CONTENTS

MSC.Patran Reference Manual
Part 6: Results Postprocessing

CHAPTER
1
Introduction to Results Postprocessing

2
Quick Plots

3
Deformation Plots

4
Fringe Plots

14

Numerical Methods

15

Verification and Validation

Verification and Validation

15

INDEX

CHAPTER

1 Introduction to Results Postprocessing

1.1 Overview

The MSC.Patran Results application gives users control of powerful graphical capabilities to display results quantities in a variety of ways:

- Deformed structural plots
- Color banded fringe plots
- Contour line plots
- Marker plots (scalars, vectors, tensors)
- Cursor plots
- Freebody diagrams
- Graph (XY) plots
- Animations of most of these plot types.

The Results application treats all results quantities in a very flexible and general manner. In addition, for maximum flexibility results can be:

- Sorted
- Reported
- Scaled
- Combined
- Filtered
- Derived
- Deleted

All of these features help give meaningful insight into results interpretation of engineering problems that would otherwise be difficult at best.

The Results application is object oriented, providing postprocessing plots which are created, displayed, and manipulated to obtain rapid *insight* into the nature of results data. The imaging is intended to provide graphics performance sufficient for real time manipulation. Performance will vary depending on hardware, but consistency of functionality is maintained as much as possible across all supported display devices.

Capabilities for interactive results postprocessing also exist. Advanced visualization capabilities allow creation of many plot types which can be saved, simultaneously plotted, and interactively manipulated with results quantities reported at the click of the mouse button to better understand mechanical behavior. Once defined, the visualization plots remain in the database for immediate access and provide the means for results manipulation and review in a consistent and easy to use manner.

1.2 How this Guide is Organized

The Guide is broken into the following chapters to provide a logical flow.

Introduction to Results Postprocessing — An overview of the Results application. It is important that first time users read this thoroughly to fully understand how the Results application works. Important definitions are defined to understand how results data are stored in the database and how they are manipulated by the Results application. An overview of the operation of the Results application is also provided to give a basic understanding of how to create and modify result plots and how to post/unpost or delete existing plots and results data.

Quick Plots — Eighty to ninety percent of all postprocessing needs are accessed through the default Results application form. This chapter explains the Results application default form and how to create quick fringe plots of any scalar data and quick structural static deformation plots and modal style animations and combinations thereof.

Deformation Plots — Detailed explanations of how to create and modify deformation plots as well as how to change display attributes, target entities and other options.

Fringe Plots — Detailed explanations of how to create and modify deformation plots as well as how to change display attributes, target entities and other options.

Contour Line Plots — Detailed explanations of how to create and modify contour line plots as well as how to change display attributes, target entities and other options.

Marker Plots — Detailed explanations of how to create and modify marker plots (scalar, vector and tensor plots) as well as how to change display attributes, target entities and other options.

Cursor Plots — Detailed explanations of how to create and modify cursor plots as well as how to change display attributes, target entities and other options. Also instructions on how to create a report from the cursor plot.

Graph (XY) Plots — Detailed explanations of how to create and modify graph (XY) plots (including beam data) as well as how to change display attributes, target entities and other options.

Animation — Detailed explanations of how to create and manipulate animations of most plot types as well as how to change display attributes and other options.

Reports — Detailed explanations of how to create and display reports of results data as well as how to change report formats and other options.

Create Results — Detailed explanations of how to derive, combine and scale results data as well as how to select target entities, define transformation, derivations and other options.

Freebody Plots

Explains the capability to graphically display freebody diagrams and create new loads and boundary conditions from MSC.Nastran grid point force balance results.

Insight

Detailed explanation of the Insight application (another MSC.Patran postprocessor). This postprocessor has much of the same functionality as the Results application with the exception of results derivations/combinations, reports, graphs, and freebody. However the Insight application allows for some additional advanced postprocessing tools such as isosurface, scalar icon, value, cursor, and streamline plots.

Numerical Methods

Detailed explanations of the many numerical manipulations that are exercised in the Results application. These include operations such as vector and tensor to scalar calculations, extrapolation methods, coordinate transformations, results derivations and averaging techniques.

Verification and Validation

Verification and Validation problems are presented to validate and verify postprocessing displays using standard and widely accepted engineering problems. This is also a good source of example problems for learning to use the Results application.

1.3 Result Definitions

In order to fully utilize the power of the postprocessor, a thorough understanding of how the results are stored and manipulated is important. To avoid confusion or the possibility of misinterpreting the graphical displays, the following definitions should be understood.

Result Types. There are really only three results types, either scalar, vector, or tensor. Aside from these there are other aspects of results data as stored in the database that need to be understood. The following table summarizes these:

Term	Description
Nodes/Elements	Results are associated with either nodes or with elements.
Scalar Results	Single results values associated with either nodes or elements. They contain a magnitude only with no direction. Examples: strain energy, temperature, von Mises stress, etc.
Vector Results	Results values with three (3) components each associated with either nodes or elements. Vector results contain both magnitude and direction Examples: displacement, velocity, acceleration, reaction forces, etc.
Tensor Results	Results values with six (6) components each (typically comprising the upper triangular portion of a symmetric matrix) associated with either nodes or elements. Examples: stress and strain components
Real/Complex Number	Results stored as real numbers have only single values associated with any node or element. Complex numbers have two values associated with any node or element and are stored in the database as real and imaginary parts or magnitude and phase.
Load Case	A group of applied loads and boundary conditions which may produce one or more result cases.
Results Case	A collection of results as stored in the database (e.g., static analysis results, results from a load step in a nonlinear analysis, a mode shape from a normal mode analysis, a time step from a transient analysis, etc.).
Result Type	Either scalar, vector, or tensor. Scalar results contain a magnitude with no direction such as temperature, strain energy, von Mises stress, etc. Vector results contain both magnitude and direction, such as displacement, velocity, and acceleration. Tensor results are symmetric with six unique values (xx, yy, zz, xy, yz, zx) such as stress or strain at a point. Each Results Case can have many Results types in them.
Global Variables	Values associated with results cases as a whole rather than to individual nodes and elements. Each result case may be associated with zero, one or more global variables, (e.g. time, frequency, load case, etc.).

Term	Description
Primary Results	Physical quantities which may contain several different secondary result types. For example, stress is a primary result and von Mises stress is a derived or secondary result.
Layer Positions	The location where element results are computed for plates and shells which may be homogenous or laminated. Other types of elements have a default non-layered ID. Beam results can also be layered. Examples are top, bottom, and middle results of plate elements, different locations in a beam cross section, etc.
Element Positions	The location within the element (at a particular layered position for plates, shells, and beams) where results are computed. These positions are the quadrature points, element centroid, or nodal points. For beam plots, results at intermediate points along the beam can also be displayed as long as the analysis code has computed results at those locations.

When postprocessing results, you should be able to answer these questions about any data that is to be evaluated:

- Is the result type scalar, vector, or tensor?
- Is the result associated with nodes or elements?
- Is the result single-valued or complex (real/imaginary)?
- What layer-position does the result belong to?
- For element results, where in the element is the result computed?

Plot Definitions. The Results application provides various different plot types for results visualization. These plots, sometimes referred to as tools or plot tools, allow graphical examination of analysis results using a variety of imaging techniques and also simultaneous display of multiple plots to aid in the understanding of interactions between results. The following table summarizes the plots available followed by a description of each.

Plot Type	Description
Deformation Plots	Display of the model in a deformed state.
Fringe Plots	Contoured bands of color representing ranges of results value.
Contour Line Plots	Colored contour lines representing result values.
Marker Plots	Colored scaled symbols representing scalar, vector and tensor plots.
Cursor Plots	Labels for scalar, vector or tensor quantities are displayed on the model at interactively selected entities.
Animation	Not technically a plot type, however most plot types can be animated in a modal or ramped style or in a transient state if more than one result case is associated with an particular plot type.
Freebody Plots	These are freebody diagrams plotted specifically from MSC.Nastran grid point force balance results.

Plot Type	Description
Graph (XY) Plots	XY plots of results versus various quantities. Results can be plotted against other results values, distances, global variables or arbitrary paths defined by geometric definitions such as a curve.
Reports	Also not technically a plot type, however report definitions of results are stored in the database like any other plot tool type and can be created and modified to write reports to text files or to the screen.

Deformation plots are used to display the current model and posted plot tools in a deformed state. Care must be taken when applying other plots on a deformation plot when more than one deformation plot is posted since multiple deformation plots can easily clutter the graphics. An optional display of an undeformed model is controlled as an attribute of the deformation tool. The targeting of deformation tools to anything other than nodes and elements or groups of nodes and element is not allowable. Deformations may be used to display any nodal vector data.

Fringe plots map color to surfaces or edges based on the result data defined for the tool. Fringes are developed from nodal-averaged scalar values. Fringes may be plotted on the model's element faces or edges. The fringe tool will supersede all existing or default color and shading definition for the entities at which the fringe is targeted.

Contour Line plots display contour lines representing result data selected. Contours line plots are developed from nodal-averaged scalar values. Contour lines may be plotted on the model's element faces or edges.

Marker plots display nodal or element based scalar, vector or tensor results as icons or arrows at the result locations. Markers may be targeted at model features such as nodes, corners, and edges or faces of elements. Individual scalar, vector and tensor plots are described below but are known generically as marker plots.

Cursor plots display nodal or element based scalar, vector or tensor results as labels. There are three types of cursor plots: (1) Scalar, (2) Vector or (3) Tensor. Scalar, vector and tensor result quantities are displayed as one, three and six labels, respectively. Labels may be targeted at model features such as nodes and elements. Cursor plots are interactive and the labels are displayed on the model as the user selects the entities. The result value labels maybe displayed in a spreadsheet and written to a file, if desired.

Scalar plots display nodal or element based scalar data and are considered special types of marker plots. Scalars may be colored and scaled based on value and may be targeted at various model features such as node, faces and edges of elements, and corners.

Vector plots display nodal or element based vector data as component or resultant vectors and are considered special types of marker plots. Vectors may be colored and scaled based on magnitude and may be targeted at various model features such as node, faces and edges of elements, and corners.

Tensor plots display an iconic representation of a symmetric tensor and are considered a special type of marker plot. Tensors may be oriented in the axes of principal stress or the tensor's defined coordinate system. Tensors may be defined by element- or nodal-based tensor data. Nodal tensors are mapped from element tensors and are used when a tensor marker tool is targeted at other tools. Tensors may be targeted at nodal- and element-based model features.

Animation of most plot types is fully supported. Deformations can be animated in modal or ramped styles as well as true deformations from transient analyses. Animations from other plot types can accompany a deformation animation such as a stress field fringe plot or they can be animated separately from the deformation. Animation can be turned on or off from any existing plot or can be designated at creation time or when modifying a plot. The number of animation frames and other parameters such as the speed of animation are all easily controllable.

Freebody plots display a freebody diagram on a selected portion of the model. The plots are in the form of vector plots showing either the individual components or resultant values. Individual components that make up the total freebody diagram can also be plotted separately such as reaction forces, nodal equivalenced applied forces, internal element forces and other forces such as those from MPCs, rigid bars, or other external influences. New loads and boundary condition sets can be created from a freebody plot.

Graph plots are XY plots generally consisting of a results value versus some variable such as time or frequency or possibly a model attribute such as distance from a hole or edge or another results value.

Plot Attributes. The Results application provides the means of Creating, Modifying, Deleting, Posting and Unposting these plots as well as means for dynamically manipulating these plots for interactive results imaging. Each plot created has assigned attributes which determine its characteristics. All plots have the following attributes.

Attribute	Description
Name	A unique user-definable string descriptor to identify the plot tool. If no plot name is specified a default name is used. The default will be used each time unless the user specifically defines a unique name.
Type	One of the plot tool types described in **Plot Definitions** (p. 6).
Result(s)	A results case or a list of results cases and the corresponding result type which the plot tool is to display.
Target	Onto where or to what entities the plot is to be displayed. This is either on a model feature such as nodes, elements, or on another plot tool.
Display Attributes	Each plot type has specific settings to control how the plot is to be displayed. These include such things as component colors, titles, label, rendering styles and a myriad of other attributes.
Animation Attributes	Attributes to describe whether the tool is to be animated and how the results are to be mapped to animation frames. For instance, is the animation modal or transient and how many frames will be used for the animation?
Posting Status	Each plot is either Posted (displayed) or Unposted (not displayed) with the exception of reports.

Plot Targets. Result plots may be displayed on selected model entities or other selected plot tools. The model based targets may be defined by a list of posted groups, by all posted entities in the current viewport, or by individual nodes or elements or by elements with certain attributes. The model entities and tools which may act as targets for Results application plots are described below.

Elements indicate that results will be displayed on all selected elements of the model. For graphs and reports the information can be extracted from the centroid, the element nodes or element data as stored in the database.

Free faces describe those element faces common to only one element. This includes faces lining the outside surface of a model or those inside surfaces exposed to internal voids. Free faces are appropriate targets for displays such as fringe plots which are normally displayed on the surface of the model or on a cutting plane through the model.

All Faces display results on each face of each element.

Free Edges display results on edges common to only one element. Use this target type when displaying results on the same edges which are used to draw the model when Free Edge is selected as the finite element display method.

All Edges display results on all element edges. Using this target selection allows mapping of results onto a wireframe representation of the model.

Nodes display the selected results at each nodal location of the model. Tensor and vector plots may all be displayed at nodal locations.

Corners display the selected results at nodes which are common to only one element. Tensor and vector plots may all be displayed at corner locations.

Paths display the selected results along a defined path. The path can be defined as either a series of beams or element edges, geometric curves, or selected points (either geometric or FEM based). This target type is used with Graphs plots.

The following table summarizes the valid targets for all plot tools. When specifying target entities in most cases you must specify both the target entities to which the plot will be assigned and the attributes or additional display information. The table below shows target entity versus attribute and which plots types are valid (D=deformation, F=fringe, Cl = Contour Lines, S= Scalar, V=vector, T=tensor, Cu=Cursor, G=graph, R=report).

Target	Attribute								
	Element	Free Faces	All Faces	Free Edges	All Edges	Nodes	Corners	Curves/ Edges/ Beams	Elem. Nodes / All Data
Current Viewport	D,S,V,T,R	F,Cl,S ,V,T	F	F,S,V ,T	F	D,S, V,T,R	S,V, T		R
Nodes						D,S,V,T, Cu,G,R			
Elements	D,S,V,T, Cu,G,R	F,Cl	F	F	F				R
Groups	D,S,V,T,G, R	F,Cl,S ,V,T	F	F,S,V ,T	F	D,S,V,T,G, R	S,V, T		G,R
Materials	D,S,V,T,G, R	F,Cl,S ,V,T	F	F,S,V ,T	F	D,S,V,T,G	S,V, T		G,R

Target	Attribute								
	Element	Free Faces	All Faces	Free Edges	All Edges	Nodes	Corners	Curves/ Edges/ Beams	Elem. Nodes / All Data
Properties	D,S,V,T,G,R	F,Cl,S,V,T	F	F,S,V,T	F	D,S,V,T,G	S,V,T		G,R
Element Types	D,S,V,T,R	F,Cl,S,V,T	F	F,S,V,T	F	D,S,V,T	S,V,T		R
Paths								G	

Other Definitions

Term	Definition
Post	To graphically display the plot or plots.
Unpost	To remove the plot or plots from the graphical display.
Range	A MSC.Patran database entity defined by a series of number and threshold values for each level within a range. Ranges are used to map spectrum colors to results values. A spreadsheet form is available to control range levels.
Viewport Range	The range entity currently assigned to the MSC.Patran viewport.
Auto Range	A range which is not a database entity but is automatically calculated for a plot based on the results values. This type of range may be manipulated dynamically to change the range extremes and the number of intermediate levels.
Extrapolation	Methods of converting results values from certain element locations to other locations (e.g., converting results at Gauss points to nodal values).
Averaging	Methods of converting several results associated to the same physical location to a single results value such as when results at nodes have contributions from all connected elements.
Derive	Methods of converting results values, for instance, when calculating von Mises stress from stress tensor components.
Interpolation	Methods of calculating new results values between existing locations of results values. For example: displaying more frames of animations than results cases available.
Coordinate Transformation	Methods of transforming results values with magnitude and direction attributes into alternate systems.

More detailed information on the numerical methods can be found in Numerical Methods (Ch. 14).

Results Label. MSC.Patran displays results labels on plots so that all labels are started at the free end of the line segment (away from the node or element centroid); and continue to the right, independent of the arrow. Often the label is obscured.

For vector and tensor plots, you can now set the label to appear at the free end of the line segment, and position it so that it appears centered with respect to the arrow. All labels are pushed “away” from the segments (i.e., an arrow that goes from the screen center to the left will have the label end at the arrowhead instead of the begin at the arrowhead, as in the past). To enable the label placement feature, you need to add a preference to the Patran db using:

```
db_add_pref(524,2,0,TRUE,0.0,"")
db_set_pref_logical(524,TRUE)
```

from the patran command window input text data box. This will remain in effect for the life of the database.

When the "VECTORTEXTCENTERED" preference (524) is in effect, the label text associated to results vectors, result tensors, lbc marker "arrows", property "arrows", and arrow created using "gm_draw_result_arrow":

- are not rendered until the end of the viewport rendering. The text that is attached to an arrow is drawn at a location so that the free end of the vector receives the text. The hang point of the text is translated (in the 2d world) such that the center of the box enclosing the text (text box) is contained in the line of the (2d) vector and the edge of the text box is just touching the free end.
- are suppressed (not rendered) if the free end of the vector to which the text is attached is occluded. That is, if the z-depth of the device coordinate for the free endpoint is greater than the current z-depth for the device x,y (something eclipses the end of the vector tail) then the text is suppressed. This does not apply if the viewport was rendered entirely in wireframe mode. All vector text is considered visible if the viewport was rendered in wireframe mode.

1.4 Capabilities and Limitations

The Results application provides the capabilities for Creating, Modifying, Deleting, Posting, Unposting and manipulating results visualization plots as well as viewing the finite element model. In addition, results can be derived, combined, scaled, interpolated, extrapolated, transformed, and averaged in a variety of ways, all controllable by the user.

Control is provided for manipulating the color/range assignment and other attributes for plot tools, and for controlling and creating animations of static and transient results.

Results are selected from the database and assigned to plot tools using simple forms. Results transformations are provided to derive scalars from vectors and tensors as well as to derive vectors from tensors. This allows for a wide variety of visualization tools to be used with all of the available results.

Results imaging routines are optimized for graphical speed but may vary depending on hardware.

Please be aware of the following limitations or constraints:

- When a Result Data quantity is deleted, it is deleted from all Results Cases that contain the Results Data quantity.
- Transient animations are not possible from the Quick Plot form. They must be created under each specific plot type option (deformation, fringe, marker, etc.).
- Multiple animations can be viewed simultaneously in a single viewport.
- Only one spectrum and range can be associated to any one viewport at a time. If multiple plots are posted to a viewport, the spectrum and corresponding range will only be applicable to one of the posted plots. Values from other plots not corresponding to the posted range will take on the posted range's spectrum. Values outside the range will appear as the highest or lowest range color. This may make some plots appear monochrome. This is done to avoid confusion and misinterpretation of result data.
- It is not recommended to calculate invariants (e.g., von Mises) from complex results because the phase is not accounted for.

