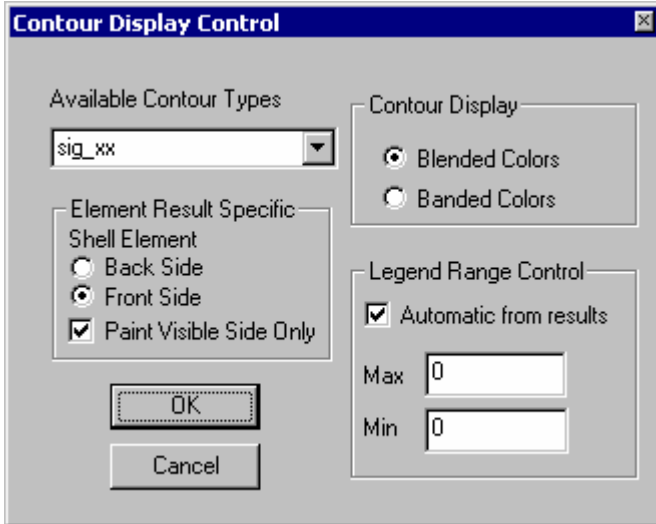
This removes the contour plots from the model.

Model Result>Contour Display>Time/Step Animate

When there are multiple time step results, this menu is active and can be used to activate color contouring in steps.

Shell/Element Contour

Model Result>Contour Display>Shell/Element Contour

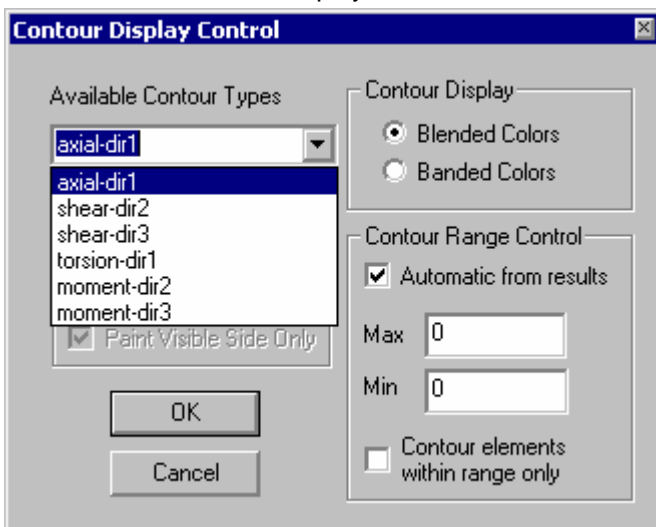


This contouring control dialog is specifically designed for shell element-based result display. For shell element, the front/back side result display will contour the result on the selected side, and are only visible from the side specified, and invisible from the other side in order to give the user the idea which side he/she is viewing

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. In the AMPS version 1.74 and above, 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 ModelResult/FE_Inquire/by_Element inquiry dialog to find out the exact value of each element.

Beam/Rod Contour

Model Result>Contour Display>Beam/Rod Contour



This contouring control dialog is specifically designed for beam/rod element-based result display. If there is no beam/rod element in the model, this menu item can not be selected. The available beam/rod results will be shown in the pull-down list. For beam/rod elements, the

contour result magnitude is plotted in scale in the local axis 2 or 3 direction as defined in the beam property orientation vector. This is for easy identification of the result, and to ensure the result display can be examined based on the relationship to the major/minor axis orientation. For axial or torsional result, it is plot on the local axis 2 direction, and for shear-dir2/dir3 and for moment-dir2/dir3, their magnitude variation will be plotted along the beam axis direction and in the corresponding 2 or 3 local axis direction. Note that for all element-based result, you can also inquire each element's corner value using the ModelResult/FE_Inquire/by_Element inquiry dialog to find out the exact value of each element. When 2D-beam element is used, the result is also treated as a three-dimensional beam and shown in the corresponding local 1, 2, and 3 orientation.

The beam/rod contour is shown with shaded plot drawn on the corresponding local axis direction. Sometimes, it is easier to turn of the shading and show only the outline if necessary. In certain situation when the contour plot showing the relative magnitude is too small to observe, the user can use the ModelResult/Vector_Display/Set_Scale to change the magnitude size. The default ratio is one, which will make the largest element result value shown as roughly corresponding to about 5% of the screen size.

Vector Display

Vector Display

This performs the post-processing of the Model Solver results.

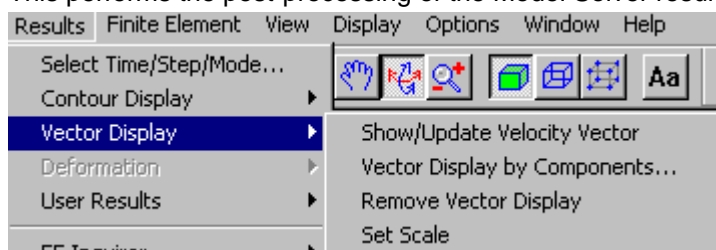


Fig. 5.1 The Model result Vector Display Menu

Model Result>Vector Display>Show/Update Velocity Vector

This allows the user to display results containing velocity information.

Model Result>Vector Display>Vector Display by Component

It is sometimes convenient to display the result in the flow vector format even there is no velocity result available. This menu allows you to specify each vector component and to perform an arrow vector display. For each node, an flow vector arrow will show the relative magnitude of the flow magnitude.

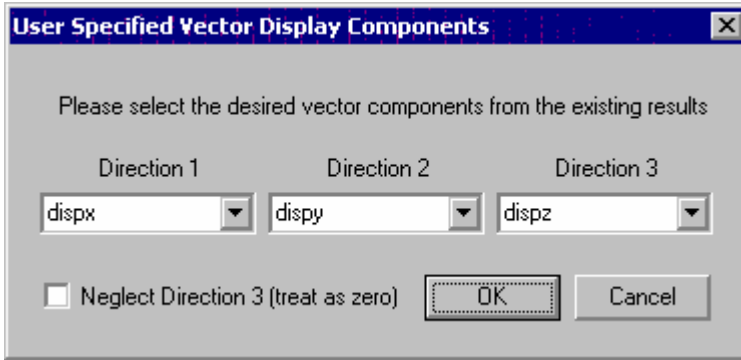
Model Result>Vector Display>Remove Vector Display

Clicking this menu will remove vector arrow display from the screen.

Model Result>Vector Display>Set Scale

This menu allows the user to set the scale of the vector arrow length. Normally, the longest vector arrow is about 5% of the screen size, and this vector size is interpreted as the unit length of the vector arrow display. The program will apply the scaling to this unit vector length, if it set.

Vector Display by Component



This dialog box allows the user to enter the vector components necessary to construct a user specified vector display. Three pull-down lists are available to select each vector component. If the vector component in the 3rd global Z-direction is not necessary or not meaningful, the check box "Neglect Direction 3" check box should be checked. For each node, an vector arrow will show the relative magnitude of the specified vector.

Deformation

The Results Deformation Display Menu

This allows the user to choose to include change of shape calculated by the solver.

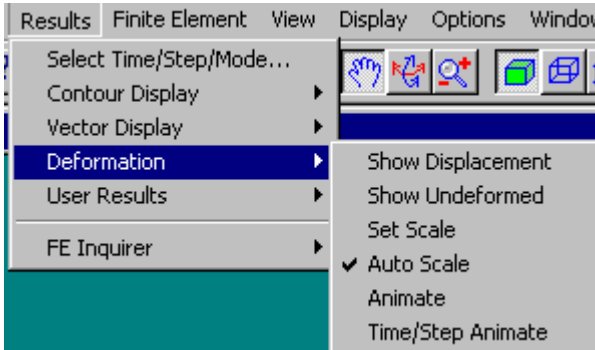


Fig. 5.2 The Model Result Deformation Page

Model Result>Deformation>Show displacement.