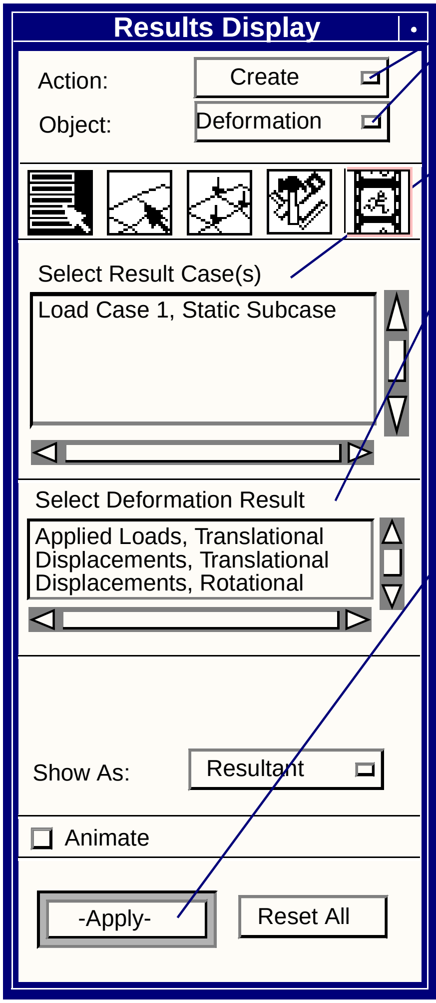
1.5 Using Results

The Results application is based on the creation and manipulation of results visualization plots. The first action to be performed using Results is to create a plot, sometimes referred to as a tool or a plot tool. This however is transparent to the user when doing basic operations such as simple deformed plots, fringes, and animation. Each plot type has its own default settings and attributes which are set and modified when a user creates a plot. Only when these settings and attributes need to be saved and restored quickly for subsequent use does the user need to concern himself about physically saving the plots. This is done using the Create action on the main Results application form. Other actions are described in the following table, and summarized in this section.

Action	Description
Create	This action is used to create Results visualization plots sometimes referred to as tools. Creating a plot will result in a graphical display with the exception of creating reports and deriving results. If you try to create a plot that already exists, you will be prompted for overwrite permission.
Selecting Results and Filtering Results	Sometime it is necessary to select only certain Results Cases or to filter the Results Cases specifically for more precise control when creating plots. A special form allows you to do this easily and efficiently as well as view all Results Cases available to you.
Modify	This action is used to modify existing Results visualization plots or tools. This action performs identically to the Create action with the exception that no overwrite permissions will be asked if plot tools already exist that are being modified.
Post/Unpost	This action is used to graphically display (post) or graphically remove (unpost) existing Results display plots or ranges/spectrums from the computer screen. The plots and ranges are not physically removed from the database with this operation. Only their graphical display is recalled or removed.
Delete	This action is used to delete existing Results visualization plots and for deleting result cases and result data associated with result cases from the database. Use this option with care. Some operations may not be undoable.
Spectrum/Range Control	There is a form which allows the currently posted spectrum to be changed and manipulated as well as deleted or new ones created. There is also a form which allows for control over which range (numbers) are assigned to newly created plots and also control over each color bar of the spectrum for the currently posted plot or plots. See **Spectrum/Range Control** (p. 29).
Animation Control	These are forms for setting up and controlling certain aspects of an animation. See **Animation** (Ch. 9) for details.

Create

Creating a plot generally involves four to six basic steps (although it may vary from plot type to plot type). For simple plots where it is acceptable to use all default values then the Quick Plot option is all that is needed. The icons on the top of the form give access to all controls necessary. For full control of most plots the steps are:

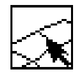


STEP 1: Set the Action to Create and select an Object (the plot type) from the Results application form.

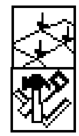
STEP 2: Select a Results Case from this listbox.

STEP 3: Select a result associated with the Results Case from this listbox.

STEP 4: Select the target entities to which the plot will be applied (optional).



STEP 5: Set any plot attributes if necessary (optional).



STEP 6: Press the Apply button on the bottom of the form. The plot will be displayed.

Note: A separate chapter is dedicated to describe in detail the creation and manipulation of each plot type.

More Help:

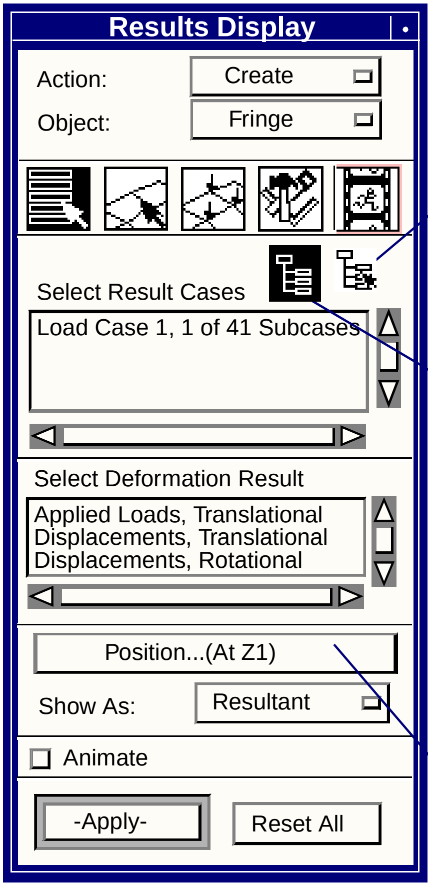
Plot Types:

- Quick Plots (Ch. 2)
- Deformation Plots (Ch. 3)
- Fringe Plots (Ch. 4)
- Contour Line Plots (Ch. 5)
- Marker Plots (Scalar, Tensor, Vector) (Ch. 6)
- Cursor Plots (Ch. 7)
- Graph (XY) Plots (Ch. 8)
- Animation (Ch. 9)
- Reports (Ch. 10)
- Create Results (Ch. 11)

Important: Plots can be optionally named and saved in the database and subsequently recalled and graphically displayed. If no name is given, a default name is assigned. If a new plot is created without specifying a name, the default will be overwritten each time. Overwrite permission will be asked if a name is given and it already exists.

Selecting Results. For all operations you must select results from a listbox. What results are displayed in this listbox is somewhat dependent on the result type (static, transient, etc.) or the number of subcases, time, frequency, or load steps associated with these results and how they have been filtered. When multiple subcases, time, frequency, or load steps are present, the display in the Select Result Cases listbox will display a title such as `LoadCase x, n of n subcases` or something similar indicating that there are multiple sets of results for this Result Case.

When multiple results exist for any given Result Case, an additional button and toggle appear on the form. One allows for filtering and selecting the desired subcases which will appear selected in the listbox and the other determines the appearance of these multiple results in the listbox itself.



This button icon brings up the Select Result Cases form to allow for selecting and filtering of the results cases based on various criteria such as a global variable (time). Once the filtering has been done only those results that passed the filter criteria will be selected for subsequent postprocessing. See **Filtering Results** (p. 19).

This button icon changes the way the Result Cases are displayed in the listbox. If toggled OFF, every individual subcase, time, frequency, or load step will be visible in the listbox. If toggled ON, then only the title of the primary Result Case will appear with a summary of how many subcases are associated with it based on the filter criteria. This is known as the *abbreviated* form.

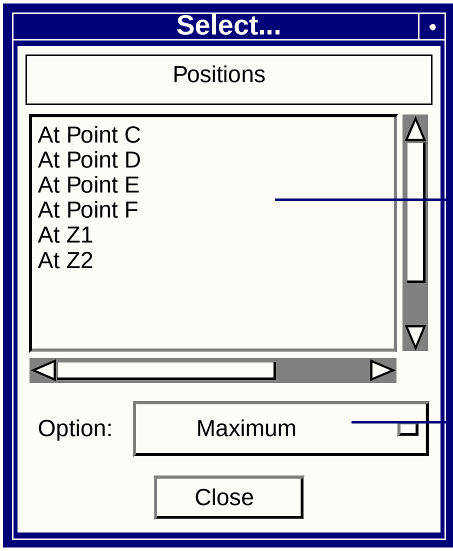
This toggle will not appear unless Result Cases with multiple subcases exist. This is true also for the Select button icon.

Additional results selection control is given when multiple layers exist.

Once Result Cases have been selected and filtered, the Result Case name will be updated to show how many subsets of that Result Case have been selected. The name will appear something similar to `LoadCase x, m of n subcases`. If the Abbreviate Subcases toggle is then turned OFF, only those subcases selected through the filtering mechanism will be highlighted in the listbox. How to filter results is explained in **Filtering Results** (p. 19).

Be aware that when selecting multiple Result Cases, such as for a transient animation, that the selected result type to plot must exist in all Result Cases selected. Otherwise an error message will result and no plot will be displayed until the Result Case selection is modified to meet this criterion.

Result Layer Positions. When multiple layers exist in a given result type, this form may be invoked by pressing the Position button in the Results application when the mode is set to Select Results. Layers may correspond to shell top / bottom, ply layups, or different element type results.



This form is accessible by pressing the Position button from the Select Results mode if multiple layers exist for a given result type.

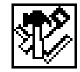
Select the position(s) you wish to be used in any subsequent plot. You may select multiple layers for any single Result Case or you may pick a single layer for use when a single or multiple Result Cases are selected.

Set the option to search and extract result quantities when multiple layers or Result Cases have been selected. These options are explained in the table below.

Action	Description
Maximum	If multiple layers or multiple Result Cases have been selected, then this option will search through all layers / Result Cases and extract the maximum value encountered for the subsequent plot. The value used in the search is the Quantity selected for resolution such as von Mises for tensor results.
Minimum	This is identical to Maximum except the minimum is extracted.
Average	Instead of extracting a maximum or minimum, values are averaged from each layer or Result Case based on the Quantity selected and graphically reported. Averaging is only performed over the number of actual layers or Result Cases that contained results at any entity. That is if 4 layers were selected and node 1 had three layers of results and node 4 had four layers of results, node 1 would be averaged only over the three that actually existed and not the four selected.
Sum	This option simply sums all values of the requested Quantity from each layer or Result Case and reports that value in the subsequent plot.
Merge	This option will use the first existing value encountered from any particular layer or Result Case. For instance if both top and bottom stresses are selected then only the top will be reported. This is useful for layers that are associated with certain element types. That way a layer with shells, a layer with solid, and a layer with beam elements can all be displayed simultaneously on the graphics screen in one operation.

Note that when performing maximum/minimum extractions or averaging and summing that the following procedures are performed in order:

1. The selected Quantity of interest is calculated for all layers or Result Cases selected, first performing any transformations, scaling, and averaging, or extrapolation as requested in the Plot Options. Quick Plot operations use standard defaults for all plot options.

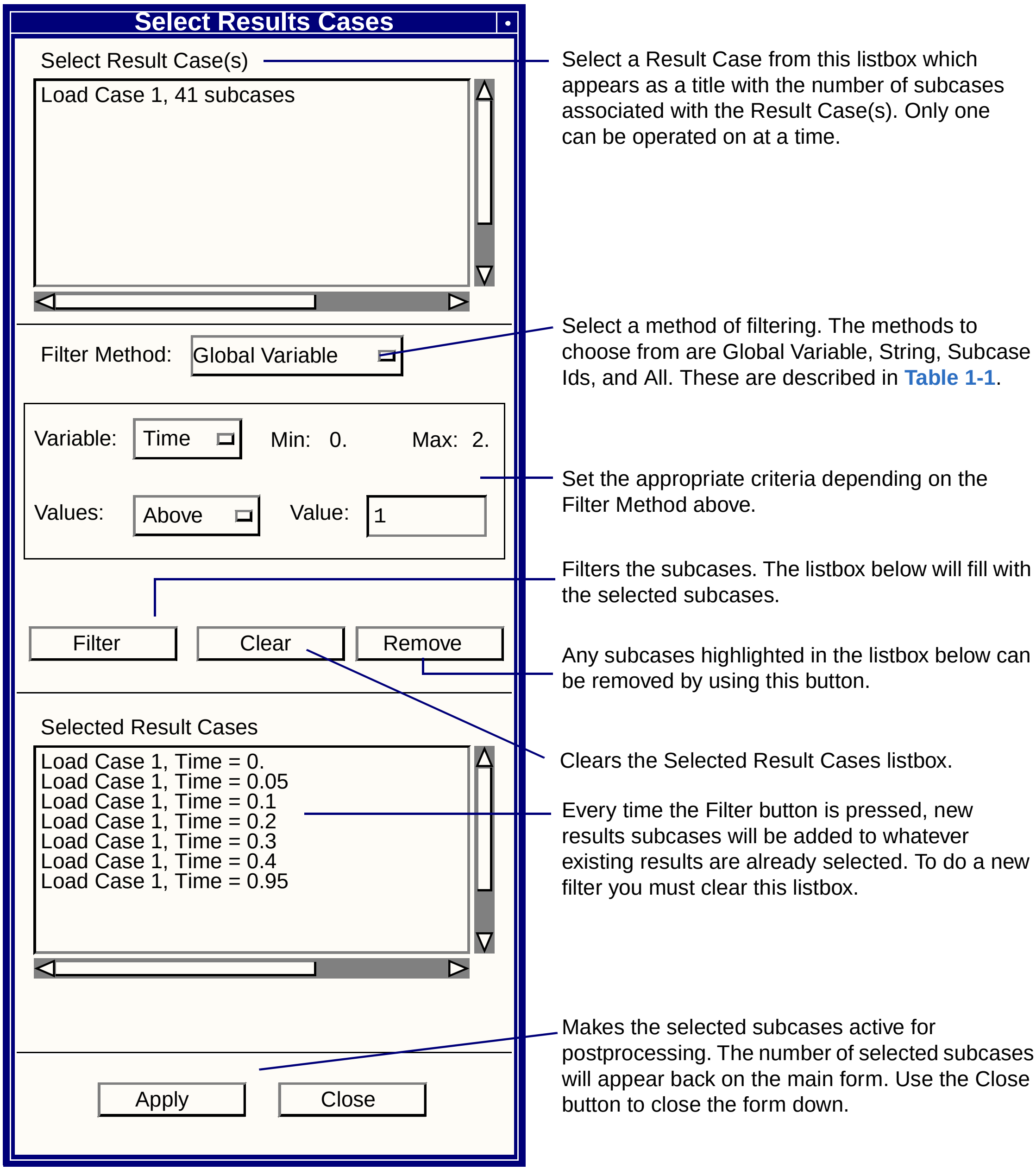


2. Once the Quantity of interest is calculated for all selected layers or Result Cases, the maximum/minimum extraction, averaging or summation is performed and reported in the subsequent plot.

   Note that this operation is different than what the results derivations do in **Derived Results** (p. 193). These operations are scalar based, meaning that the maximum, minimum, average, or sum operations are done based on the requested scalar quantity.For instance, you would not be able to properly calculate von Mises stress at the neutral axis of a beam in pure bending by selecting the top and bottom layers and requesting an average where the expected von Mises stress should be zero. The von Mises will be calculated at top and bottom and then averaged. For this type of operation where the components of a vector/tensor need to be averaged or summed before the requested result quantity is calculated or the vector/tensor components based on maximum or minimum comparisons of the requested scalar quantity are required, you must use **Derived Results** (p. 193).

Important:	It is important to note that if multiple Result Cases have been selected and only a single layer exists or has been selected that the default plot will result in a maximum plot of all selected results.

Filtering Results. Filtering results is accomplished from the Select Result Cases form which is accessible from the Results application when the first icon button (Select Results) is active and multiple subcases exists. An icon button appears when Result Cases are in their abbreviated form to access the filter form which can also be accessed by clicking on the Result Case name.



This form is expandable to allow you to view the entire Result Case names and global variable if necessary.

The different filter methods are explained in **Table 1-1**.

Table 1-1 Filter Methods

Method	Description
Global Variable	Any global variables associated with the selected Result Case will show up in the Variable option menu. Select the one you would like to filter with, change the criteria using the Value option menu and enter the value or range to filter by. Press the Filter button to complete the filter action. Press the Apply button at the bottom of the form to activate the filtered subcase selection.
String	Enter a string and use wild cards (the * character) to filter results. For example if you wanted all subcases with the string Time in it then you would use *Time* as the string with wild cards on each end of the word. Press the Apply button at the bottom of the form to activate the filtered subcase selection.
Subcase IDs	Subcases can be filtered on Subcase IDs by entering the appropriate IDs. To select separate IDs, separate them by spaces (1 3 5). To select a range use a colon between the numbers (1:5). To select by increments use two colons, for example: 1:10:2, which interpreted means select subcases one through 10 by twos. Or use any combination of spaces and colons between subcase IDs to select as many as you wish. Press the Apply button at the bottom of the form to activate the filtered subcase selection.
All	No filter method is selected. No options are available. Simply press filter and all subcases will be selected from whatever primary Result Case is selected. Press the Apply button at the bottom of the form to activate the filtered subcase selection.

Important:	Only one Result Case can be filtered at a time. If you need to filter subcases from more than one Result Case then you will need to perform the operation once for each Result Case.

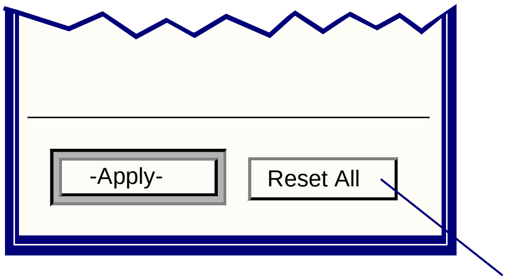
Note on Result Case Names:

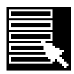
By default if a Result Case has more than 30 subcases (time steps, load steps, etc.) then the Result Case name will be displayed in an abbreviated form to reduce clutter in the listboxes. The default number at which this abbreviated form takes over can be changed with a `settings.pcl` parameter:

```
pref_env_set_integer( “result_loadcase_abbreviate”, 30 )
```

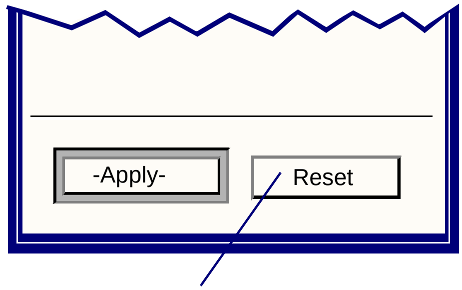
See **The settings.pcl file** (p. 41) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*. It is possible to toggle back and forth from abbreviated form and full form at any time by pressing the icon button shown here.

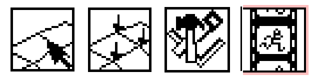
Default Settings. For all modes of the Results application (selecting results, target entities, display attributes, plot options, and animation options) logical defaults have been set. In general, when an option on a form is changed, it remains until the user modifies or resets it. On the bottom of the Results application form is a Reset button that will restore default settings. Pressing the Reset button only affects the particular plot type currently set.





In the Select Results mode of the Results application, the Reset All button will restore all default settings for any particular plot type. This includes target entities, display attributes, plot and animation options.





In any other mode of the Results application, the Reset button will only restore that modes settings for any particular plot type.

In some instances it is possible to modify these defaults to the user's preference. Not all default attributes and setting can be altered by the user since certain dependencies exist on result types and available options. However for display attributes, default setting may be altered in a template database. This template database can then be saved and made available to all users that wish to use the altered default attributes. See **The Template Database File (template.db)** (p. 47) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*

In order to accomplish this, the standard MSC.Patran database is pre-loaded with invisible plot tools called `MSC_Initialize`. There is one for each plot type (Deformation, Fringe, Vector, Tensor, Graph). This plot is never visible to the user but default display attributes are extracted from these plot tools.


Toggles the form to change display attributes for all plot types.

To modify the default display attributes, you simply need to modify the `MSC_Initialize` plot tool for the plot type in question. These basic steps need to be followed:

1. Open a MSC.Patran database that already has a model and results or simply create a new database and model using Demo results (See **Demo Results** (p. 202).)
2. Create a plot of the type you wish to modify with the display attributes that you want. A PCL command will be issued in the command line window of MSC.Patran. It will also be output to a session file typically called `patran.ses.01` (the version number may vary).
3. Either edit the session file or edit the PCL command from the command line by replacing the plot name (which will probably be blank something like `default_XXX` where `XXX` is `Fringe`, `Tensor`, `Vector`, etc.) with the name `MSC_Initialize`. See the example below.
4. Also edit this PCL command such that it is a modify command as opposed to a create command.
5. Close down the current database and open a new blank database.

6. Run the edited session file or re-issue the PCL command to modify the `MSC_Initialize` plot tool.
7. Save this database as the new `template.db`.

   Display attributes for this modified plot tool have now been set.

As an example, say that the default display attributes for a deformation plot are to be modified. The following PCL command is issued when creating a deformation plot with the desired attributes:

```
res_display_deformation_create(““,“Elements”,0,[““],9,[“DeformedStyle:White,Solid,1,Wire
frame”,“DeformedScale:Model=0.1”,“UndeformedStyle:ON,Blue,Dash,1,Wireframe”,“TitleDispla
y:ON”,“MinMaxDisplay:ON”,“ScaleFactor:1.”,“LabelStyle:Fixed,8,White,4”,“DeformDisplay:Re
sultant”,“DeformComps:OFF,OFF,OFF”])
```

The PCL command should then be edited as follows:

```
res_display_deformation_modify(“MSC_Initialize“,“MSC_Initialize”,“Elements”,0,[““],9,[“D
eformedStyle:White,Solid,1,Wireframe”,“DeformedScale:Model=0.1”,“UndeformedStyle:ON,Blue
,Dash,1,Wireframe”,“TitleDisplay:ON”,“MinMaxDisplay:ON”,“ScaleFactor:1.”,“LabelStyle:Fix
ed,8,White,4”,“DeformDisplay:Resultant”,“DeformComps:OFF,OFF,OFF”])
```

Note that the only modifications are to change the *create* to *modify* in the PCL function name and enter the name of the plot tool `MSC_Initialize` twice. This PCL command should then be issued either via a session file or directly from the command line after opening a new empty database. The above example simply sets the undeformed line style to dashed as opposed to the standard solid line.

Use Templates

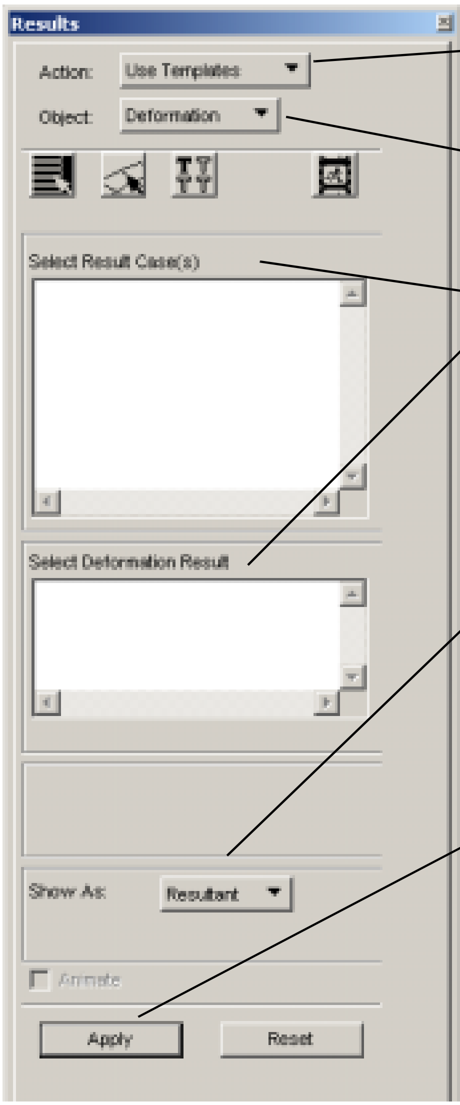
This Action menu provides the means to select and use Results Templates to make Deformation Plots, Fringe Plots, Marker Vector Plots, Marker Tensor Plots, Graphs, and Reports.

The menu is similar to the Create menus, except that the Results Display Attributes and Plot Options icons and associated menus have been replaced with a Display Templates icon and associated menu. The Display Attributes and Plot Options values will be determined by the Results Template selected instead of the many individual menu settings on the Display Attributes and Plot Options forms of the Create menu.

For Results plots (Deformation, Fringe, Marker Vector and Marker Tensor) you may chose a title either from the template or as determined by the load selection on the Select Results form. Either can be edited once selected by the corresponding switch for “Title From:” “Template” or “Load Selection”.

Graphs do not use titles.

Report Titles are accessed via the “Format...” button.



STEP 1: Set the Action to Use Template.

STEP 2: Select the type of plot for which you will use a template.

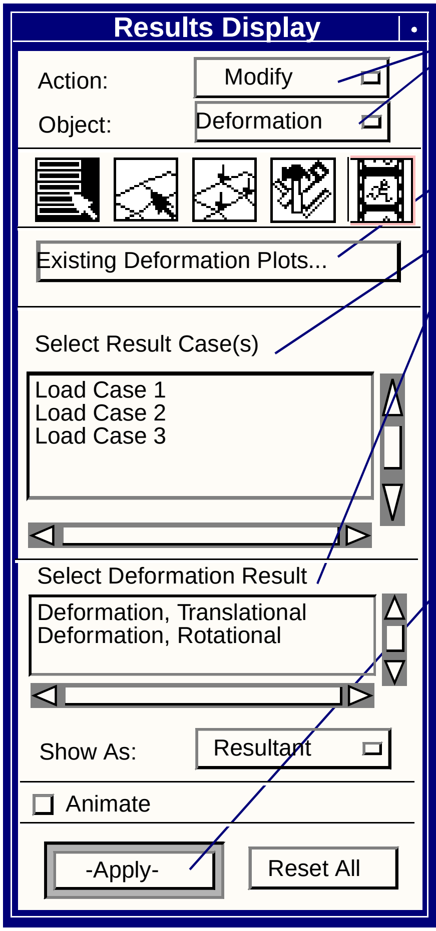
STEP 3: Change the Result Cases and/or results type assigned to this plot (optional).

STEP 4: Select to show as either Resultant or Component.

STEP 5: Press the Apply button on the bottom of the form.

Modify

Once a plot has been created, it may be modified using the Modify action on the Results application form. It is only necessary to actually modify a plot if it has been optionally named and saved in the database. Otherwise the Create action can be used exclusively. Default plots can be overwritten with the Create action. To modify a named plot, follow these general steps:

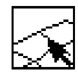


STEP 1: Set the Action to Modify and select an Object (the plot type) from the Results application form.

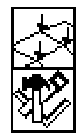
STEP 2: Select the named plot to be modified.

STEP 3: Change the Result Cases and/or results type assigned to this plot (optional).

STEP 4: Change the target entities to which the plot will be applied (optional).



STEP 5: Modify the plot attributes and other options if desired (optional).



STEP 7: Press the Apply button on the bottom of the form. The plot will be modified and displayed if not already.

Note: A separate chapter is dedicated to describe in detail the creation and modification of each plot type.

More Help:

Plot Types:

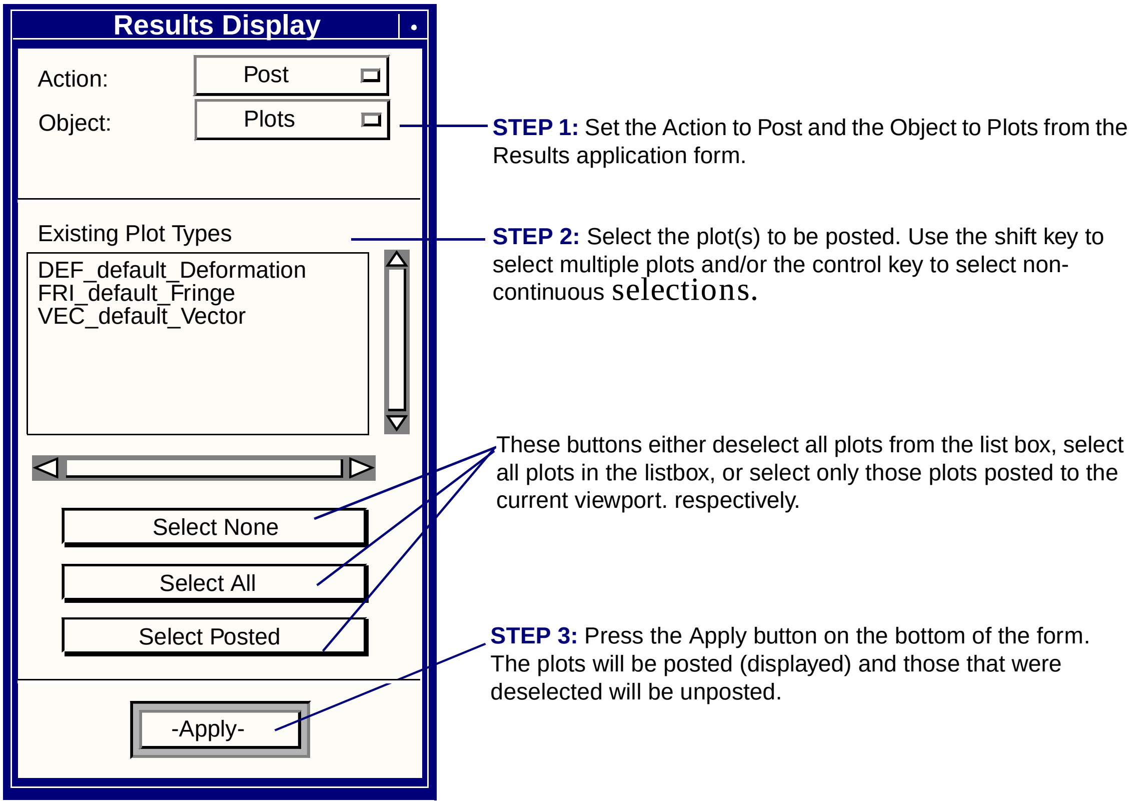
- Quick Plots (Ch. 2)
- Deformation Plots (Ch. 3)
- Fringe Plots (Ch. 4)
- Contour Line Plots (Ch. 5)
- Marker Plots (Scalar, Tensor, Vector) (Ch. 6)
- Cursor Plots (Ch. 7)
- Graph (XY) Plots (Ch. 8)
- Animation (Ch. 9)
- Reports (Ch. 10)
- Create Results (Ch. 11)

Important:	It is suggested to only modify plots that have specifically been given names. It is not necessary to modify the default plots. Default names are given to the plots when no specific name is specified. The Create action continually overwrites these default plots with their corresponding names, therefore it is not necessary to use the Modify action on them.

Post/Unpost

Posting and unposting of plots to the graphics viewport(s) can be performed. Posting or unposting of ranges and their corresponding spectrum is also allowable. See the next section **Posting/Unposting Ranges** (p. 26).

Posting/Unposting Plots. Once a plot or set of plots has been created, they may be posted (displayed) or unposted (removed) with the Post action on the Results application form. (This is also true for Ranges. See**Posting/Unposting Ranges** (p. 26) and **Spectrum/Range Control** (p. 29) for more detail. Multiple plots may be posted simultaneously. To post or unpost a plot, do the following:



When multiple viewports are in use, make sure that you make the viewport to which you want to post the plots active. The current viewport always has a red border around the graphics. To change the current viewport, place the cursor in the border of the graphics window (the cursor will change to hand icon) and click the mouse button. The Post/Unpost listbox plot will update itself to show what plots are posted to the currently active viewport.

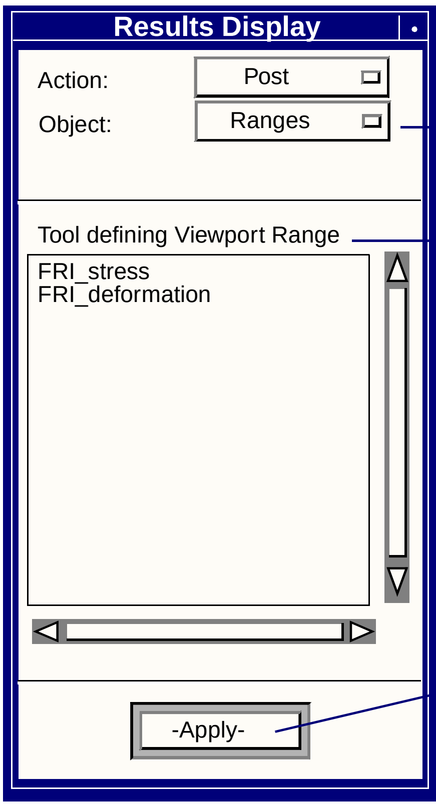
By default all posted plots will be re-posted when a database is opened. This can be overridden by using a special setting parameter in the `settings.pcl` file. The function is

```
pref_env_set_logical("result_dbopen_display",TRUE/FALSE)
```

The default is TRUE. See **The settings.pcl file** (p. 41) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Important:	Most plots can be targeted to or displayed on a deformed shape plot. When more than one deformation plot is posted, those plots that have been targeted at deformed plots will be displayed on all deformed plots that are posted unless specified differently under the Target Entities.

Posting/Unposting Ranges. Each plot created is assigned a range according to the results values it is associated with. It is possible to put up multiple plots that are associated with varying types of results. It is possible that the result values from each plot vary by orders of magnitude (displacement and stress for example). Posted plots will always take on the color spectrum currently posted. This means that some plots may turn monochrome if their results values are outside the range of the color spectrum posted. You may post and unpost the ranges associated with the various plots that are posted. Each posted plot associated to a color spectrum will be updated accordingly.



STEP 1: Set the Action to Post and the Object to Ranges from the Results application form.

STEP 2: Select the range to be posted. Only one range can be selected and posted at any one time. There will be a range for each plot posted unless deleted by the user.

This is a list of existing plot tools and not a list of actual ranges. The range associated with the selected plot tool will be assigned to the current viewport.

The plot whose range is currently displayed is noted at the bottom of the spectrum on the graphics window.

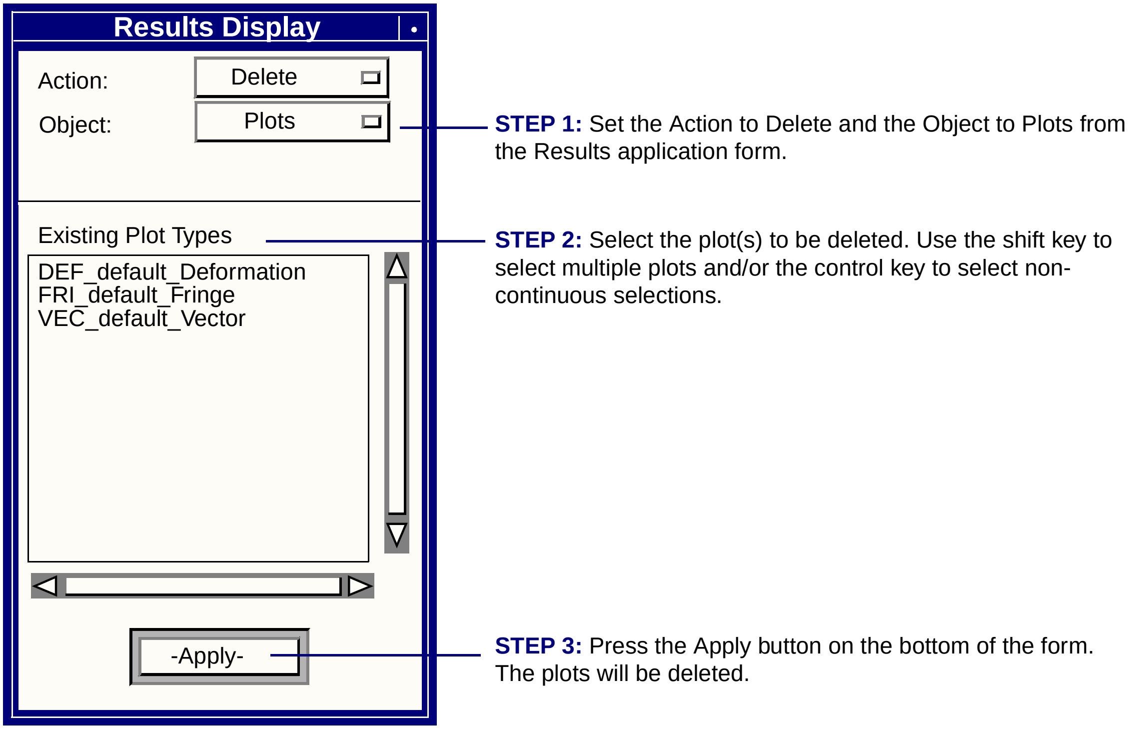
STEP 3: Press the Apply button on the bottom of the form. The range will be posted (displayed) and posted plots will be updated to reflect the new range.

More information on how the Results application uses ranges can be found at the end of this chapter in **Spectrum/Range Control** (p. 29).

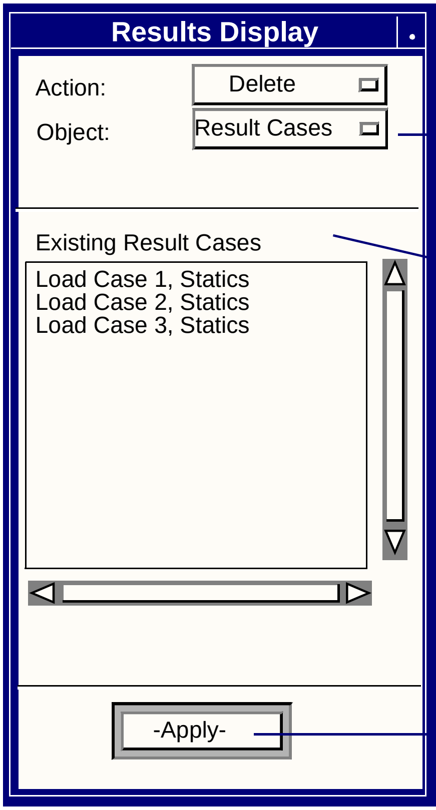
Delete

Two items may be deleted: Plots and Results (see **Delete Results** (p. 28)).

Delete Plots. Plots that have been created and stored in the database can be deleted and removed from the database.



Delete Results. Results can be removed from the database with this function. Both Result Cases and / or the results data associated with Result Cases can be deleted. Please note that any Result Cases deleted will cause Result Case selections to be reset.



STEP 1: Set the Action to Delete and the Object to Result Cases or Result Data from the Results application form.