This will display the deformed shape of the solid. If the "Auto Scale" is on, the deformation scale is automatically computed such that they will show visible change on the screen.

Model Result>Deformation>Show undeformed

This shows the finite element model on the original solid.

Model Result>Deformation>Set Scale

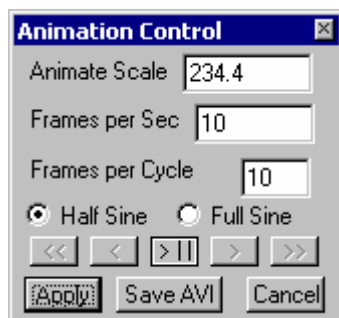
The user can change the scaling factor at any time. A deformation scale control dialog will be show for setting.

Model Result>Deformation>Auto Scale

When this flag is on, the program will automatically scale the deformation pattern such that the displacement is scaled to about 5% of the screen size.

Deformation Animation

Model Result>Deformation>Animate



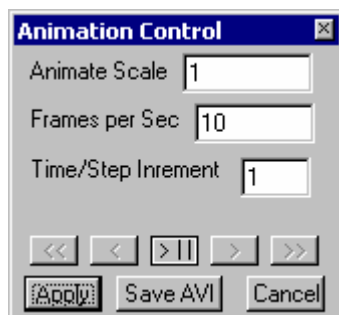
This animates deformation of the selected step result on a continuous basis. The user can choose from a dialog box either to alter the scaling factor or to terminate the animation, or to apply a full/half sinusoidal variation of the deformation pattern for visual effect. When the Full Sine option is used, the current deformation will oscillate from scale factor of -1 to 1 , while the default Half Sine option varies from 0 to 1 scaling.

This animation process is designed as a multi-threaded task, so the user can continue to do other graphic operations such as changing the color contour type or rotate the scene for a better view while this animation is on. If the animation is too fast, the user can limit the refresh rate "Frame per Sec" to a lower rate, but the animation will not be as smooth.

The buttons \ll , \lt , \gt , $\gt;$, \gg are for animation fast backward, backward, pause/play, forward, and fast forward correspondingly. Clicking the pause button will activate other motion control buttons, and the subsequent clicking of the pause button will resume to normal animation.

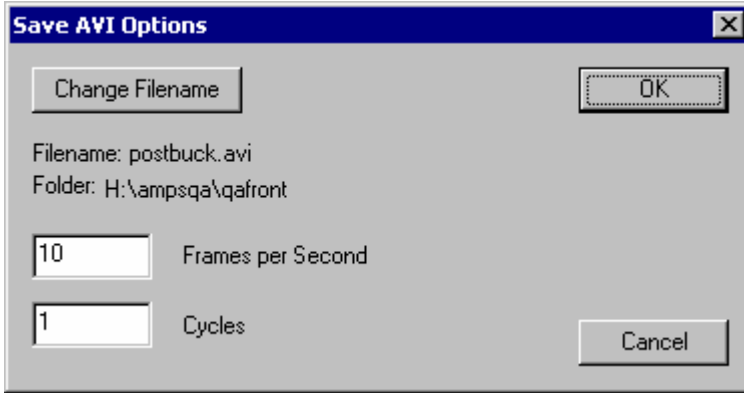
When the model is very large, it is useful to turn off the face display to speed up the animation and uses only the element sides and nodes. This is useful for examining result since when the color contouring of the result on FE sides and nodes is still available. Reducing the "Frames per Cycle" will also increase the animation speed.

Model Result>Deformation>Time/Step Animate



This animates deformation from the available time/step results on a continuous basis, as compared to the single step deformation scaling animation. The user can choose from a dialog box either to alter the scaling factor or to terminate the animation, or to select the step increment number in order to speed up the animation when lots of time steps results are produced. When the Full Sine option is used, the result animation will start from step 0 to step n at an increment as specified, then decrease back to step 0 with the decreasing steps entered. The Half Sine will just repeat from step 0 again, instead of gradually decreasing the result step animated.

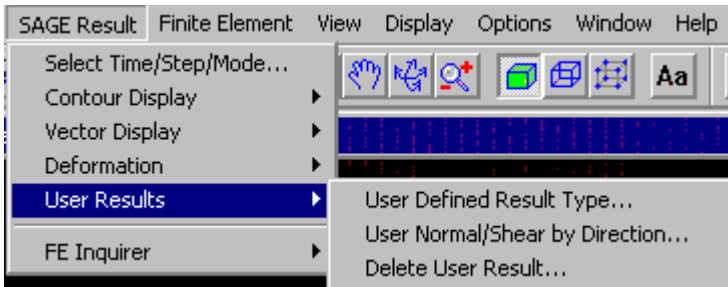
The Save AVI option will capture the animation frame and convert them into Windows' movie AVI animation for later viewing. When this button is pressed, an optional dialog box will pop up for AVI capturing setup for either a single or multiple animation cycle capturing.



The AVI format contains a wide spectrum of compression methods pending on the installed AVI COM tools on the Windows system. Usually, the "Microsoft Video 1" 75% compression is a reliable choice.

User Result Type

User Results



This menu provides tools to create additional results that the user may create from the current available results. The "User Defined Result Type" allows a symbolic math type operation on the existing result variables. The "Normal/Shear by Direction" allows the user to create stress result on a specific direction so they can be used for additional contouring or inquiry purpose. Once the additional results are created, they will be in the active section till the user deletes it from the memory by using the "Delete User Result" menu.

Customized User Result Type

This option allows the generation of a new user defined result type by specifying a mathematical expression of how to compute the value from the existing result types that are created by AMPSol calculations. The user can create a new result type using the provided mathematical expression and the existing result type names. After the result type is created, it can then be used in contouring, node/element result inquiry, plotting, or integral inquiry.

Currently, the support expressions are:

+	Addition
-	Subtraction
*	Multiplication
/	Division
%	Mod (remainder)
^	Power
(Left parenthesis
)	Right parenthesis
sin()	Sine function
cos()	Cosine function
tan()	Tangent function
abs()	Absolute function
ln()	Natural log
exp()	Natural power
atan()	Arc-tangent
sqrt()	Square root
0..9	Numerical numbers
e	Scientific number (e.g. 1.e-5)

For example, the deformation magnitude can be defined as

$$\text{disp_mag} = \text{sqrt}(\text{dispx}^2 + \text{dispy}^2 + \text{dispz}^2)$$

Currently, only five lots of user defined variables are available, but they can re-use or deleted.

User Result Type>Generate/Compute New Result Type for all Nodes

Pressing this button will create a new result item base don the result expression give. It is suggested that the user test the result expression by pressing the "Test/Evaluate New Result for the Specified Node" to check the validity of the expression.

User Result Type>Delete the Specified user Result Type

Delete the already specified result item, if exist.