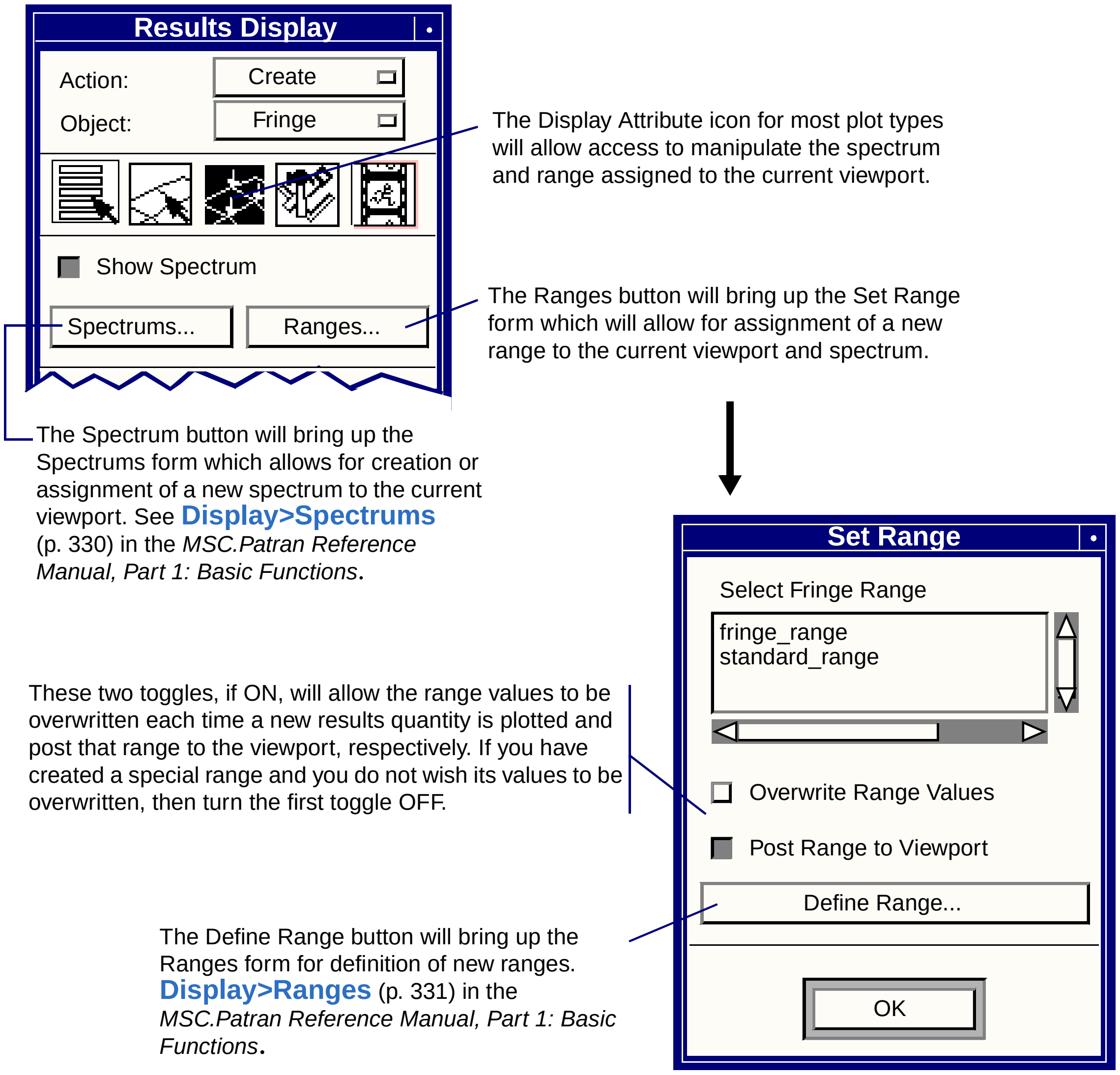
STEP 2: Select the Result Cases to be deleted. Use the shift key to select multiple plots and/or the control key to select non-continuous selections. If results data is being deleted, select all the results data and the Result Cases from which the results data are to be deleted.

STEP 3: Press the Apply button on the bottom of the form. The results will be deleted.

Spectrum/Range Control

A range is a set of numbers or range of numbers each assigned a specific color to be displayed in the viewport on a color spectrum bar. The colors in the spectrum bar and the number of ranges assigned to them correspond to the color bands plotted graphically on the finite element model to indicate levels of stress, displacement or other results quantities.

Selecting and manipulating the active range and/or spectrum is done in the Ranges and Spectrums forms, which are accessible by two different methods. Most plot types allow for manipulation of the range and spectrum directly from their Display Attributes form:



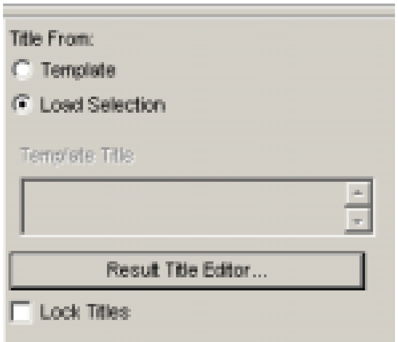
Access to the Spectrum and Range forms is also available under the Display pull-down menu on the main MSC.Patran form. Creation and modification of the actual spectrums and ranges is done in these forms which are described in **Display>Ranges** (p. 331) and **Display>Spectrums** (p. 330) in the *MSC.Patran Reference Manual, Part 1: Basic Functions.*

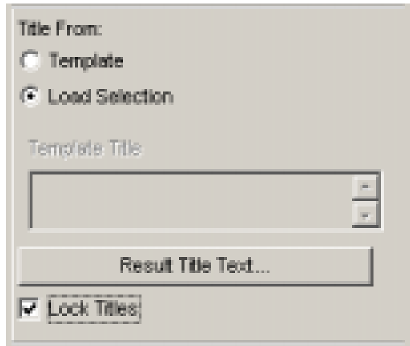
Things to note about ranges for the various types of result plots:

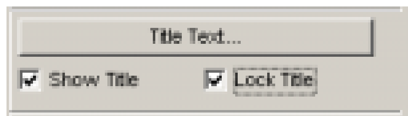
1. By default, a new range will be assigned to every plot created if an existing one has not been selected. The range will be assigned to the current viewport. The name of the range will be the plot type with the plot name concatenated, e.g., FRI_`default_Fringe`, VEC_`myvector`.
2. If you wish to assign a certain range to a new plot at creation time, simply select the range from the listbox as shown on the previous page. The range will remain as defined unless the Overwrite Range Values toggle has been set ON. If ON, then new maximum and minimum values will be calculated based on the result values and a new range calculated for the selected named ranged. This will permanently change the range until changed again; so care should be taken when using this option. This is true under the Modify Action also.
3. Although many plots may be displayed simultaneously, only one spectrum and range is available for display in the current viewport at any one time. By default the range of the last plot created or posted will be displayed. Any existing, posted plots will take on the color spectrum of the posted range. This can cause some plots to appear monochrome indicating that their result values are outside the range of the current spectrum, either above or below it. This is done to avoid confusion and misinterpretation of results.
4. Any existing range may be selected as being the active range by posting it to the screen. There are two ways to change the current range posted to any particular viewport. The first is under Viewport / Modify on the main MSC.Patran form. See **Viewport>Modify** (p. 268) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*. Using the Viewport / Modify or Range Update forms will assign the range to the currently posted plots (The spectrum legend can also be controlled. The second method is done directly in the Results application under the Post / Ranges action / object. See **Post/Unpost** (p. 25). A list of ranges is not supplied here, but a list of posted plots. By selecting one of the posted plots, its assigned ranged will be posted. Any other plots posted will update to reflect the posted range.
5. If you delete a plot, the associated range with the same name will NOT also be deleted. You will need to physically delete it under the Define Range form.

1.6 Results Title Editor

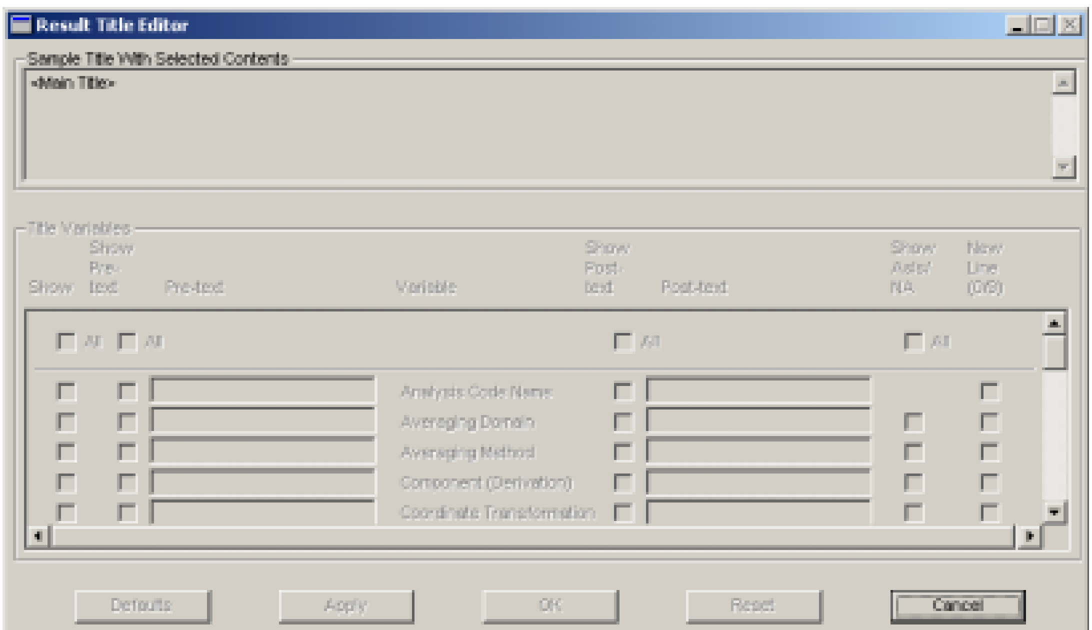
Results attributes forms that contain the Title Editor... or Results Title Editor button shown below open the Results Title Editor form when the button is pressed. If the Lock Titles checkbox is checked, the Title Editor button label changes to indicate that the Results Title Editor form will be opened in a readonly mode. In readonly mode the title may be viewed but not edited and the only active button is Cancel.



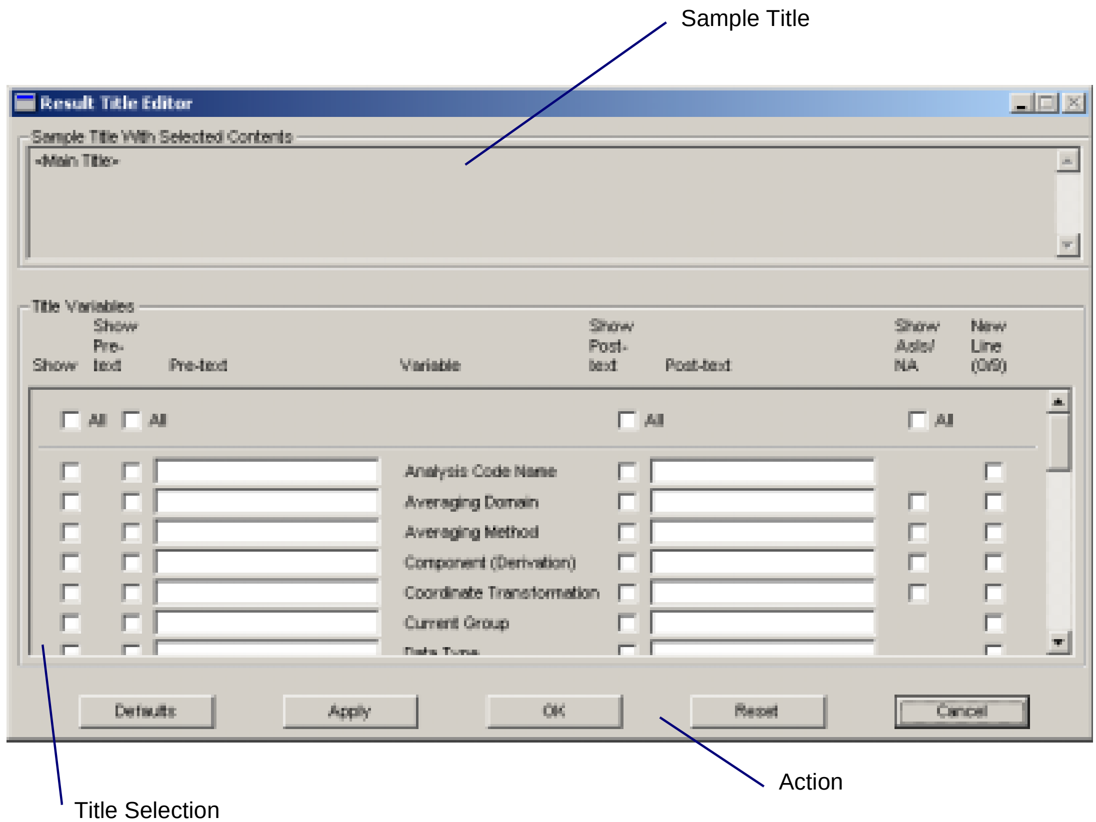




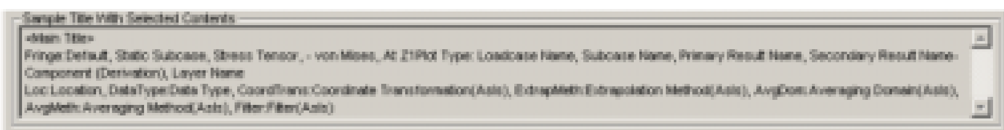
The readonly version of the Results Title Editor form is shown below.



The writable version of the Results Title Editor form is shown below.



The Results Title Editor form has three main regions. Each region, and its usage are described below.

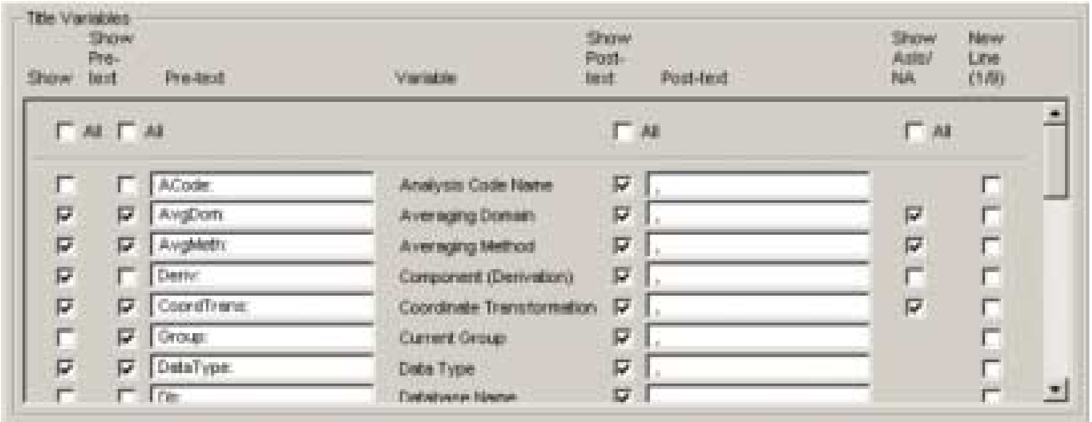


The **Sample Title** textbox shows the current title similar to how it will look on the Results plot. If the Main Title variable is enabled, the string <Main Title> is shown to indicate that the main title will appear. The Main Title is the first title line. It includes the MSC.Patran version and the date and time of the plot.

Any enabled Results variables, are replaced by the Variable description shown in the Title Selection region. Any enabled pre-text or post-text is shown as it appears in the Title Selection region. Any variables with enabled As Is / NA have (AsIs / NA) appended to the variable description. Any enabled new lines cause a new line to be started.

The text in this region is updated each time a change is made in the Title Selection region, either by switching a toggle or by pressing the keyboard Enter button while focus is in one of the databoxes.

This textbox is readonly, however the cursor may be used to set the insertion position, which is then used as the insertion point for subsequently enabled variables. This procedure is explained in the **Variable Insert Location** section.



The **Title Selection** region is where most of the user interaction will occur. Control over which variables and text are included in the title is provided here. At the top of this region a set of column headings describe the meaning of the items in the rows below. Due to the number of items, a scrollframe is used to keep the form size reasonable. The scrollframe has three regions each with a line separator. At the top there are controls to set all of the toggles in the middle region either ON or OFF. Controls for all variables except Main Title are in the middle region. The bottom region provides control for the Main Title and two General Text entries.

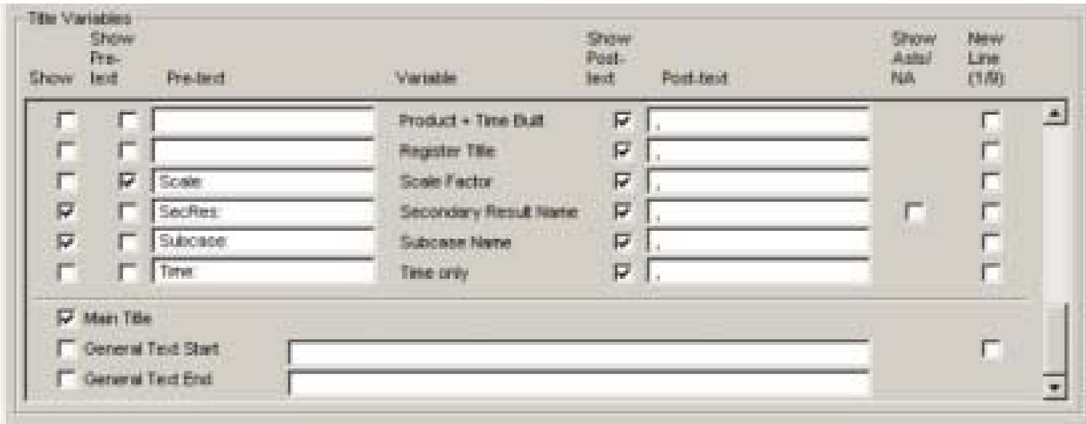
The toggles in the **Show** column control the display of the information in each row. If the toggle is off in a row, none of the information in that row is shown. If a toggle is on, the enabled information in that row is shown. The toggles in the **Show Pre-text** column control the display of any text entered in the corresponding Pre-text databox. The databoxes in the **Pre-text** column provide a means of entering text to precede each variable. In a given row, if the Show toggle and the Show Pre-text toggle are enabled, the text in the Pre-text databox will be inserted before the variable. The text in the **Variable** column provides an easy-to-recognize meaning for the variable in each row. It is also the text that appears in the Sample Title as the variable value.

The toggles in the **Show Post-text** column control the display of any text entered in the corresponding Post-text databox. The databoxes in the **Post-text** column provide a means of entering text to follow each variable. In a given row, if the Show toggle and the Show Post-text toggle are enabled, the text in the Post-text databox will be inserted after the variable.

Show As Is/NA only comes into consideration if a variable's value is AsIs or blank "". The toggles in the **Show As Is/NA** column control the display of the value of certain variables depending on their value. For example, if the Averaging Domain value is blank "" or AsIs, its value will not be shown unless its Show As Is/NA toggle is enabled. Note that many of the variables do not have a Show As Is/NA toggle. This is because a value of AsIs has no meaning for these variables.