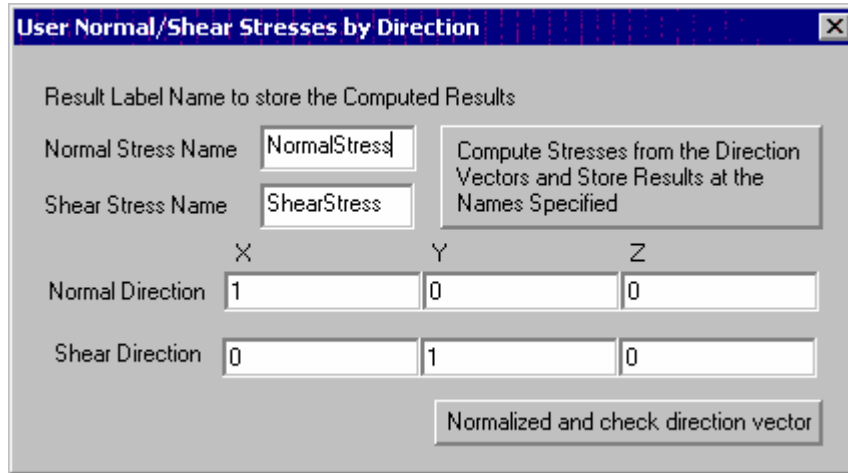
User Result Type>Retrieve the Expression of the User Result Name

If the specified result type name is valid and exist, pressing this key will retrieve the original result expression and put the numerical expression into the equation field at the lower portion of the dialog box edit field.

User Result Type>Test/Evaluate New Result for the Specified Node

Usually, after enter the numerical expression in the equation edit field at the bottom of the dialog box edit field, it is recommended to test the expression by checking the numerical result at the node user specified. This is to ensure a correct expression before the program compute and generate the new results for all nodes.

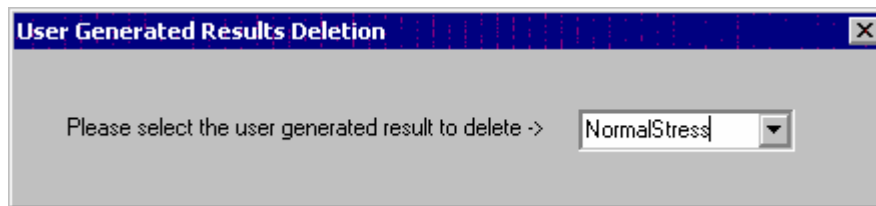
Normal/Shear Stress by direction



This control allows the user to generate normal and shear stress at the direction that of interest in interpreting the result. The default result names generated are "NormalStress" and "ShearStress". The normal stress direction is computed on the specified Normal Direction, and the shear stress is computed on the Shear Direction as entered. Before the normal and shear stresses are computed, both direction vectors are first normalized, and the Shear Direction vector is adjusted to make sure it is perpendicular to the normal direction. The user can examine this normalization process by clicking the "Normalized and check direction vector" to see the new direction vectors to be used in the calculations.

After the direction vectors are entered, clicking the "Compute Stresses..." button will start the generation of these two additional stress results to be used in contouring or inquiry.

Deleting Generated User Result



This menu allows the user to delete the user-generated results after they are no longer needed. This is meant to conserve the limited user result slots initially allocated by the system.

FE Inquirer

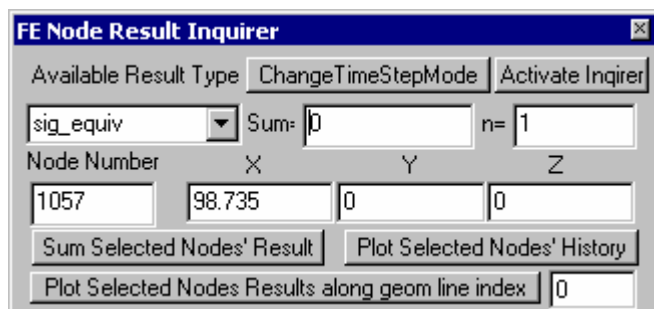
FE Inquirer by Node

This option allows the user to select a node and display type. Its value will be displayed in a dialog window as shown. For more information about the Result Type, please refer to the descriptions in the Contour display session.

While the inquirer is active, as the cursor move across the model, the nodal result is automatically retrieved. The information includes the node number, the coordinate values, and the associated value of the selected result type and time/step. If the user assigns the mouse to

other pointing function such as rotation, the inquirer becomes inactive sleep mode. It can be activated again with the Activate/inquirer button.

Another good use of the inquirer is to add the result value of different nodes. To use this function, first switch the mouse function to the FE entity selection, then select the nodes of interest, then press the button of "Sum Selected Nodes' Result". This is useful for user to find out data such as the total reaction on a fixed face, or the total heat flow from the convective boundary condition, or merely to sum the nodal results.



Plot Selected Nods' History

The FE Inquirer is also used to produce nodal time/step history plots. The user first selects a node or nodes using the node selection control. The node selection control may be found in the Selection>FE select>node only option or its icon. Then the user selects a desired type from the Result Type pull-down list. Finally, the plot Selected Node's History is pressed in the FE Inquirer. This produces a time/step history plot for the node with the selected result type. The history plot can be deleted and a plot can then be repeated by selecting another result variable or by selecting another node. If you select multiple nodes by depressing the shift key, these nodes' history will be shown as multiple curves with different colors.

Plot Selected Nodes Results along geom Line index

The FE Inquirer can also generate a nodal result plots along a user define line based on the distance from the line origin. The nodal coordinate is first projected to this geometry line by the closest position to the line, and the distance of this projected point to the line origin is computed as the horizontal coordinate of the history plot diagram. If the geometry line ID specified does not exist, the user is prompted to enter the required information to define the analytical geometry line. This process is repeated for all nodes selected and the current time/load step nodal result values are plotted along based on their distance from the line's origin.

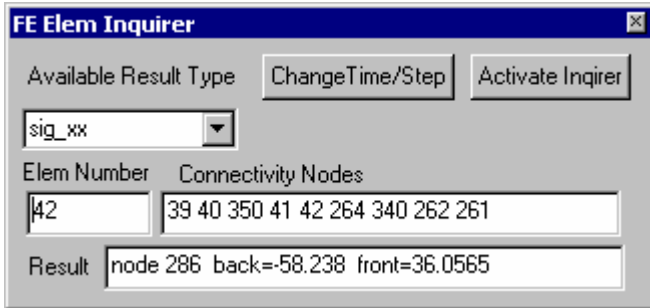
FE Inquire by Element

This option allows the user to inquire an element's connectivity and, if available, the result closest to the point under the cursor.

While the inquirer is active, as the cursor move across the model, the element's connectivity nodes are automatically retrieved. The nodes defining the current element is displayed in the dialog box, and the node number is also shown graphically next to the node's position for easy inspection. If the user assign the mouse to other pointing function such as rotation, the inquirer becomes inactive sleep mode. It can be activated again with the Activate/inquirer button.

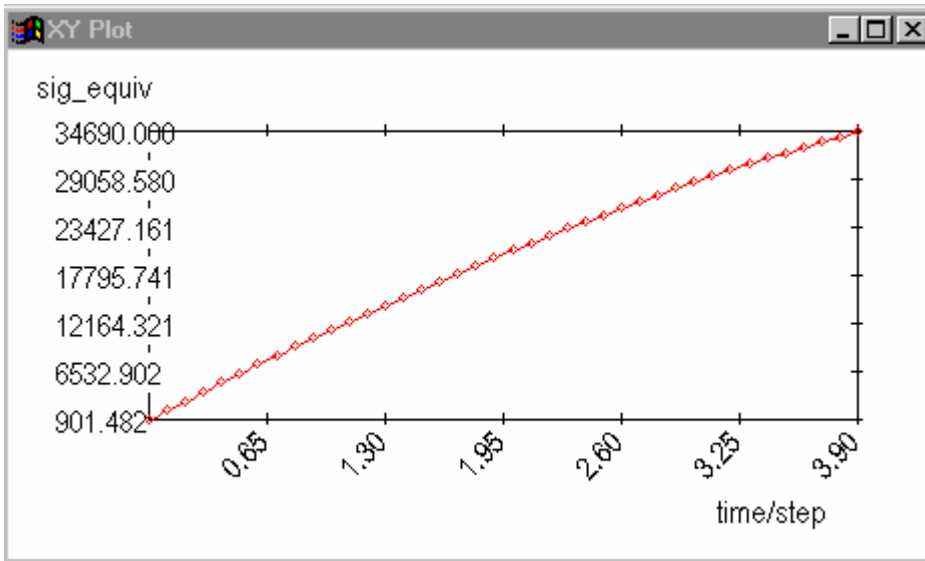
When there are element-based results available, the "Result" box will show the result closest to the cursor position, usually at the corner node area. Note that element-based results could be discontinuous across the element boundary, as compared with node-based results that are shared across the element connecting to that node. Typically, these element-based results are at the integration location without any additional extrapolation. For beam/rod element, the result type is their end forces (local forces and moments). For shell element, the results are

their nodal end forces (global forces and moments), and the closest integration point result for the shell's front and back face. When contouring the result using the default "Contour Type" option, the shell's mid-plane results are shown. Using this element inquiry option will allow you to see the results of the front and the backsides of shell element analysis.