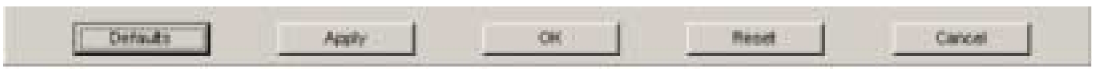
The toggles in the **New Line** column control the insertion of new line control variables. In a given row, if the Show toggle and the New Line toggle are enabled, a new line will be inserted after the variable and its post-text. Note that there is a 10-line limit to Results titles, which limits the number of new lines to nine. A counter is provided next to the New Line column heading for reference. No more than nine new lines will ever be entered into the title. No more than nine New Line toggles will ever be enabled in rows that also have the Show toggle enabled. Enabled New Line toggles in rows with disabled Show toggles do not affect the count.

All of the Show, Show Pre-text, Show Post-text and Show As Is / NA toggles in the middle region of the scrollframe may be turned ON or OFF by using the toggles labeled All in the top scrollframe region.



The bottom scrollframe region provides control for the Main Title and two General Text entries. The Main Title has already been described. The General Text entries provide a means of adding text to the title that is not associated to any variable. The General Text Start entry will appear at the start of the title while the General Text End entry will appear at the end of the title. Their Show toggles control their display.

When its parent form opens the Results Title Editor form, the current title is decoded and the Title Selection region toggles and databoxes are set. If no variables are found, the entire title is entered into the General Text Start databox.



The **Action** region has five buttons. The **Defaults** button sets the title to the default variables, pre-text's, post-text's, AsIs / NA and new line settings for the Results tool type. All current settings are overwritten. The **Apply** button applies the current title selection to the Results tool and leaves the form open. The **OK** button applies the current title selection to the Results tool and closes the form. The Apply button on the Attributes form must be selected to see the title update in the graphics viewport. The **Reset** button resets the title to the variables, pre-text's, post-text's, AsIs and new line settings it had when the form was last opened or when the Apply button was last selected. The **Cancel** button closes the form with no effect.

The settings.pcl preference:

```
pref_env_set_string( "results_title_editor_defaults", "my_defaults" )
```

may be used to override the standard defaults. If defined, and the function specified exists, the function will be called whenever the Defaults button is pressed. The prototype for this function is given below. An example of this function will be provided.

Variable Insert Location

Variables are inserted based on the current insert location. The insert location can be changed by placing the cursor into the Sample Title textbox and clicking to set the insertion point. The insertion point is indicated by a vertical bar. The editor determines which variable the insertion point is in. The insertion of a new variable will be immediately after the variable containing the insertion point. If the insertion point is never set, it will be at the end. If the insertion point is placed at the beginning of the textbox, the next variable inserted will be at the beginning. Turning a variable off and then on will remove it and then insert it back into its previous location. The insertion point is automatically updated such that subsequent insertions are placed after the previously inserted variable.

User-defined Defaults Function

To provide maximum flexibility to the user, a user-defined function may be provided to override any and all of the default title settings. Placing the following definition in the MSC.Patran settings.pcl file specifies a user-defined function.

```
pref_env_set_string( "results_title_editor_defaults", "my_defaults" )
```

Where `my_defaults` is the name of a function. If the `results_title_editor_defaults` preference is defined, and the specified function exists, the function will be called whenever the Defaults button is pressed. The prototype for this function is shown below. An example of this function will be provided.

```
FUNCTION my_defaults( tool_class, var, description, show, showPrefix, prefix, @
                         showSuffix, suffix, asIs, position, newLine )
/*
 * FUNCTION my_defaults
 * Returns default values for a given input variable.
 * All output variable values will already have been set to the Patran defaults.
 * This function needs only to modify those that the user wants modified.
 *
 * Input
 *   tool_class_in     class name of tool type (default == Fringe)
 *                     One of the following from "res_1_5_include.h":
 *
 * #define RES_TOOL_DEFORM_ATTR_CLASS_Q      "res_display_def_attr"
 * #define RES_TOOL_FRINGE_ATTR_CLASS_Q      "res_display_fri_attr"
 * #define RES_TOOL_2DCONTOUR_ATTR_CLASS_Q   "res_display_cont_attr"
 * #define RES_TOOL_3DCONTOUR_ATTR_CLASS_Q   "res_display_cont_3d_attr"
 * #define RES_TOOL_VECTOR_ATTR_CLASS_Q      "res_display_vec_attr"
 * #define RES_TOOL_TENSOR_ATTR_CLASS_Q      "res_display_ten_attr"
 * #define RES_TOOL_CURSOR_ATTR_CLASS_Q      "res_display_cur_attr"
 *
 *   var               A variable name
 *                     One of the following from "res_display.h":
 *
 * #define RES_DISP_TITL_VAR_DATE2                "$DATE2"
 * #define RES_DISP_TITL_VAR_DATE                 "$DATE"
 * #define RES_DISP_TITL_VAR_TIME                 "$TIME"
 * #define RES_DISP_TITL_VAR_PRODUCT              "$PRODUCT"
 * #define RES_DISP_TITL_VAR_PROD                 "$PROD"
 * #define RES_DISP_TITL_VAR_DB_NAME              "$DB_NAME"
 * #define RES_DISP_TITL_VAR_DB_PATH              "$DB_PATH"
 * #define RES_DISP_TITL_VAR_JOB_NAME             "$JOB_NAME"
 * #define RES_DISP_TITL_VAR_CODE_NAME            "$CODE_NAME"
 * #define RES_DISP_TITL_VAR_GV                   "$GV"
 * #define RES_DISP_TITL_VAR_LC_NAME              "$LC_NAME"
 * #define RES_DISP_TITL_VAR_SC_NAME              "$SC_NAME"
 * #define RES_DISP_TITL_VAR_PRES_NAME            "$PRES_NAME"
 * #define RES_DISP_TITL_VAR_SRES_NAME            "$SRES_NAME"
 * #define RES_DISP_TITL_VAR_LYR_NAME             "$LYR_NAME"
 * #define RES_DISP_TITL_VAR_LOCATION             "$LOCATION"
 * #define RES_DISP_TITL_VAR_PLOT_TYPE            "$PLOT_TYPE"
 * #define RES_DISP_TITL_VAR_DATA_TYPE            "$DATA_TYPE"
 * #define RES_DISP_TITL_VAR_DERIVATION_L         "$DERIVATION_L"
 * #define RES_DISP_TITL_VAR_DERIVATION           "$DERIVATION"
 * #define RES_DISP_TITL_VAR_COORD_TRANS          "$COORD_TRANS"
```

```
 * #define RES_DISP_TITL_VAR_EXTRAP_METH          "$EXTRAP_METH"
 * #define RES_DISP_TITL_VAR_AVG_DOM              "$AVG_DOM"
 * #define RES_DISP_TITL_VAR_AVG_METH             "$AVG_METH"
 * #define RES_DISP_TITL_VAR_SCALE_FACT           "$SCALE_FACT"
 * #define RES_DISP_TITL_VAR_FILTER               "$FILTER"
 * #define RES_DISP_TITL_VAR_CUR_GROUP            "$CUR_GROUP"
 * #define RES_DISP_TITL_VAR_FRINGE_STYLE         "$FRINGE_STYLE"
 *
 * Output
 *   description       description (text that appears under "Variable" label)
 *   show              controls whether variable is shown
 *   showPrefix        controls whether prefix is shown
 *   prefix            prefix
 *   showSuffix        controls whether suffix is shown
 *   suffix            suffix string
 *   asIs              controls whether asIs is shown
 *   position          position of variable in title starting with 1
 *   newLine           controls whether a new line will follow the variable
 */
   STRING tool_class_in[], var[], description[], prefix[], suffix[]
   LOGICAL show, showPrefix, showSuffix, asIs, newLine
   INTEGER position
```

Examples of User-defined Defaults Function

Two examples of user-defined functions are given in this section. The first is simple and changes only a few defaults. The second is more complex, and is similar to the function used by MSC.Patran. See the section **User-defined Defaults Function** (p. 35) for a description of the function arguments.

To use the simple function, place the following definition in the MSC.Patran settings.pcl file:

```
pref_env_set_string( "results_title_editor_defaults", @
" simple_user_default_function " )
```

and make the function available by either compiling it in MSC.Patran using !!input or compiling it outside of MSC.Patran into a .plb and using !!library in MSC.Patran.

```
FUNCTION simple_user_default_function( @
tool_class_in, var, description, show, @
            showPrefix, prefix, showSuffix, suffix, @
            asIs, position, newLine )

   STRING tool_class_in[], var[], description[], prefix[], suffix[]
   LOGICAL show, showPrefix, showSuffix, asIs, newLine
   INTEGER position

   /*
    * Simply change the prefix and description of
    * some of the variables.
    */
   SWITCH( var )
      CASE( RES_DISP_TITL_VAR_CODE_NAME )
         prefix = "My Language A Code:"
         description = " My Language Code Name"
      CASE( RES_DISP_TITL_VAR_CUR_GROUP )
         prefix = " My Language Group:"
         description = " My Language Group"
      CASE( RES_DISP_TITL_VAR_DB_NAME )
         prefix = " My Language DbName:"
         description = " My Language Database Name"
      CASE( RES_DISP_TITL_VAR_DB_PATH )
         prefix = " My Language DbPath:"
         description = " My Language Path Name"
   END SWITCH
```

```
END FUNCTION
```

To use the complex function, place the following definition in the MSC.Patran settings.pcl file:

```
pref_env_set_string( "results_title_editor_defaults", @
" complex_user_default_function " )
```

and make the function available by either compiling it in MSC.Patran using !!input or compiling it outside of MSC.Patran into a .plb and using !!library in MSC.Patran.

```
FUNCTION complex_user_default_function( tool_class_in, var, description, show, @
                                        showPrefix, prefix, showSuffix, suffix, @
                                        asIs, position, newLine )
   STRING tool_class_in[], var[], description[], prefix[], suffix[]
   LOGICAL show, showPrefix, showSuffix, asIs, newLine
   INTEGER position

   STRING tool_class[MAX_TEXT_LENGTH]

   SWITCH( tool_class_in )
      CASE( RES_TOOL_FRINGE_ATTR_CLASS_Q, @
            RES_TOOL_DEFORM_ATTR_CLASS_Q, @
            RES_TOOL_2DCONTOUR_ATTR_CLASS_Q, @
            RES_TOOL_3DCONTOUR_ATTR_CLASS_Q, @
            RES_TOOL_VECTOR_ATTR_CLASS_Q, @
            RES_TOOL_TENSOR_ATTR_CLASS_Q, @
            RES_TOOL_CURSOR_ATTR_CLASS_Q, @
            RES_TOOL_CURSOR_ATTR_CLASS_Q )
         tool_class = tool_class_in

      DEFAULT
         tool_class = RES_TOOL_FRINGE_ATTR_CLASS_Q
   END SWITCH


   /* These are the most common. */

   show = TRUE
   showPrefix = FALSE
   showSuffix = TRUE
   suffix = ", "
   asIs = FALSE
   position = 0
   newLine = FALSE

   /* Set show. */

   SWITCH( var )
      CASE( RES_DISP_TITL_VAR_CODE_NAME,    @
            RES_DISP_TITL_VAR_CUR_GROUP,    @
            RES_DISP_TITL_VAR_DB_NAME,      @
            RES_DISP_TITL_VAR_DB_PATH,      @
            RES_DISP_TITL_VAR_DATE,         @
            RES_DISP_TITL_VAR_DATE2,        @
            RES_DISP_TITL_VAR_FRINGE_STYLE, @
            RES_DISP_TITL_VAR_GV,           @
            RES_DISP_TITL_VAR_JOB_NAME,     @
            RES_DISP_TITL_VAR_PRODUCT,      @
            RES_DISP_TITL_VAR_PROD,         @
            RES_DISP_TITL_VAR_DATA_TITLE,   @
            RES_DISP_TITL_VAR_SCALE_FACT,   @
            RES_DISP_TITL_VAR_TIME )

         show = FALSE
```

```
   CASE( RES_DISP_TITL_VAR_FILTER )
      IF( tool_class == RES_TOOL_DEFORM_ATTR_CLASS_Q ) THEN
         show = FALSE
      END IF
END SWITCH

/* Set showPrefix. */

SWITCH( var )
   CASE( RES_DISP_TITL_VAR_AVG_DOM,      @
         RES_DISP_TITL_VAR_AVG_METH,     @
         RES_DISP_TITL_VAR_COORD_TRANS,  @
         RES_DISP_TITL_VAR_CUR_GROUP,    @
         RES_DISP_TITL_VAR_DATA_TYPE,    @
         RES_DISP_TITL_VAR_EXTRAP_METH,  @
         RES_DISP_TITL_VAR_FILTER,       @
         RES_DISP_TITL_VAR_FRINGE_STYLE, @
         RES_DISP_TITL_VAR_GV,           @
         RES_DISP_TITL_VAR_JOB_NAME,     @
         RES_DISP_TITL_VAR_LOCATION,     @
         RES_DISP_TITL_VAR_SCALE_FACT )

      showPrefix = TRUE
END SWITCH

/* Set asIs. */

SWITCH( var )
   CASE( RES_DISP_TITL_VAR_AVG_DOM,      @
         RES_DISP_TITL_VAR_AVG_METH,     @
         RES_DISP_TITL_VAR_COORD_TRANS,  @
         RES_DISP_TITL_VAR_EXTRAP_METH,  @
         RES_DISP_TITL_VAR_FILTER )

      asIs = TRUE
END SWITCH

/* Set position. */

SWITCH( tool_class )
   CASE( RES_TOOL_DEFORM_ATTR_CLASS_Q )
      SWITCH( var )
         CASE( RES_DISP_TITL_VAR_PLOT_TYPE )
            position = 1
         CASE( RES_DISP_TITL_VAR_LC_NAME )
            position = 2
         CASE( RES_DISP_TITL_VAR_SC_NAME )
            position = 3
         CASE( RES_DISP_TITL_VAR_PRES_NAME )
            position = 4
         CASE( RES_DISP_TITL_VAR_SRES_NAME )
            position = 5
         CASE( RES_DISP_TITL_VAR_LYR_NAME )
            position = 6
         CASE( RES_DISP_TITL_VAR_LOCATION )
            position = 7
         CASE( RES_DISP_TITL_VAR_DATA_TYPE )
            position = 8
         CASE( RES_DISP_TITL_VAR_COORD_TRANS )
            position = 9
         CASE( RES_DISP_TITL_VAR_EXTRAP_METH )
            position = 10
         CASE( RES_DISP_TITL_VAR_AVG_DOM )
            position = 11
         CASE( RES_DISP_TITL_VAR_AVG_METH )
            position = 12
```

```
         END SWITCH

      DEFAULT
         SWITCH( var )
            CASE( RES_DISP_TITL_VAR_PLOT_TYPE )
               position = 1
            CASE( RES_DISP_TITL_VAR_LC_NAME )
               position = 2
            CASE( RES_DISP_TITL_VAR_SC_NAME )
               position = 3
            CASE( RES_DISP_TITL_VAR_PRES_NAME )
               position = 4
            CASE( RES_DISP_TITL_VAR_SRES_NAME )
               position = 5
            CASE( RES_DISP_TITL_VAR_DERIVATION_L )
               position = 6
            CASE( RES_DISP_TITL_VAR_LYR_NAME )
               position = 7
            CASE( RES_DISP_TITL_VAR_LOCATION )
               position = 8
            CASE( RES_DISP_TITL_VAR_DATA_TYPE )
               position = 9
            CASE( RES_DISP_TITL_VAR_COORD_TRANS )
               position = 10
            CASE( RES_DISP_TITL_VAR_EXTRAP_METH )
               position = 11
            CASE( RES_DISP_TITL_VAR_AVG_DOM )
               position = 12
            CASE( RES_DISP_TITL_VAR_AVG_METH )
               position = 13
            CASE( RES_DISP_TITL_VAR_FILTER )
               position = 14
         END SWITCH
   END SWITCH

   /* Set show suffix. */

   SWITCH( var )
      CASE( RES_DISP_TITL_VAR_LYR_NAME, @
            RES_DISP_TITL_VAR_FILTER )
         showSuffix = FALSE
      CASE( RES_DISP_TITL_VAR_AVG_METH )
         IF( tool_class == RES_TOOL_DEFORM_ATTR_CLASS_Q ) THEN
            showSuffix = FALSE
         END IF
   END SWITCH

   /* Set suffix. */

   SWITCH( var )
      CASE( RES_DISP_TITL_VAR_PLOT_TYPE )
         suffix = ": "
      CASE( RES_DISP_TITL_VAR_GV )
         suffix = "Mode ,"
   END SWITCH

   /* Set newLine. */

   SWITCH( var )
      CASE( RES_DISP_TITL_VAR_LYR_NAME )
         newLine = TRUE
   END SWITCH

END FUNCTION
```

CHAPTER 2

Quick Plots

- Overview
- Quick Plot Usage
- Animation Notes
- Examples of Usage

2.1 Overview

Quick Plot is the default object of the Create action in the Results application and is designed to meet the needs of 80-90% of all postprocessing.

Quick Plot allows a user to quickly display a deformed plot, a scalar fringe plot, or a modal or ramped style animation. Both vector and scalar data can be animated, either separately or simultaneously. Transient animations are not supported from Quick Plot. For transient animations or more complicated postprocessing needs such as coordinate transformations or other derivations, the user will need to change the object to the specific type of plot to be created.

Quick Plot is designed to use all default settings for display attributes, target entities, and other options. By default the plot will display on everything posted in the current viewport. The display attributes for deformations, fringe plots and animations are also the default settings used under the Display Attributes forms for deformations **(p. 50)** and fringes **(p. 68)** plots. Animation defaults are explained later in this chapter. See **Animation Notes** (p. 36).

The goal of Quick Plot is to give the user a quick, meaningful plot without having to worry about making sure all attributes, settings, and coordinate transformations are set properly. When fringe plots of element components are displayed using Quick Plot, the component data is oriented in a reasonable coordinate system to assure a meaningful plot whenever possible. The definitions of reasonable coordinate systems are as follows:

1. Element based result component data will be left in the coordinate system in which they were imported with the exception of two dimensional elements (plates, shells) that are oriented in an element connectivity based system. In this particular case, the component data will be transformed to the Projected Global system for fringe plot display.
2. For one dimensional (1D) and three dimensional (3D) elements, the Projected Global system is the Global system and therefore no projection is performed.
3. If data are in an Unknown coordinate system, no transformation will occur. If the Unknown system happens to be an element system, then nodal averaging may not be correct, resulting in a meaningless plot. This puts the burden on the user to ensure correctness.
4. If data are in the Global system, a user defined Local, or an analysis specific projected system, then no transformations will occur.
5. Nodal based results are not effected by any transformations.

For more detailed definition of these coordinate systems see **Coordinate Systems** (p. 305).

Display attributes are accessible from Quick Plot for modification as explained in the next section.

No modifications can be made directly in Quick Plot for other plot options such as coordinate transformations as explained above. However you can change the Quick Plot default for two commonly used parameters. These parameters are the Coordinate Transformation and the Averaging Method. They are modified in the `settings.pcl` file. See **The settings.pcl file** (p. 41) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

For Coordinate Transformations use the setting

```
pref_env_set_string( “result_quick_transform”, “Default”)
```

Valid values are `Default`, `Global`, `CID`, `ProjectedCID`, `None`, `Material`, `ElementIJK`.

To change the default Averaging Method, use the setting

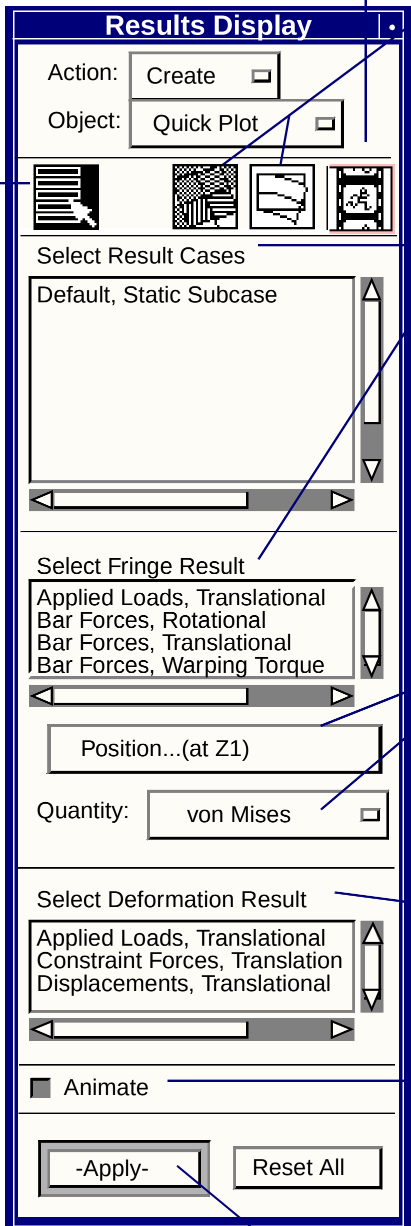
```
pref_env_set_string( “result_quick_avg_method”, “DeriveAverage”)
```

Valid values are `DeriveAverage`, `AveragDerive`, `Difference`, `Sum`.

2.2 Quick Plot Usage

This is the default form that appears when the Results application is selected. Use this form to plot fringes, deformations and do simple animations.

The default appearance of this form is as shown to allow selection of results for subsequent plotting. The form can also be changed by selecting the right most icon from which animation options can be set. Selecting the left most icon will toggle the form back to select results. See **Animation Notes** (p. 36).



These two button icons are for access to Display Attribute options. See **Display Attributes** (p. 50) for deformations, and **Display Attributes** (p. 68) for fringes.

Select the desired Result Case. This will fill out the Fringe Result and Deformation Result listboxes below. If this listbox is empty, no results exist in the database. Results are imported from the Analysis application.

Select a result type from which to make a fringe plot, if desired.

If layered results exist a subordinate form of the existing layer positions can be opened. See **Result Layer Positions** (p. 16).

A menu to display the valid transformation derivations. Used when a Vector or Tensor result is chosen in the Fringe Result listbox above. If the selected fringe result is a scalar value, this menu does not appear.

The possible transformations are **(p. 286)**:

1. Vector to Scalar**:** Magnitude, X component, Y comp., Z comp.
2. Tensor to Scalar: von Mises, XX, YY, ZZ, XY, YZ, XZ, Minor, Intermediate, Major, Hydrostatic, 1st Invariant, 2nd Invariant, 3rd, Invariant, Tresca, Max Shear, Octahedral.

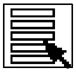
Be aware that for certain element based results, coordinate transformations may automatically occur to produce a meaningful plot. See **Overview** (p. 32) for an explanation.

Select a displacement results from which to make a deformation plot, if desired. If a fringe plot is also selected, it will appear on the deformed structure.

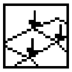
When toggled ON, the selected results will be animated. When the Animate toggle is turned ON, this button appears for selecting animation options. These are described in **Animation Options** (p. 37). It is not necessary to open this subordinate form to create an animation. Defaults will be used.

Creates the plot and/or animation. The Apply button can be pressed with the form in either state (results selection or animation options). More details follow on the next page.

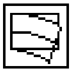
For more information on the use of these button icons, see the appropriate section.



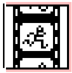
Selecting Results (p. 15)



Fringe **Display Attributes** (p. 68)



Deformation **Display Attributes** (p. 50)



Animation Options (p. 156).

The derivation of each scalar quantity from either a tensor or vector that is available in the Quick Plot application is described in detail in **Derivations** (p. 286).

It is important to note also that when deriving a scalar quantity for a fringe plot from an element based vector or tensor that results are averaged at the nodes due to the contributions from the surrounding elements. The default is to derive the desired quantity from the tensor or vector quantities (such as von Mises), then to average at the nodes. By default the averaging is done over all entities. This default Averaging Method can be changed with a `settings.pcl` parameter. See **Overview** (p. 32).

When multiple layers exist for a specific Result Case and quantity, five additional options are also presented to the user from the Position button. These are Maximum, Minimum, Average, Sum, and Merge. By selecting one of these layers, the minimum, maximum, average, sum or merging of all layers will be calculated for display in the subsequent plot. These plots may be more computationally intensive and take longer to display the final plot due to the results extraction of the maximum, minimum or average for all layers. The results of these derivations are not stored in the database. Use the Create / Results action and object to perform this task. See **Result Layer Positions** (p. 16) for more details.

Fringe plots and deformation plots created from the Quick Plot form are assigned default names that can be seen when the object is changed to Deformation or Fringe or from the Post and Delete forms. These names are `default_Deformation` and `default_Fringe`. The Quick Plot form will always operate on these named plots. If more than one viewport is open then the default names are incremented such as `default_Deformation`2, `default_Deformation`3, etc.

2.3 Animation Notes

There are two forms for controlling animations from the Results Quick Plot form and the manner in which the animations display on the graphics screen. These forms are described in this section. Transient animations are not allowed from the Quick Plot form.

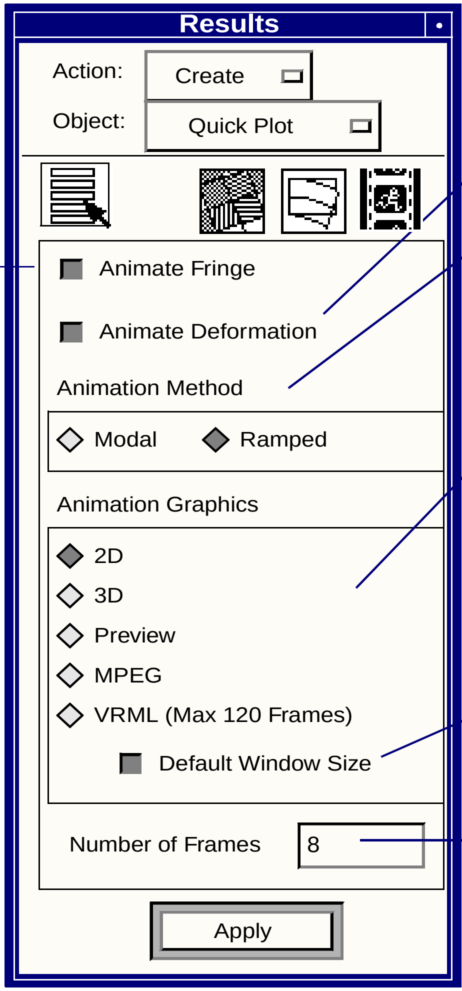
The first form is for controlling animation attributes, such as the number of frames, or the animation method and is invoked before an animation is created. This form is described in **Animation Options** (p. 37).

The second form is for actual control of the animation as it is animating and is the same for all animations. It is described fully in **Animation Control** (p. 158). The form remains on the screen until either the Cancel button is pressed or the user presses the Abort (the hand) or Cleanup (the broom) icons on the main form.

Animation Options

This form is accessible from the Quick Plot form in the Results application when the Animate toggle is turned ON. It allows for modification of the display attributes of the animation before the animation is created. The default setting is shown in the following form.

If this toggle is OFF and a fringe result has been selected to animate with a deformed shape, the resulting fringe plot will appear static during the animation. It will not change from frame to frame. If ON, it will change frame to frame if a fringe result has been selected from the Quick Plot form. The default is ON.



If this toggle is ON, then the deformation results selected will animate (change from frame to frame). If OFF, it will not animate. This is really only applicable when a fringe is being animated with a deformed plot. Turning this toggle OFF will allow the fringe to animate while the deformation remains static. Needless to say, only one of these two toggles can be OFF at any one time.

Modal animation allows animations from +MAX to -MAX, whereas Ramped animation only ranges from ZERO to +MAX of the given results values.

2D animation allows for animation frames to be created only in a 2D plane. This simply means that dynamic rotation with the mouse is not possible without recreating all the animation frames again. 3D animation allows for dynamic rotation with the mouse. The advantage of 2D over 3D animation is speed, although this is highly hardware and model size dependent. Preview will simply step through each frame and then stop. This is best used for transient animation. MPEG or VRML allows for output of animations to these standard formats, in addition to the display in the viewport. See **File>Images** (p. 197) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Controls the window size of image output when the MPEG or VMRL outputs are requested.Turning this toggle ON sets the output file window size to an acceptable size for most image view programs.

Enter the number of frames for the animation to build. The default is 7. There is no currently imposed limit to the number of frames that may be used. The more frames used, the smoother the animation will appear, however practical limits such as available memory and model size will quickly dictate the limit.

The Apply button will create the plot or animation. Note that the plot or animation will occur by pressing the Apply button in either state that the form appears. In order for there to be any animation the Animate toggle must be turned ON from the Select Results form. Otherwise only a deformed or static fringe plot will appear.

2.4 Examples of Usage

All examples assume the Action is set to Create and the Object is set to Quick Plot.

Create only a Fringe Plot of a Scalar Result

1. From the Select Results form (left most icon) select the Result Case from the first listbox.

2. Select the Fringe Result from the second listbox.

3. (Optional) If the Fringe Result contains more than one layer, select the layer using the Position button that appears below the Fringe Result listbox. The first layer will automatically be selected by default.

4. (Optional) If the Fringe Result is a vector or tensor quantity, select the scalar Quantity to be derived for the fringe. The default for tensors data is von Mises, and for vector data, is Magnitude.

5. Press the Apply button with the Animate toggle OFF.

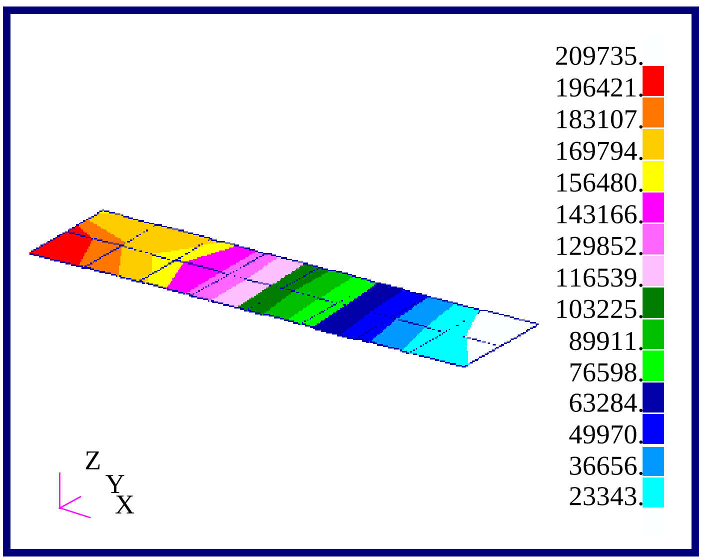


Figure 2-1 Fringe Plot of Stresses in a Cantilever Plate.

Create only a Deformation Plot

1. From the Select Results form (left most icon) select the Result Case from the first listbox.
2. Select the Deformation Result from the bottom listbox.
3. (Optional) Be sure that no Fringe Result has been selected in the Fringe Result listbox. Deselect it if one has.
4. Press the Apply button with the Animate toggle OFF.

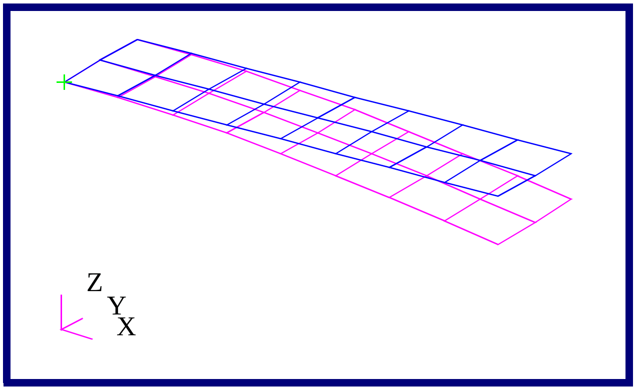


Figure 2-2 Deformation Plot of Cantilever Plate with Undeformed Shape.

Create a Fringe on a Deformed Plot

1. From the Select Results form (left most icon) select a Result Case from the first listbox.

2. Select a Fringe Result from the second listbox.
3. (Optional) If the Fringe Result contains more than one layer, select the layer using the Position button that appears below the Fringe Result listbox.The first layer will automatically be selected by default.

4. (Optional) If the Fringe Result is a vector or tensor quantity, select the scalar Quantity to be derived for the fringe. The default for tensors data is von Mises, and for vector data, is Magnitude.

5. Select a Deformation Result from the bottom listbox.
6. Press the Apply button with the Animate toggle OFF.

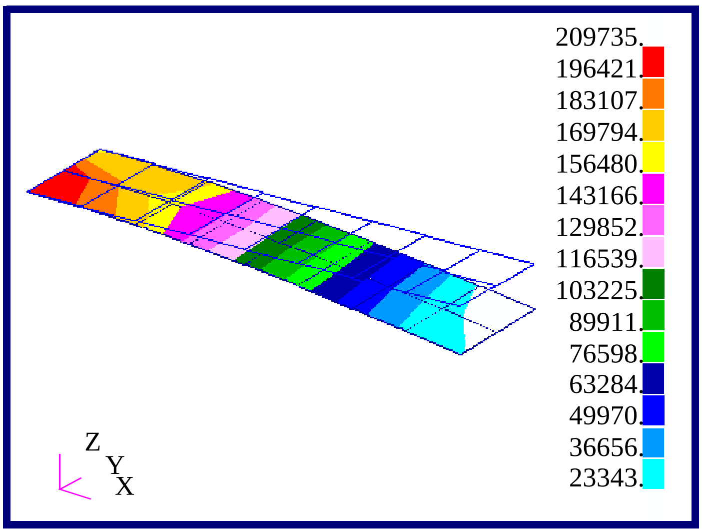


Figure 2-3 Fringe Plot on a Deformation Plot of Cantilever Plate with Undeformed Shape.

Create a Maximum Fringe from Multiple Layers

1. From the Select Results form (left most icon) select a Result Case from the first listbox.
2. Select a Fringe Result from the second listbox.
3. The Fringe Result must contain more than one layer. Select all the layers using the Position button that appears below the Fringe Result listbox.

   Position...(at Z1)

4. From the form that appear when selecting positions, change the Option to Maximum.

   Option: Maximum

5. (Optional) If the Fringe Result is a vector or tensor quantity, select the scalar Quantity to be derived for the fringe. The default for tensors data is von Mises, and for vector data, is Magnitude.

   Quantity: von Mises

6. Press the Apply button with the Animate toggle OFF. A fringe plot will result by performing a maximum comparison and extraction of the selected layers for the requested scalar quantity.

   Apply

Animate a Mode Shape

1. From the Select Results form (left most icon) select the Result Case from the first listbox.
2. Select the Deformation Result from the bottom listbox.
3. (Optional) Be sure that no Fringe Result has been selected in the Fringe Result listbox. Deselect it if one has.
4. Turn the Animate toggle ON. Modal animations are the default.

   Animate

5. Press the Apply button.

   Apply

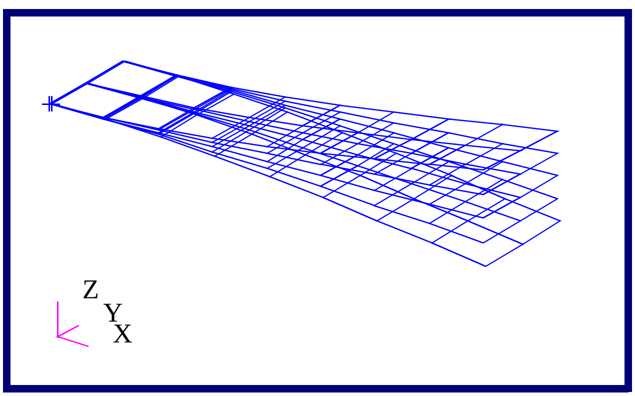
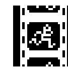


Figure 2-4 Modal Animation of Cantilever Plate.

Animate a Deformed Shape

1. Follow the previous example of **Animate a Mode Shape** (p. 41) up through step **4.**
2. Turn the Animate toggle ON.

   Animate
3. Press the Animation Options button icon.

   
4. Change the Animation Method from Modal to Ramped. This will allow animation from ZERO to +MAX as opposed to the default -MAX to +MAX of the selected results quantities.

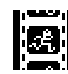
   Ramped
5. Press the Apply button.

   Apply

Animate a Fringe Result and a Deformation Simultaneously

1. Follow the procedure to **Create a Fringe on a Deformed Plot** (p. 40) above.
2. Turn the Animate toggle ON.

   Animate
3. (Optional) Change any necessary Animation Options.

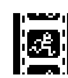
   
4. Press the Apply button. Both the fringe and deformation plots will animate together in a linear fashion.

   Apply

Animate a Fringe Plot Only

1. Follow the procedure to **Create only a Fringe Plot of a Scalar Result** (p. 38). Make sure that no deformation plot has been requested.
2. Turn the Animate toggle ON.

   Animate
3. (Optional) Change any necessary Animation Options.

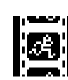
   
4. Press the Apply button.

   Apply

Animate a Deformed Plot or Mode Shape with a Static Fringe

1. Follow the procedure to **Create a Fringe on a Deformed Plot** (p. 40) above through step **5.**
2. Turn the Animate toggle ON.

   Animate
3. Press the Animate Options icon button.

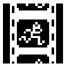
4. Turn the Animate Fringe toggle OFF on the Animate Options form.

5. Press the Apply button. The fringe plot will remain static as the deformation animates.

Animate a Fringe on a Static Deformed Plot

1. Follow the procedure to **Create a Fringe on a Deformed Plot** (p. 40) above through step **5.**
2. Turn the Animate toggle ON.

3. Press the Animate Options icon button.



4. Turn the Animate Fringe toggle OFF on the Animate Options form.

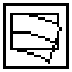
5. Press the Apply button. The fringe will animate on the static deformed shape of the model.

Change Display Attributes of a Deformed Plot, Fringe Plot or Animation

Follow any of the procedures described above to create a deformation or fringe plot or animation or combination thereof. But before pressing Apply do the following:

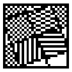
1. (Optional) Change deformation attributes by pressing the Deform Attributes button icon.

   For example, change the Line Style to Dashed for the Undeformed geometry.



2. (Optional) Change fringe attributes by pressing the Fringe Attributes button icon.

   For example change the fringe Style to Element Fill.