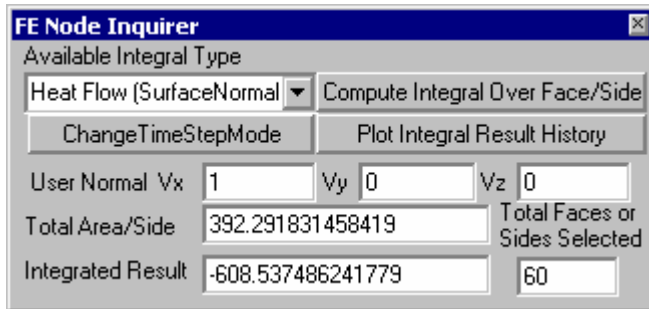
Inquirer Time Plot

The FE Inquirer is also used to produce nodal time/step history plots. The user first selects a node using the node selection control, and selects a desired type from Result Type pull-down list. Then the plot Selected Node's History is pressed in the FE Inquirer. This produces a time/step history plot for the selected result type. The history plot can be deleted and a plot can then be repeated by selecting another result variable or by selecting another node. If you select multiple nodes, then these nodes' history will be shown as multiple curves with different colors.



FE Inquirer by Integral

This option allows the user to perform an integral calculation over the specified surface (3D) or side (2D). To perform the integral calculation, specify the integral type, then select the geometry surfaces or edges, or element faces or sides where integration is desired.



FE Node Inquirer

Available Integral Type
Heat Flow (SurfaceNormal) Compute Integral Over Face/Side

ChangeTimeStepMode Plot Integral Result History

User Normal Vx 1 Vy 0 Vz 0

Total Area/Side 392.291831458419 Total Faces or Sides Selected

Integrated Result -608.537486241779 60

Two special integration calculations are provided. 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. The second special heat flow integral uses the user-specified normal instead of the surface normal. The resultant integral is the heat toward the direction of the user normal. For the rest of the result type integral, it simply integrated the result over the specified region.

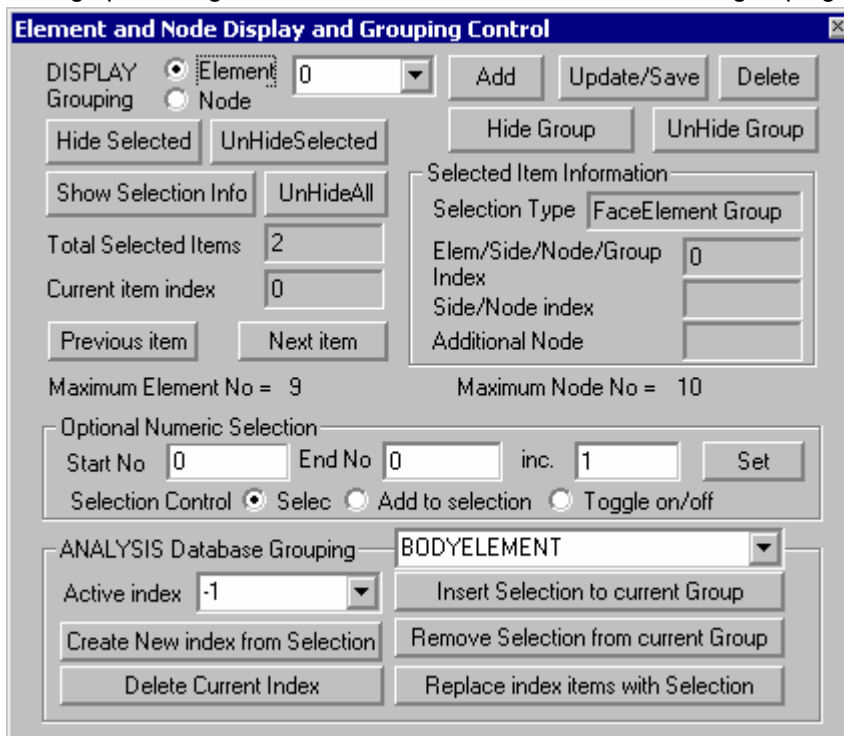
If multiple-step results are available, "Plot Integrated Result History" will also plot the integrated result against time/step to give a history diagram for examination.

Finite Element Menu

FE Group/Display

Selection>FE Group/Display

Bring up a dialog box for finite element nodes and elements grouping and display control.



Element and Node Display and Grouping Control

DISPLAY Element Node 0 Add Update/Save Delete

Grouping Node Element

Hide Selected UnHideSelected Hide Group UnHide Group

Show Selection Info UnHideAll

Total Selected Items 2

Current item index 0

Previous item Next item

Selected Item Information

Selection Type FaceElement Group

Elem/Side/Node/Group 0

Index

Side/Node index

Additional Node

Maximum Element No = 9 Maximum Node No = 10

Optional Numeric Selection

Start No 0 End No 0 inc. 1 Set

Selection Control Selec Add to selection Toggle on/off

ANALYSIS Database Grouping BODYELEMENT

Active index -1

Create New index from Selection Insert Selection to current Group

Delete Current Index Remove Selection from current Group

Replace index items with Selection

Finite Element>FE Group/Display>Display Grouping

The radio selection button selects the "Node" or "Element" type to control

Finite Element>FE Group/Display>

HideSelected/UnHideSelected/ShowSelectoinInfo/UnHideAll

After making a graphic element or node selections, clicking the hide button will automatic remove their's appearance, or to show index info/unhide the selected index group. UnHideAll button can be used to show all hidden FE items.

Finite Element>FE Group/Display>Add/Update/Save/Delete

After select the elements or nodes graphically, clicking the Add button to add a new group to the database. The index display will increase by one to show the index used in storing this display group. The Update/Save button will replace the current shown index group with the selected items. Delete will delete the current database index.

Finite Element >FE Group/Display>HideGroup/UnHideGroup

Hide or unhide/display the element or node group items stored in the current database index.

Finite Element >FE Group/Display>Show Selection Info/Previous Item/Next Item

If there are highlighted selected items, clicking the "Show Selection Info" will display the total items in the selections, their index value or element/node/Face/Side index. This is strictly for information purpose. If there are more than one items selected, clicking "Previous" or "Next" item button will cycle through selection list and display the proper info.

Finite Element >FE Group/Display>Optional Numeric Selection

Instead of using graphic mouse selection, the user can enter the items desired by using this numeric selection by specifying range of "Start No" and "End No", and optionally specifying the increment during this range calculation. Press the "Set" button to activate the selection highlight for visual inspection and for later hiding or grouping function. This numeric selection method is useful when you want to precisely select a range of elements or nodes.