3. Press the Apply button.

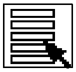
CHAPTER

3

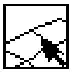
Deformation Plots

3.1 Overview

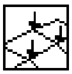
For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). To specifically make or modify a deformation plot, select Create or Modify from the Action pull-down menu on the Results application form; and select Deformation from the Object pull-down menu.



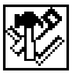
Selecting Results (p. 16)



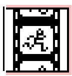
Target Entities (p. 58).



Display Attributes (p. 60).



Plot Options (p. 62).



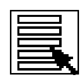
Animation Options (p. 156).

There is only a slight difference between Create and Modify. The main difference is that Create must be used to make a new deformation plot and Modify is used to change an existing one. If you try to modify an existing plot with Create you will be asked for overwrite permission, whereas Modify assumes that the action is desired, so no overwrite permission is requested.

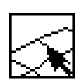
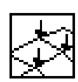
Toggles the form to select results for deformation plots. This is the default mode of the Deformation form.

For both Modify and Create the same basic operations and options are available. To create or modify a deformed plot the following basic steps must be followed:

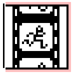
1. Set the Action to Create or Modify and the Object to Deformation.
2. Select a Result Case or Cases from the Select Result Case(s) listbox. See **Selecting Results** (p. 16) for a detailed explanation of this process as well as **Filtering Results** (p. 19).



3. Select a deformation result from the Select Deformation Result listbox.
4. At the bottom of the form, indicate whether the deformation plot is to be plotted in Resultant or individual Component form. Individual components are indicated as XX, YY, or ZZ which for cylindrical and spherical coordinate systems translates to r, θ, Z and r, ϕ, θ, respectively. Toggle the component(s) that you don't wish to plot OFF.
5. Optionally select the target entities, change display attributes, or invoke other plot options by changing these settings using the three middle icons at the top of the form. These are described in detail later in this chapter.


6. If animation is desired, turn the Animate toggle ON in the main form where results are selected and optionally change animation options with the right most icon at the top of the screen. For detailed explanations of animation options see **Animation Options** (p. 156) and **Animation Control** (p. 158)



7. Press the Apply button when ready to create the deformation plot.

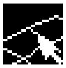
To modify an existing deformation plot, simply follow the above procedure with the Action set to Modify. However, you must first select an existing plot using the Existing Deformation Plots button on the main form where results are selected. When an existing deformation plot is selected, all results, attributes, and options in the various widgets associated with that plot are updated to reflect that plot's settings. You may then proceed to modify the plot.

By default a deformation plot with the name `default_Deformation` will be created unless the user specifically gives a different name. Multiple deformation plots can only be created and posted by giving separate names. Multiple deformation plots can be posted to the same viewport or to separate viewports. Each viewport can have its own set of deformation plots or other plot types posted. This is also true for animation of these plots. Only animation in the same viewport will be synchronized.

Each plot can have its own attributes. Each plot can also target or be displayed on separate entities and have its own associated options. These are detailed in the next sections.

3.2 Target Entities

Deformation plots can be displayed on various model entities. By default deformation plots are displayed using the entire model displayed in the current viewport. To change target entity selection for deformed plots on the Results Display form, set the Object to Deformation, and press the Target Entities selection button.



Toggles the form to select target display entities for deformation plots.

The following table describes which entities deformation plots can target:

Entity	Description
Current Viewport	By default all deformation plots are displayed on all finite element entities displayed in the currently active viewport (the entire displayed model).
Nodes	Individual nodes may be selected on which to display the deformed shape. You may type in any node numbers manually or by selecting them graphically from the screen. Be sure to include the word Node in front of the IDs you type in manually, (i.e., Node 1 5 55 100 etc.). To select all nodes use the syntax “Node 1:#.”
Elements	Individual elements may be selected on which to display the deformed shape. You may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.). To select all elements use the syntax “Elem 1:#.”
Groups	Deformation plots can be limited to only selected groups. A listbox will appear allowing selection of the groups to which the deformation plot will be applied. This is handy in that the same finite element entities can belong to multiple groups. Only those groups selected will be displayed in a deformed condition while all other non-selected groups will remain in their undeformed shape and retain their own display attributes.
Materials	Deformation plots can be targeted at only those finite elements which have certain material properties assigned to them. A listbox will appear allowing selection of the materials for whose elements will be displayed in a deformed shape.
Properties	Deformation plots can be targeted at only those finite elements which have certain element properties assigned to them. A listbox will appear allowing selection of the properties for whose elements will be displayed in a deformed shape.
Element Types	Deformation plots can be limited to only certain element types also.

In addition to displaying a deformation on selected entities, you may also specify the entity attributes. These are explained in the following table:

Attribute	Description
Elements	This is the default attribute. This simply means that deformation plots will show the nodes in their deformed positions with the element connectivity following them, thus showing a true deformed shape.
Nodes	Nodes as the attribute will only display the nodes in their deformed state without the element connectivity. The nodes will be displayed as small circles for better viewing.

Important: Once a target entity has been selected, it will remain the target entity for the deformation plot until the user physically changes it.

3.3 Display Attributes

Deformation plots can be displayed in various forms. Display attributes for deformed plots are accessible by pressing the Display Attributes selection button on the Results application form with the Object set to Deformation.

Toggles the form to change display attributes for deformation plots.

The following table describes in detail the deformation display attributes which can be modified:

Attribute	Description
Deformed/Undeformed Color	The deformed and undeformed plots can be colored in any of 16 distinct colors as displayed by this color selection widget. The deformed and undeformed plots can have different colors assigned to them.
Render Style	Render styles are Wireframe, Free Edge, Hidden Line, and Shaded and can be applied to both the deformed and undeformed shapes. The default is Wireframe which displays all visible finite elements. Free Edge displays only those edges that pass the feature angle setting. Hidden Line hides finite elements that are behind other. Shaded will be displayed with the selected color setting.
Line Style	The styles of the lines plotted for the deformed and undeformed shapes can be set to a solid line or variations of dotted lines.
Line Width	The thickness of the lines in the deformed and undeformed plots is set with this attribute setting.
Scale Interpretation	The visual amount of deformation is set with this parameter. The deformation can be scaled relative to the model size (Fraction of Model Size) or can be a truly proportional representation (True Multiplier) of the actual values. The default is Fraction of Model Size.
Scale Factor	The scale factor for both the Fraction of Model Size and True Multiplier are set in this databox. The defaults are 10% (0.1) for Fraction of Model Size and 100% (1.0) for True Multiplier.
Show Undeformed	This is a toggle to control the display of the undeformed shape. If the toggle is ON then the undeformed shape will also be plotted along with the deformed shape. If the toggle is OFF, then no undeformed shape will be plotted. Bear in mind that only one undeformed shape can be displayed in the same viewport at the same time if they are targeted at the same entities. You may not see changes to the undeformed shape if another plot has an undeformed shape plotted also.
Title Editor	Selecting this button opens a form that allows the deformation plot title to be edited. See Results Title Editor (p. 31).

Attribute	Description
Show Title	If this toggle is ON, then a title for the deformed plot is displayed. Otherwise no title is displayed.
Lock Title	If this toggle is ON, then the title for the deformed plot is not modified by results form selections. Otherwise some results form selections modify the title.
Maximum Label Display	If this toggle is ON, then the maximum value in the selected results set is displayed in the viewport. Otherwise it is not displayed. For transient results or when multiple subcases have been selected, the maximum value is the maximum encountered in all the subcases.
Label Style	Label styles may be changed such as the label color, format (fixed, integer, or exponential), and the number of significant digits.

Important:	Once display attributes have been selected, they will remain in effect for the deformed plot until the user physically changes them. Also changes to undeformed tool attributes may not be visible if multiple deformation plots are posted to the same viewport.

3.4 Plot Options

Deformation plots have various options. These options for deformed plots are accessible by pressing the Plot Options selection button on the Results application form with the Object set to Deformation.

 Toggles the form to select plot options for deformation plots.

The following table describes the deformation plot options which can be modified:

Option	Description
Coordinate Transformation	Vector results to display deformed plots can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran Global system, and the nodal (analysis) coordinate system. See **Coordinate Systems** (p. 305) for a definition of each of these coordinate systems. The default is no transformation. Resulting deformation plots should look the same no matter what coordinate system they are in as long as the resultant or all components are plotted. Where coordinate transformations play a role for deformed plots is when individual components are masked out and then transformed to another coordinate system.
Scale Factor	An additional scale factor can be given to scale the results plot above and beyond the scale factor available in the Tool Attributes form. This scale factor has the effect of simply scaling the results up or down by the specified amount.
Complex No. as	If complex results exist this option will be visible and specifies how and what you would like to view from results that exist as complex numbers. The options are Magnitude, Phase, Real, Imaginary, and Angle. It is not recommended to calculate invariants (e.g., von Mises) from complex results because the phase is not accounted for. Phase (this is specific to Deformation Plots, Fringe Plots, and Marker Plots) If the user selects Phase their results will be generated in degrees.
Phase	If Phase is selected, results will be generated in degrees. This is specific to Deformation Plots, Fringe Plots, and Marker Plots.
Existing Deformation Plots	This listbox displays all existing deformation plots. You may select one of these plots from the listbox and all settings of that plot including plot attributes, target entities, option, and selected results will be restored. This is an easy mechanism to help make many plots with the settings of an existing plot without modifying the selected plot. When the Action is set to Modify, this listbox appears under the Select Results display of the Results application form also.

Option	Description
Save Deformation Plot As	Deformation plots can be saved by name and recalled later for graphical display. Multiple deformed plots can be saved in the database and displayed simultaneously. Be aware that when multiple deformed plots are display simultaneously there could be some display problems. These deformation plots can be posted / unposted and deleted as explained in **Post/Unpost** (p. 25) and **Delete** (p. 27) respectively. Once a plot has been created and named it retains all results, attributes, target entities, and options assigned to it. If no plot name is specified a default is created called `default_Deformation`. As long as no plot name is specified, the `default_Deformation` will be overwritten each time a plot is created or modified.

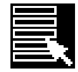
Important: Once plot options have been selected, they will remain in effect for the deformed plot until the user physically changes them.

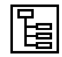
3.5 Examples of Usage

The following are some typical scenarios for usage of the Deformation plot tool. These instructions assume that the Action is set to Create and the Object is set to Deformation unless otherwise specified.

Create a Simple Static Deformation Plot

1. From the Select Results form (left most icon) select a Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.





2. Select the Deformation Result from the next listbox.

3. Press the Apply button with the Animate toggle OFF.

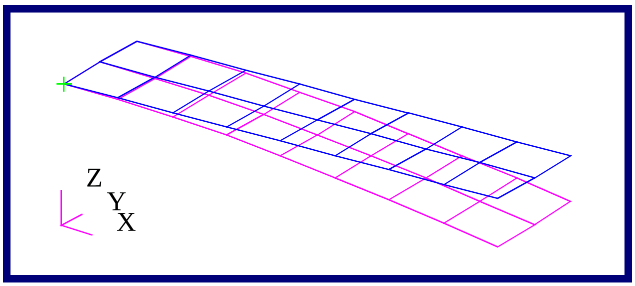
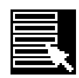


Figure 3-1 Deformation Plot of Cantilever Plate with Undeformed Shape.

Animate a Mode Shape

1. From the Select Results form (left most icon) select a Result Case from the first listbox.



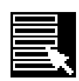
2. Select the Deformation Result from the next listbox.

3. Turn the Animate toggle ON.

4. Press the Apply button. Modal animations are the default therefore it is unnecessary to change any animation options.

Animate a Static Deformation

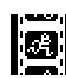
1. From the Select Results form (left most icon) select the result case from the first listbox.



2. Select the Deformation Result from the next listbox.

3. Turn the Animate toggle ON.

4. Press the Animation Options icon button.



5. Change the Animate By option pulldown menu to Ramp.

6. Press the Apply button.

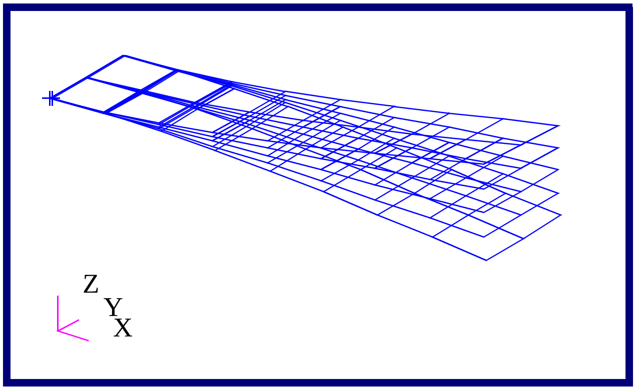


Figure 3-2 Modal Animation of Cantilever Plate.

Put a Fringe on a Deformed Plot

1. Create a deformation plot as explained in this chapter and make sure it is posted to the current viewport.
2. Set the Object to Fringe.
3. Create a Fringe plot in the same viewport as the Deformation plot but change the Display Attributes before pressing Apply. See **Fringe Plots** (p. 73) for an explanation of fringe plot creation.

4. In the Display Attributes for creating Fringe plots, make sure the Show on Deformed toggle is turned ON.

5. Press the Apply button for the fringe plot with the Animate toggle OFF.

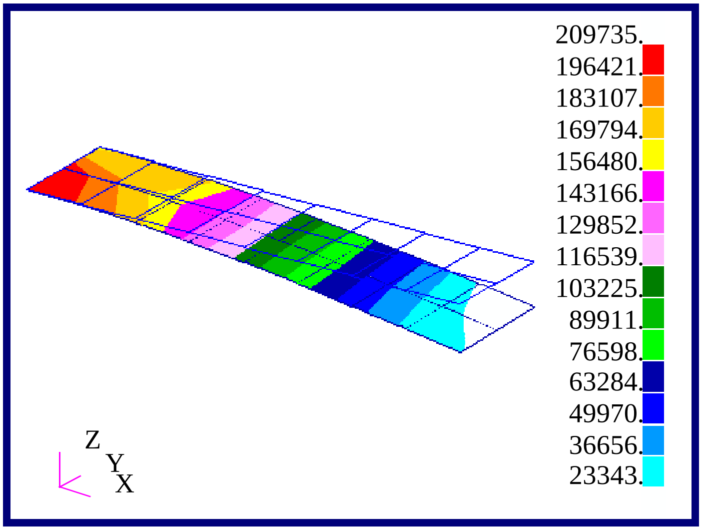
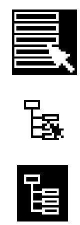


Figure 3-3 Fringe Plot on a Deformation Plot of Cantilever Plate with Undeformed Shape.

Display a Transient Animation of a Deformed Shape

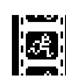
1. From the Select Results form (left most icon) select the Result Cases (time steps) from the first listbox that you wish to include in the transient animation. You must select more than one. Use the mouse and the control key to select discontinuous selections or the shift key to select a continuous selection. You can also use the Select button when the Result Cases are being displayed in abbreviated form to filter and select the Result Cases (time steps) you want. In abbreviated form the Result Case name will only appear in the listbox as the name with the number of subcases (time steps) selected, (i.e., Load Case 1, 6 of 41 subcases).



2. Select the Deformation Result from the next listbox.

3. Turn the Animate toggle ON. (If the Animate toggle is OFF then the resulting plot will be the maximum of all selected Result Cases.)

4. Press the Animation Options icon button and then select a global variable (time) to animate by. Make any other optional modifications you want.

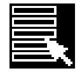


5. Press the Apply button. This method also works with load steps, frequency steps, or simply with multiple load cases that you may wish to animate.

Mask Out Deformation Components

1. Follow the instructions for **Create a Simple Static Deformation Plot** (p. 64) above.

2. Go to the Select Results mode of the form.



3. At the bottom of the Select Results form change the Show As pull down menu to Components.

4. Select or de-select the components that you wish to be used in creating the deformation plot.

5. Press the Apply button.

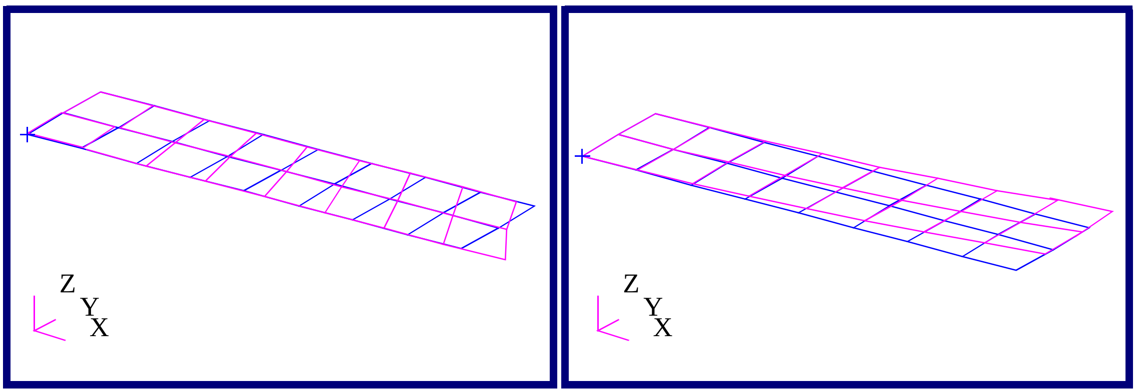


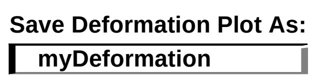
Figure 3-4 X and Y Component Only Deformation Plots of Cantilever Beam.

Save a Deformed Plot

1. Set up the plot for a deformation as explained in **Create a Simple Static Deformation Plot** (p. 64) above but don't press the Apply button.

2. Before pressing the Apply button to create a plot or an animation, press the Options icon button.

3. Type a name in the Save Deformation Plot As databox.



4. Then press the Apply button. The plot is now saved under a specific name which can be recalled (posted / unposted) graphically when desired.

Display Multiple Deformations in the Same Viewport

1. Set up the first deformation plot as explained in the examples above but don't press the Apply button.

2. (Optional) You will most likely have to select target entities if each plot will consist of the same deformation results.

3. Save the deformed plot as explained in **Save a Deformed Plot** (p. 67).

4. Repeat this process for as many deformation plots necessary in the same viewport. You will most likely want to change plot attributes from plot to plot also so as to be able to see the different deformations, otherwise they will all plot on top of each other.

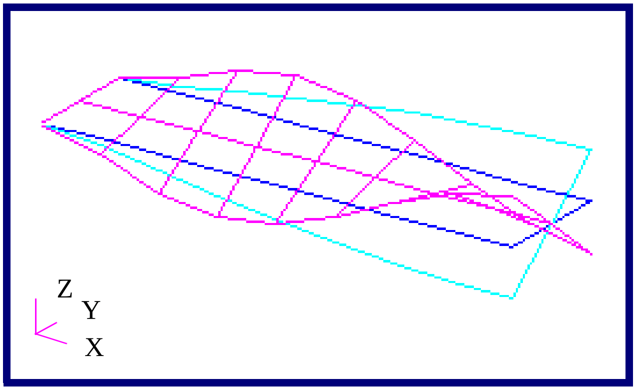


Figure 3-5 Two Deformation Plots in the Same Viewport, First Torsional Mode in Wireframe and Second Torsional Mode in Free Edge Display with Undeformed Shape.