Finite Element >FE Group/Display>ANALYSIS Database Grouping

AMPView maintains a geometry database for geometry items such as solid body, face, edge, and vertices selection. The geometry database relates the geometry to the element, element face, element side, or nodes. This database is automatically created within the AMPS/AMPSolid mesh generator and is shown here for user reference. These insert/remove/replace/create/delete geometry database button can also served as a tool for the advanced to modify these geometric relation to the finite elements and nodes. Sometimes, if an external mesh is imported into AMPView, these geometry database may not exist, and these database grouping tools can be used to create the elements and nodes relationship to the geometric entities.

Currently, the geometry database entities used in AMPS system are:

BODYMATERIAL

Assigns the association of the 3D elements' material type

BODYELEMENT

Relate the association of the 3D elements to the geometric solid entity

FACEMATERIAL

Assigns the association of the 2D/shell elements' material type

FACEELEMENT

Relate the association of the 2D/shell elements to the geometric face entity

FACEELFACE

Relate the association of the 3D elements' face to the geometric face entity

EDGEELSIDE

Relate the association of the 2D/shell elements' side to the geometric edge entity
 FACENODE

Relate the association of the FE nodes to the geometric face entity
 EDGENODE

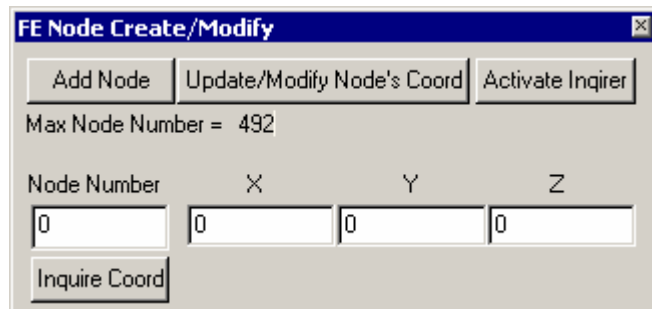
Relate the association of the FE nodes to the geometric edge entity
 VERTEXNODE

Relate the association of the FE node to the geometric vertex entity

Create/Modify/Inquire Nodes

This menu allows the user to create/modify/inquire the node, also informs the user of the maximum node number used so far. To create/change the node, enter the node number and coordinates desired. Sometimes when the mouse function is switched into other function, it is necessary to "Activate Inquirer" to attach the mouse function back to this dialog control.

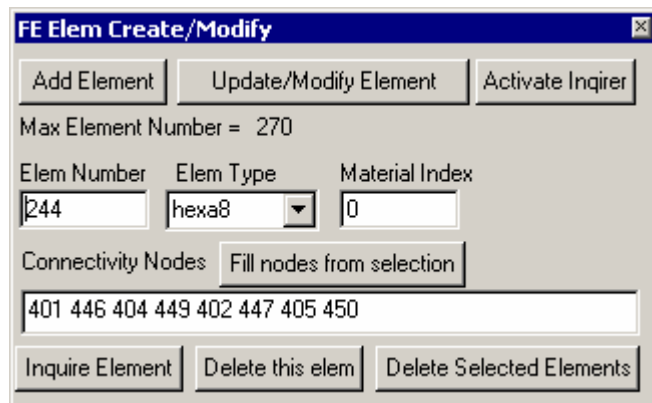
Since it is not safe to delete node, the unused nodes will only be deleted internally when "Optimize Mesh" is performed after checking the nodes are not used and are safe to delete.



Create/Modify/Inquire Element

This menu allows the user to create/modify/delete/inquire the element, also informs the user of the maximum node number used so far. To create/change the element, enter the node number of the element. Sometimes when the mouse function is switched into other function, it is necessary to "Activate Inquirer" to attach the mouse function back to this dialog control.

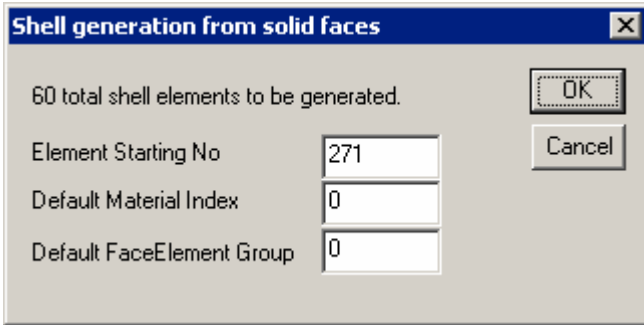
When creating/modifying the nodes, the user can enter the node numbers desired, or use the node selection mouse function to selected the nodes in the sequence forming the connectivity of the element, and click the "Fill node from selection" button.



Coat Shell Elem on the Selected Elem Faces

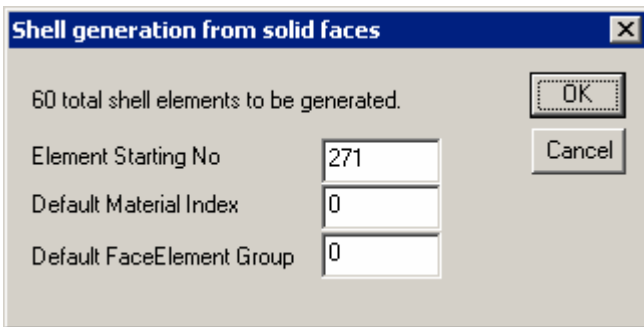
This menu allows the user to create the 3D shell element on the solid elements' face. Using this option, the shell element is created on the pre-selected solid element face and will share the same nodes with the underneath solid element to ensure tight continuity in the constitutive

behavior. The user has the choices of changing the shell elements' starting number and the material property group, also the geometric association of the FACEELEMENT group if desired.

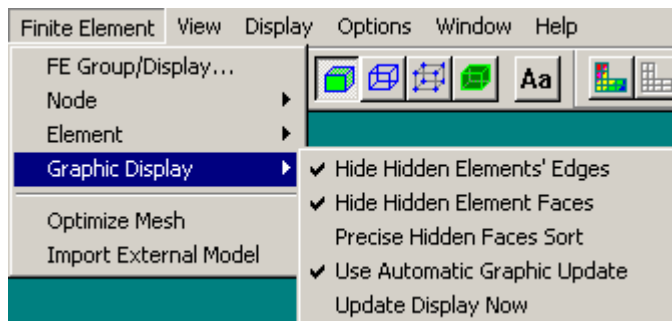


Coat Beam/Rod element on the Slected Element Sides

This menu allows the user to create the 2D/3D beam or rod elements on the existing element side or geometry edges. Using this option, the beam/rod element is created on the pre-selected edge will share the same nodes with the existing 2d/3d element to ensure tight continuity in the constitutive behavior. The user has the choices of changing the elements' starting number and the material property group, also the geometric association of the LINEELEMENT group if desired.



Finite Element Graphic Display



Finite Element>Graphic Display>Hide Hidden Elements' Edges

When this option is checked, the element's side will not be shown if it is a hidden element. This option is useful if very complex elements exist in the model and the user desire a different option of displaying the hidden elements.

Finite Element>Graphic Display>Hide Hidden Elements' Faces

When this option is checked, all elements' internal sides that are shared with other elements are hidden. It is mainly used to show only the solid surface without showing the internal faces. This option is useful when checking whether there is an internal gap or void in the solid

elements. It is also useful in large model animation since the internal invisible faces are removed from the view scene for faster display. When it is activated the first time, an internal database will check these internal faces and hide them after sorting, for large model, this may cause a little delay in graphic response. However, subsequent operations will no longer need this sorting. In AMPS version 1.72 and above, a fast hashing table searching algorithm is used to quickly find out these internal faces, but may still last for couple seconds if a large model is being viewed.

Finite Element>Graphic Display>Precise Hidden Faces Sort

The menu is an obsolete control since all models are now processed through a precise hidden faces sort since a very fast sorting algorithm is used and examining millions of elements only take few seconds.

Finite Element>Graphic Display>Use Automatic Graphic Update/Update Display Now

When this option is checked, if any element or node is modified, the graphic will be automatically updated. However, in situation of a very large model, it may be desirable to disable this function during the element or node editing/modifying process, and manually update the graphics using the "Update Display Now" option.

Finite Element>Graphic Display>Update Display Now

Clicking on this menu will force the program to regenerate all graphic display and refresh the screen as if it has just been freshly loaded. The main use of this control is to overcome situation when a corrupted display may occur due to poor graphic card's update ability or any unexpected runtime error.

Optimize Mesh

AMPView will try to optimize the finite element ordering by reordering the nodes and elements using a customized graph optimization method, and then remove the gap in the order, if any. If there are dummy nodes that are not used in the elements' connectivity, the user will be prompted to either delete them, or to attached them at the end of the model. It is recommended to always optimize the mesh for medium and large problem since the cost of the optimization is fast, and the saving in solution time could usually be as high as 50% or more. If the model contains nodes that are not used in any element reference, they will be removed after the user confirm that these nodes are not necessary.

AMPView uses an advanced graph traversal scheme to optimize the node and element numbering scheme so they will produce both the minimum frontal width and the smallest direct/profile solver storage requirement. Usually, one pass of the graph searching will produce the optimum ordering, and subsequent optimization will not produce any improvement. More information of the graph traversal theory can be found in "Fundamental of Computer Algorithms," E. Horowitz and S. Sahni.

Import External Model

This menu allows the user to import another AMPSol model into the current active model in AMPView. The new model is attached to the existing model by renumbering their nodes and elements after the existing ones. Since there are geometric entities associate with the AMPSol finite element model, these data are also appended after the existing one. To avoid conflict with the current model definition, only the mesh-associated information is merged into the current model, and the external model's material and boundary condition are neglected.

View Menu

The View Menu

The view menu allows the user to select the details of the graphic components



Fig. 6.1 The View Menu

These menu items are used to rotate, zoom, pan the view camera, and they have their associated icons for more convenient activation. There are several preset camera view angles predefined for common view settings.

To use the view camera orbit/zoom/pan feature, hold the left mouse button, and move the mouse's cursor across the screen. This will trigger the camera rotation, panning and zooming function.

Display Menu

The Display Menu

This menu allows the user to choose some display controls.



Fig. 7.1 The display Menu

Display>Orthographic, Perspective

The user can change whether to use an orthographic or a perspective display. The default setting is to use the perspective display with depth effect.

Display>Display Lists

On some machines with high end display memory controls, it is much faster to store the OpenGL data in a display list memory buffer and then, as the user refresh the screen, just to reuse these objects from the display list buffer. The user can compare the difference and select this option if it helps in displaying large models.

Display>Smooth View Transition

When this option is on, when the user change from one view camera angle to another one, the program will use a transitional effect to rotate the view from the old position into the new position. Using such effect, it is usually easier to follow the geometry details of the model.

Display>Zoom to Selection

When this option is on, the selected entities (elements, nodes, face, sides) will be automatically placed in the center of the screen and magnified. This option is useful when the user is reviewing a model, and prefers the program to automatically change the view and zoom into the geometry selection corresponding to the items being reviewed. For a very large model, sometimes this automatic zooming may take some time, and can be optionally turned off, if necessary.

Display>Zoom+Rotate to Selection

Other than just zoom to the selected items, if this option is used, a proper camera angle will be computed for the best viewing angle to these items. Since some selections contain only nodes or items are hidden inside the body, the computed angle may not be the optimal view angle for some cases.

Display>Transparent View

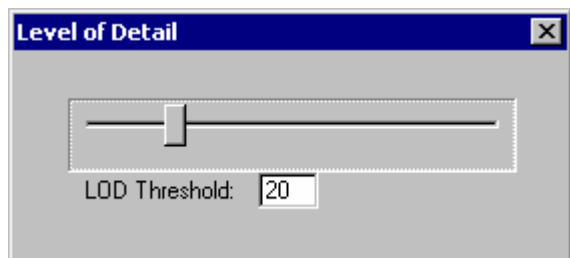
Sometimes, it is difficult to identify the object/entity buried in the model (such as internal faces, side, etc). Normally, it is necessary to hide the exterior elements so the inner elements faces/side can be selected or examined. The transparency is another convenient way of identifying such selection. If the option is on, the user can examine the internal semi-transparent object to ensure whether they selected or not, or to examine their results. This option is also very useful when it is used in Selection Filter (see Selection/SelectionFilter) control to confirm the Depth Selection operations.

Display>Level of Detail

To speed up the display of some extremely large model, the "Level of Detail" option can be constraint can be used. To use this option, activate this selection, and the desired level of details in the "Level of Detail Options" slide bar.

Display>Level of Detail Options

This will show up a slide bar to control the selection of the desired level of detail.

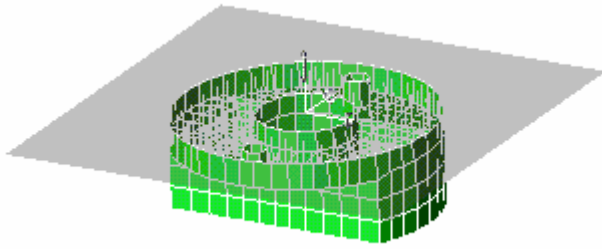
**Cutting Plane****Cutting Plane Menu**

These cutting plane toolbar icons allow the user to insert/define/translate/rotate/delete a cutting plane.



The cutting plane passes through the centroid of the solid model and is normal to the screen. All parts closest to the user are eliminated. To be precise all elements cut by the slicing plane are hidden. Contours are then redrawn on all the faces. The user may also display a grid on the cutting plane. Multiple cutting planes can be specified in order to zoom in to the desired

region. Details of the display controls can also be found in Options/Visibility/Cutplanes/cutlines/cutgeometry



There are three preset cutting planes, XY, XZ and YZ cut planes. After clicking on each icon, the cut plane will show up. By clicking the icon again, the cut plane will be removed. Three cutting planes can all be active. For each cutting plane, there is a tri-axis cut plane translation arrow along the cut-plane normal direction, and two cut-plane rotation control arrows. By holding the mouse left button while pointing the cursor at the either the rotational or translational arrow and move the mouse, the cut-plane can be placed into the desired orientation.

Options Menu

The Options Menu

This menu allows the user to control the display of the F.E. geometric entities such as elements and nodes

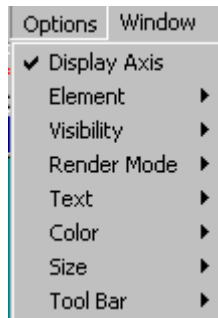


Fig.8.1 The Options Selection Level Menu

Options>Display Axis

When this option is turn on, the Cartesian coordinate axes are displayed at the origin. Note that for easy manipulation and identification, a mini axes display is always available at the upper right corner.

Option Visibility

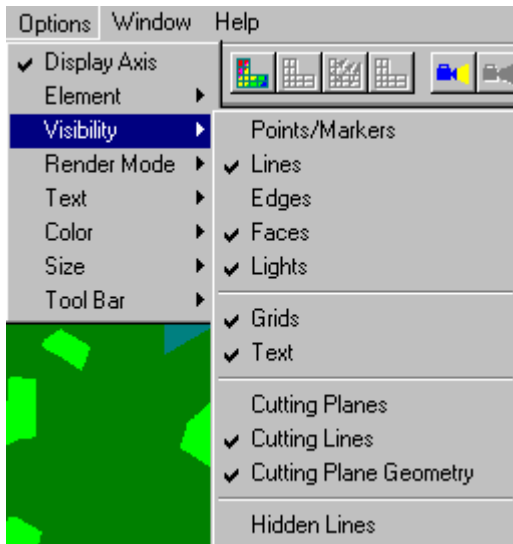


Fig. 8.2 The Options Visibility Menu

Options>Visibility>items

This allows the user to select or deselect the geometric and control entities to be displayed by toggling the appropriate menu item.

Points/Markers: these are used for nodes and geometric vertex display.

Lines: Toggles geometric line display.

Edges: Toggles edge display. They are usually for display F.E. sides

Faces: Toggles face display. These are for displaying element surfaces/faces

Lights: These are the light effect display controls

Grids: Grids are used to display objects such as cutting planes

Text: Text strings

Cutting Planes: The switch toggles the cut-plane filler visibility.

Cutting Lines: The cutting plane creates a cutting lines when it cuts across the elements. This option toggles the cutting line display.

Cutting Plane Geometry: This option toggle the visibility of the transparent cut-plane image.

Hidden Lines: Toggle the hidden line display option.

Option Render Mode

Options>Render Mode

This allows the user to select the type of rendering

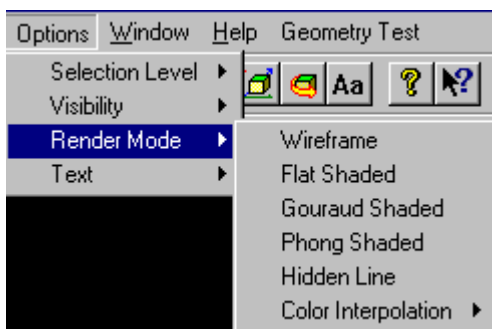


Fig. 8.3 The Options Render Mode Menu

Wireframe: display only the element sides

Flat shaded: this is the default option of shading for F.E. display. Although more complex shading options such as Gouraud/Phong shading can be used, flat shading represent the discrete nature of element surface best.

Gouraud Shaded: This is an option for a geometry object rendering.

Phong Shaded: Another option for geometry object rendering.

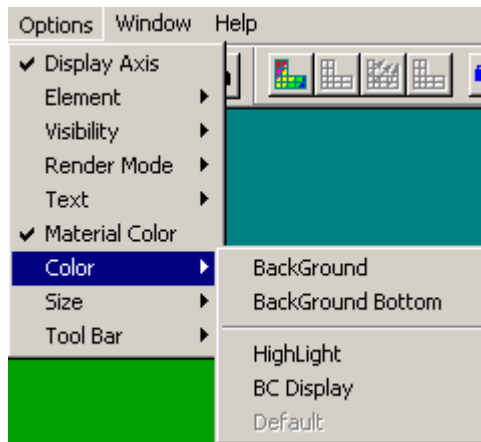
Hidden Line: display only hidden lines.

Color Interpolation: this option allows changing the coloring algorithm. For finite element objects, the color setting is completely automatic, and this option does not affect all F.E. display, except for imported geometry objects (such as data from AutoCad metafile). In such case, the coloring can be either by value or by an index mapping.

Option Color

Options>Color

This allows the user to change the screen background color and the highlight color used in the selection operation.



Options>Color>Material Color

When selected, the model will be color coded to represent different material property assignment.

Options>Color>Background

When selected, a color menu will pop up to allow background color selection.

Options>Color>Background Bottom

When selected, a color menu will pop up to allow the optional bottom background color selection.

Options>Color>Highlight

When selected, a color menu will pop to allow the selection of the color to be used in selection highlight.

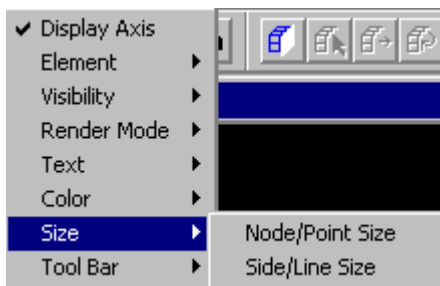
Options>Color>BC Display

When selected, a color menu will pop to allow the selection of the color to be used in display Boundary Condition Geometry highlight.

Option Size

Options>Size

This allows the user to change the default node or element side display size for visibility convenience.



Options>Text>Node/Point Size

This will bring up a menu to inform the user of the current node/point display size. Depending on the graphic hardware, this is usually the pixel size. The user can change it to larger or smaller to ease the visibility. To reset to the default setting, enter zero size.

Options>Text>Side/Line Size

This will bring up a menu to inform the user of the side/line display weight. The default weight is one. The user can increase or decrease it to ease the visibility. To reset to the default setting, enter zero weight (or one).

Option Text

Options>Text

This allows the user to insert an annotation text into the scene and control the type of text to be displayed

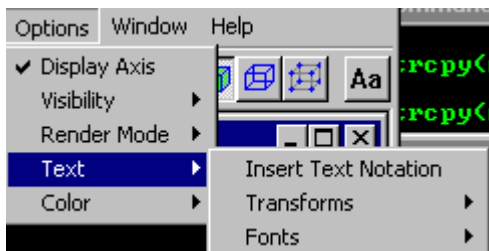


Fig. 8.4 The options text transform menu.

Options>Text>Insert Text Notation

This allows the insertion of a string of text with pointing line to annotate the model for additional clarity or presentation purpose. After activate this icon, click on the part of the object where the root of the pointing line should be, then click a 2nd time to indicate the text location. A pointing line will be immediately inserted to the scene. Finally, finish this operation by typing a string of text to be inserted at the end of the pointing line.

Options>Text>Transforms>off || Character Position || On

The inserted text string can be applied with transformation matrix. It is normally not necessary to do so, and the default is off. If desired, the transformation can be applied.

Options>Text>Fonts>Roman || Aerial || Cave

The inserted text string's font can be changed using this option.

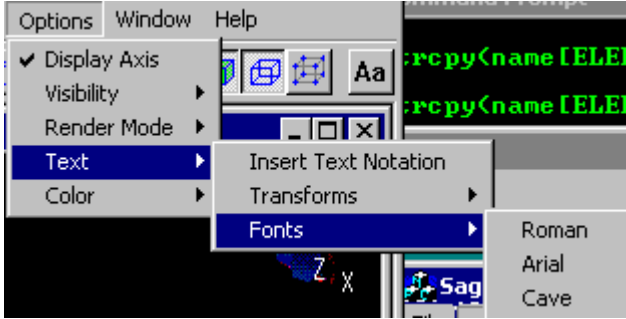


Fig. 8.5 The options Text Fonts Menu

Options>Color>Background || Highlight || Default

These options are not active as of the current release, but have been included in the next development planning.

Window Control Menu

The Window Control Menu

This allows the user to control the contents and view of the active window. The active view is always display at the end of this menu with a check mark, and can be selected to activate the view focus, that is, a view that has the mouse/keyboard attention.

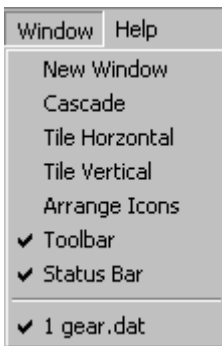


Fig. 9.1 The Window Control Menu

Window>New Window

This New Window feature allows adding additional view of the same model to the view list. Using this multiple-view option, the user has more flexibility to examine the model and results. There is no limit to the number of additional views can be added, as long the system has enough memory. Care should be exercised when multiple views are active, because each view window has it own associated selector, inquirer, etc.

The current version of AMPView is designed to handle a single model, but can have multiple views for model display. This is due to the internal animation control design. If the user desire to view a different model, the current model will need to close and the new model opened, or to activate another instance of AMPView and examine the new model there. The future version will allow multiple models to be opened in the same AMPView execution.

Window>Cascade

When multiple views are active, this option will cascade the view windows.

Window>Tile Horizontal/Vertical

When multiple views are active, this option will arrange the view windows in the desired tile fashion.

Window>Arrage Icons

When multiple views are active and ionized, this option will arrange these icon so they will be visible and in order.

Window>Toobar/Status Bar

These are options to turn on/off the toolbars.

File Toolbar

File control bar. The first icon saves the model data into a file. The second icon allows copying the scene image to the Windows' clipboard for other standard applications image pasting. The last icon prints the scene in the current display window.



Help bar. The first icon selects the online help file. The second is the interactive context sensitive "What is this" help. To use this feature, click on the menu, icon, or dialog, or position, and the program will automatically bring up the related help information.

View Toolbar

View bar. This gives us the isometric and the three standard views.

Display Toolbar

Display Control bar. The first icon allows a prefixed zoom-all view, while the second is a zoom by window control. The next 3 icons are the interactive pan, rotate, and zoom size control icons. To use these three controls, hold the left-hand-side button and drag the mouse across the screen to achieve the desired motion.



Finite element display control bar. This bar allows the various finite element features to be displayed. These are element faces, sides and nodes display controls. The last icon, when clicked, will display the model in transparent mode.



Annotation bar. This allows the insertion of a string of text with pointing line to annotate the model for additional clarity or presentation purpose. After activate this icon, click on the part of the object where the root of the pointing line should be, then click a 2nd time to indicate the text location. A pointing line will be immediately inserted to the scene. Finally, finish this operation by typing a string of text to be inserted at the end of the pointing line.

Select Geometry FE Toolbar



Geometry selection bar. This selection bar defines the type of finite element entities that can be selected by referring to the solid model entities. The first allows all solid elements to be selected through solid block selection. The second allows all 2D elements corresponding to a surface to be selected. The third allows all 3D element faces matching a solid face to be selected. The fourth allows all solid element sides corresponding to the geometry edge to be selected. The fifth allows all solid element nodes corresponding to the geometry edge to be selected. The sixth allows all 2D element sides corresponding to the surface edge to be selected, while the last allows a solid element node matching the solid vertex to be selected.

By using these type solid/surface model based element, face, side, node selections, the user can use the FE property or boundary condition even after the meshes are refined. Also, a single solid, face, or edge selection will immediately include all associated elements, element faces, element sides, or nodes.



F.E. selection bar. This filter bar defines the type of finite element entities that can be selected based on the finite element mesh. The first allows selection of elements. The second allows selection of element faces. The third allows selection of element edges. The fourth and last allows selection of nodes.



Select clear bar. The will clear all the current selection.

Result Display Toolbar



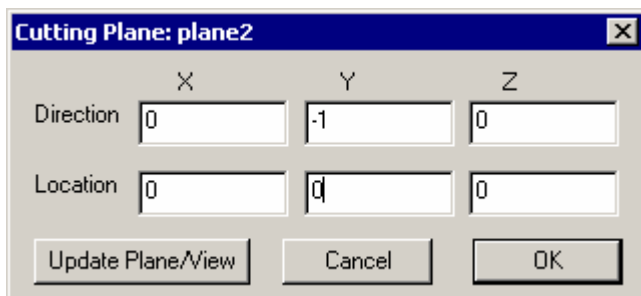
Result Display toolbar. These are the toolbar for quick access to result display. The first one select the type of contour display option, and the second one remove the contour display. The 3rd one presents a dialog box for vector result plotting control, and the 4th one remove the vector result display.

For animation, the camcorder icon (the 5th one) will animate the current result with an animation controls. The last camcorder icon is for multi-step result display control.

Cut Plane Toolbar



Cutting Plane Bar. These are the transparent cutting plane control icons. The first three icons activate a cutting plane insertion operation in the predefined YZ, XZ, and XY planes. When the cutting plane is active, the user can drag the cutting plane translation handle to move the cutting plane, and to rotate the rotational handle to rotate the plane. To remove the cutting plane, click the corresponding active cutting plane icon.



For precise cutting plane placement, an additional cutting plane placement control can be activated by clicking the last cutting plane icon to manually input the cutting plane's normal direction and an origin of the cutting plane.

Help Menu

The Help Menu

This allows the opening of the online Users manual or a brief description of the AMPView Program

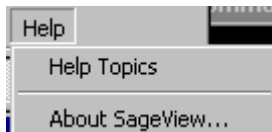


Fig. 10.1 The Help Menu

Help>Help Topics

This will bring up the on-line reference help.

Help>About AMPView

This will bring up the AMPView version and user's system information. They are necessary and useful when a user run into a problem, and need technical assistance.

Finite Element Features of the Solvers

The finite Element features of the solver are listed here. The actual implementation is by means of the appropriate selection in AMPView. Where possible, the online version of this document will have links to the relevant section of AMPView.

Element features

The current supported element types are:

ROD,

BEAM,

TRI3, TRI6

QUAD4, QUAD9, QUAD16, QUAD25

HEXA8, HEXA27, HEXA64, HEXA125,

TETRA4, TETRA10

PYRAMID5, PYRAMID13

SHELL3, SHELL4, SHELL6, SHELL9, SHELL16, SHELL25

Element side or surface Boundary Condition Types

The boundary conditions that can be applied to the side or surface are given in the following table.

Bc element side	BC type description
Global Force	Force1,force2,force3. Surface global direction traction (s11,s22,s33)
Normal Pressure	Surface normal pressure (positive when applied away from surface)
Radiation	Alpha and T ambient. Radiation= $\alpha(t^4-T_{amb}^4)$
Convection	Conv. coeff and tconv required, conv to ambient heat flux= $a(t-t_{conv})$
Flux	Flux value
Velocity/Vorticity	Fluid Velocity or Vorticity values
Pressure	Fluid flow pressure scalar value

Tutorials

AMPSol Tutorial

If you are first time user of AMPS AMPSolid AMPSol and AMPView, we strongly recommend that you run through the AMPSolid and AMPSol tutorial included in the AMPS package. That will give you a more overall view of how the program is designed to help you solve real life problem, and the suggested method of performing finite element analysis using AMPS.

Index			
B			
Basic nodal dof choices	52		
E			
Element side or surface Boundary Condition Types	91		
Enquirer Time Plot	76		
F			
FE Enquirer	74		
Finite Element Features of the Sage Solvers	91		
G			
Geometric Entities conversion to FE geometry	8		
I			
Insert a Cutting Plane	83		
Introduction	1		
M			
Material model types	40		
P			
Piezoelectrical Material Data	35		
R			
Result Display Toolbar	90		
S			
System Storage Files	4		
T			
The Boundary Condition Menu	40		
The Display Menu	82		
The File Menu	3		
The Help Menu	91		
The Mode Menu	10		
		The Model Condition Geometry Menu	42
		The Model Condition Menu	41
		The Model Condition Time Menu	44
		The Model Condition Type Menu	46
		The Model Condition Values Menu	52
		The Model Define Element Behavior Menu	25
		The Model Define Material Library Menu	64
		The Model Define Properties Menu	23
		The Model System Menu	
		Controls Menu	10
		Inertia Effects	16
		Iteration Control	14
		The Model/System/Unit Scaling Controls	21
		The Options Menu	84
		The OverSet Condition	63
		The Results	70
		Deformation Display Menu	70
		The Results Menu	65
		The Selection Menu	5
		The View Menu	82
		The Window Control Menu	88
		U	
		User Defined Initial Stress and Strain	39
		User Defined Plasticity Material	39
		W	
		Welcome	1