Display Multiple Deformations in Separate Viewports

1. Set up the first deformation plot as explained in the examples above but don't press the Apply button.

2. (Optional) Save the deformed plot as explained in **Save a Deformed Plot** (p. 67). If you don't specifically save the plot then a default name will be given for each viewport such as `default_Deformation2`, `default_Deformation3`, etc.

3. Create a new viewport (from the Viewport menu on main MSC.Patran form) and make it the active viewport. This is done by placing the cursor at the edge of the viewport and clicking with the mouse. The current viewport will have a red border around it. This is the area where you click the mouse.

4. Repeat this process for as many deformation plots as necessary but change the current viewport each time.

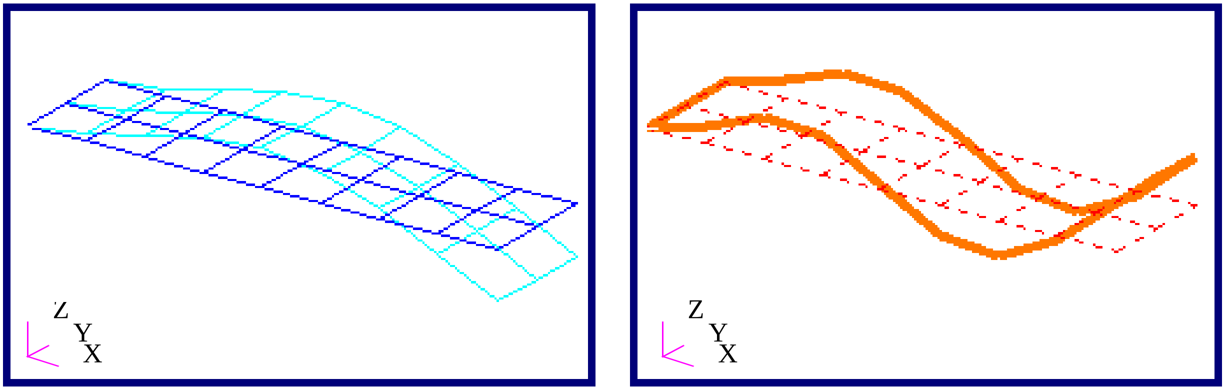
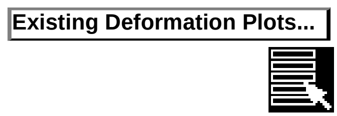


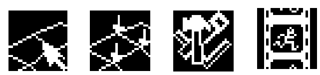
Figure 3-6 Two Deformation Plots in Separate Viewports, 2nd Bending Model in Wireframe and 3rd Bending in Free Edge Display.

Modify a Deformation Plot or Animation

1. Set the Action to Modify with the Object set to Deformation.
2. Select an existing deformation plot using the Existing Deformation Plots button.



3. Change results, target entities, display attributes, plot or animation options as required.



4. Press the Apply button at any time to see the results of your modifications.

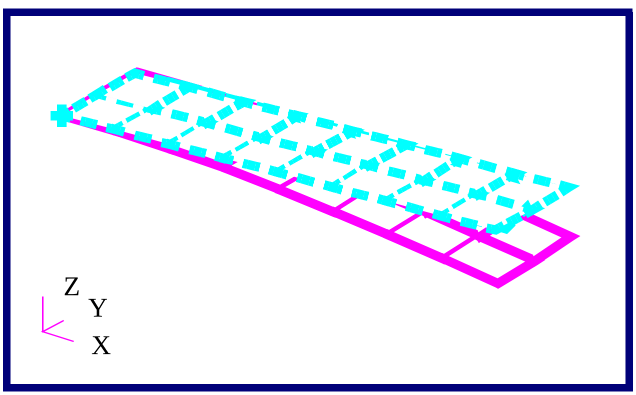
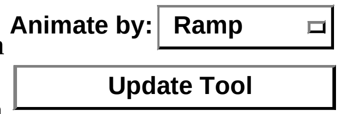


Figure 3-7 Existing Deformation Plot Modified to Display Thicker Lines with Different Undeformed Color, Line and Render Styles.

Display Multiple Deformation Animations in the Current Viewport

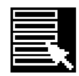
1. First create and save the deformation plots (transient or static) as explained in **Display Multiple Deformations in the Same Viewport** (p. 67) above but do not animate them. Do not turn ON the Animate toggle.
2. (Optional) Set the Action to Post and the Object to Plots. Post the plots that you want to animate to the current viewport if they are not already posted.
3. Set the Action to Create and the Object to Animation. One by one, select the posted plots that you wish to animate from the top listbox and modify their animation method if necessary. This is done by pressing the Update Tool button.



4. Press the Apply button. See **Animation** (p. 153) for more details on this procedure.

Component Suppression with Coordinate Transformation

1. Create a deformation plot as explained in **Mask Out Deformation Components** (p. 67).

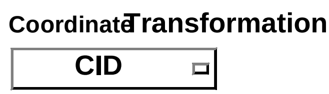


2. On the Select Results form for Deformation plots, set the Show As pulldown to Component.

3. Turn OFF the components that you wish to suppress.

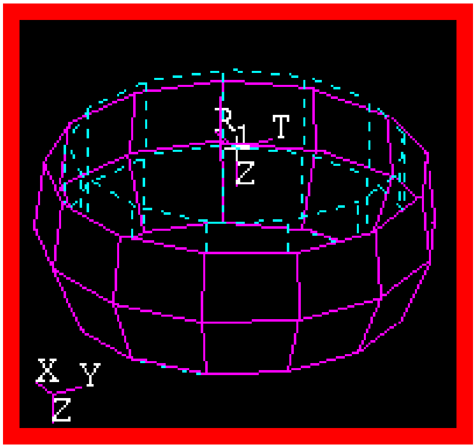
4. Go to the Plot Options form.

5. Set the Coordinate Transformation to CID and graphically select the coordinate system of interest.

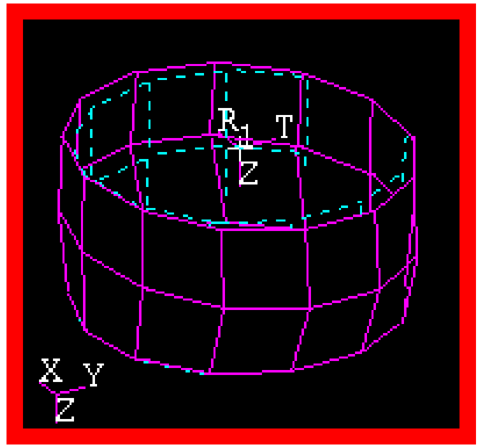


6. Press the Apply button. Only the non-suppressed components of the deformation plot will be displayed.

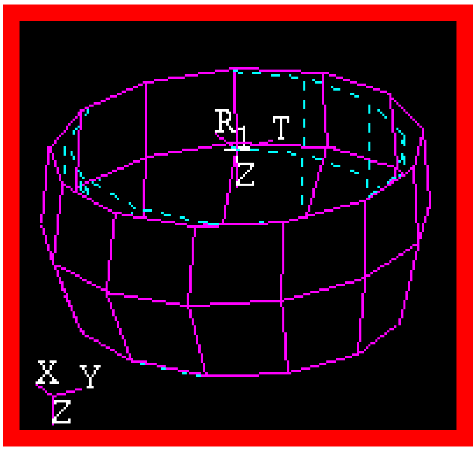
   In the example below (**Figure 3-8**), a deformation plot is posted of a cylindrical component. The coordinate components of the global rectangular system are suppressed in adjacent plots. Finally a coordinate system transformation is performed to a cylindrical system with coordinate suppression.



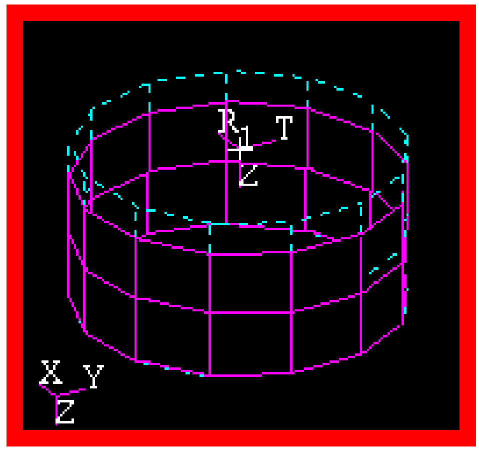
Total Deformation (all components)



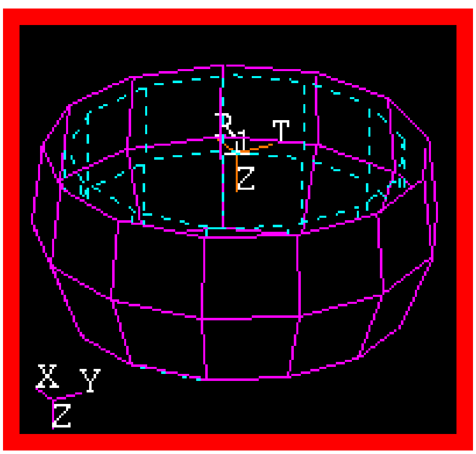
X-Deformation (global rectangular)



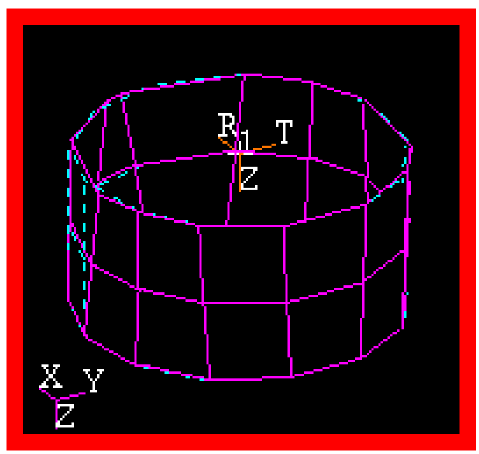
Y-Deformation (global rectangular)



Z-Deformation (rectangular & cylindrical)



R-Deformation (cylindrical)



θ-Deformation (cylindrical)

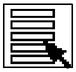
Figure 3-8 Coordinate Suppression of Deformation Plots with Coordinate Transformation into Cylindrical System.

CHAPTER 4

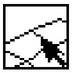
Fringe Plots

4.1 Overview

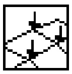
For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). To specifically make or modify a fringe plot select Create or Modify from the Action pull-down menu on the Results application form; and select Fringe from the Object pull-down menu.



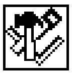
Selecting Results (p. 16)



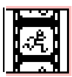
Target Entities (p. 76).



Display Attributes (p. 78).



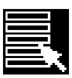
Plot Options (p. 80).



Animation Options (p. 156).

A fringe plot is like a contour plot where wide color bands each representing a range of results values are plotted onto the finite element model. They differ from contour plots in that no control over the width of the color band is allowed. The color bands cover the entire finite element model.

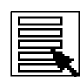
There is only a slight difference between Create and Modify. The main difference is that Create must be used to make a new fringe plot and Modify is used to change an existing one. If you try to modify an existing plot with Create you will be asked for overwrite permission whereas Modify assumes that the action is desired, so no overwrite permission is requested.



Toggles the form to select results for fringe plots. This is the default mode of the Fringe form.

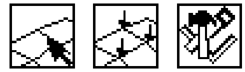
For both Modify and Create the same basic operations and options are available. To create or modify a fringe plot the following basic steps must be followed:

1. Set the Action to Create or Modify and the Object to Fringe.
2. Select a Result Case or Cases from the Select Results Case(s) listbox. See **Selecting Results** (p. 16) for a detailed explanation of this process as well as **Filtering Results** (p. 19).

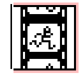


3. Select a fringe result from the Select Fringe Result listbox.
4. If more than one layer is associated with the results, select the layer (using the Position button) you wish to plot. These can be top or bottom results of shell elements, beam locations or laminate layers.
5. Optionally change the result Quantity. This is only possible if the selected result allows for this. If a tensor or vector result has been selected, it must be resolved to a scalar value. The various resolutions are:

6. Optionally select the target entities, change display attributes, or invoke other plot options by changing these settings using the three middle button icons at the top of the form. These are described in detail later in this chapter.



7. If animations are desired, turn the Animate button ON in the main form where results are selected and optionally change animation attributes with the right most icon at the top of the screen. For detailed explanations of animation options see **Animation Options** (p. 156) and **Animation Control** (p. 158)



8. Press the Apply button when ready to create the fringe plot.

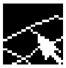
To modify an existing fringe plot, simply follow the above procedure with the Action set to Modify. However, you must first select an existing plot using the Existing Fringe Plots button on the main form where results are selected. When an existing fringe plot is selected, all results, attributes, and options in the various widgets associated with that plot are updated to reflect that plot's settings. You may then proceed to modify the plot.

By default a fringe plot with the name `default_Fringe` will be created unless the user specifically gives a different name. Multiple fringe plots can only be created and posted by giving separate names. Multiple fringe plots can be posted to the same viewport or to separate viewports. Each viewport can have its own set of fringe plots or other plot types posted. This is also true for animation of these plots. Only animation in the same viewport will be synchronized.

Each plot can have its own attributes. Each plot can also target or be displayed on separate entities and have its own associated options. These are detailed in the next sections.

4.2 Target Entities

Fringe plots can be displayed on various model entities. By default fringe plots are displayed on all free faces of everything displayed in the current viewport. To change target entity selection for fringe plots, press the Target Entities selection button with the Object set to Fringe.



Toggles the form to select target display entities for fringe plots.

The following table describes in detail to which entities fringe plots can be targeted. Fringe plots can only be plotted onto elements or surfaces of elements.

Entity	Description
Current Viewport	By default fringe plots are displayed on all finite element entities displayed in the currently active viewport. The only exception to this is when layered results exist and are associated to only certain element types. Then only those elements will display the fringe plot.
Elements	Individual elements may be selected on which to display the fringe plot. You may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.). To select all elements use the syntax “Elem 1:#.”
Groups	Fringe plots can be limited to only selected groups. A selected group or groups must have elements in them otherwise the plot will not appear. A listbox allows selection of the group(s) to which the fringe plot will be applied. This is handy in that the same finite element entities can belong to multiple groups. Only those groups selected will be displayed with a fringe plot while all other non-selected groups will remain unaffected and retain their own display attributes.
Materials	Fringe plots can be targeted at only those finite elements which have certain material properties assigned to them. A listbox appears allowing selection of the materials for whose elements will be targeted for a fringe display.
Properties	Fringe plots can be targeted at only those finite elements which have certain element properties assigned to them. A listbox appears allowing selection of the properties for whose elements will be targeted for a fringe display.
Element Types	Fringe plots can be limited to only certain element types also.

In addition to targeting the above entities for a fringe plot, the fringe plot can be isolated to attributes of the entities as described in the following table:

Attribute	Description
Free Faces	By default all free faces of the target entities will display the fringe plot.
Faces	The fringe display will be plotted on all faces of every element for the target entities.
Free Edges	This will display a fringe plot on the edges that only have one common element. This results in a 1D line or edge type fringe as opposed to a 2D surface display and will generally give you the outline of your model.
Edges	This will display a fringe plot on all the edges of every element. This results in a 1D wireframe fringe type plot as opposed to a 2D surface display.
Target Deformations	By default, fringe plots that are to be displayed on the deformed shape will be displayed on all deformation plots posted. You can select which deformation plots to target the fringe plot to by selecting deformation plots form this listbox. This listbox will only appear when more than one deformation plot exists and is posted.

Important: Once a target entity has been selected, it will remain the target entity for the fringe plot until the user physically changes it.

4.3 Display Attributes

Fringe plots can be displayed in various forms. Display attributes for fringe plots are accessible by pressing the Display Attributes selection button on the Results application form with the Object set to Fringe.

Toggles the form to change display attributes for fringe plots.

The following table describes in detail the fringe display attributes which can be modified:

Attribute	Description
Show Spectrum	This toggle will remove or display the spectrum bar on the right of the current viewport. If the spectrum does not disappear from the graphics screen when the toggle is OFF, make sure other plots also do not have the spectrum turned ON.
Spectrum	This button will bring up the Spectrums form from which you can set the spectrum for the current viewport or create new spectrums. This form operates independently of the Results application. Therefore you do not need to press the Apply button in the Results application to effect changes in the spectrum. To learn more about changing the spectrum see **Display>Spectrums** (p. 306) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.
Range	This button will bring up a subordinate form for reassigning the range to the current spectrum in the current viewport. See **Spectrum/Range Control** (p. 29) for details. Also from this form you can invoke the Ranges form for creating and modifying new ranges. To learn more about changing ranges see **Display>Ranges** (p. 307) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.
Style	Four fringe styles are available. The default is a Discrete/Smooth plot which will give distinct color bands which begin and end sharply. Continuous style will blend the colors together giving a smoother transition between color bands but no distinctive beginning or end to any particular color band. The Element Fill style will simply color code each element. Discrete/Flat is similar to Discrete/Smooth but may give a better or worse image depending on the graphics device.
Shading	On true color machines, fringe plots may be shaded. On machines that do not support true color, a shaded fringe plot will not look good at all. Best results are with Hardware Rendering on true color machines.
Element Shrink Factor	You can display a fringe plot with the elements shrunk a certain percentage. It is best to experiment with this.
Fringe Edges	The edge color display of the fringe plot can be changed as desired.

Attribute	Description
Display (Edges)	The edge display is simply the display of the finite element mesh or the display of the free edges of the model. You can choose to display either of these or none at all.
Style (Edges)	The styles of the edge lines plotted on the target entities can be set to a solid line, or variations of dotted lines.
Width (Edges)	The thickness of the edge lines on the target entities is set with this attribute setting.
Title Editor	Selecting this button opens a form that allows the fringe plot title to be edited. See **Results Title Editor** (p. 31).
Show Title	If this toggle is ON, then a title for the fringe plot is displayed. Otherwise no title is displayed.
Lock Title	If this toggle is ON, then the title for the fringe plot is not modified by results form selections. Otherwise some results form selections modify the title.
Show Max/Min Label	If this toggle is ON, then the maximum and minimum values of the selected results set are displayed in the viewport. Otherwise they are not displayed.
Show Fringe Label	If this toggle is ON, then the fringe label will be displayed.
Label Style	You can change the style of the numbers displayed on the spectrum bar and labels from the subordinate form that appears when you press this button. Styles can be integer, fixed, or exponential with specification of the number of significant digits.
Show on Deformed	The fringe plot will be displayed on the deformed shape of the model if a deformation plot has also been posted the current viewport. The fringe plot will display on all deformation plots if more than one is posted to the viewport by default. If this is not desired, see **Target Entities** (p. 76).

Important: Once display attributes have been selected, they will remain in effect for the fringe plot until the user physically changes them.

4.4 Plot Options

Fringe plots have various options. Plot options for fringe plots are accessible by pressing the Plot Options selection button.

Toggles the form to select plot options for fringe plots.

The following table describes in detail the fringe plot options which can be modified:

Option	Description
Coordinate Transformation	Vector and tensor results to display fringe plots can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran global system, a material coordinate system, element IJK coordinate system or the nodal (analysis) coordinate system depending on the type of result (vector, or tensor). See **Coordinate Systems** (p. 305) for a definition of each of these coordinate systems. The default is no transformation, which will plot data in the coordinate frame as stored in the database. Typically the solver code will calculate results at nodes in the analysis coordinate system specified by the user. These can vary from node to node. Element data can be stored from the analysis code in any coordinate system. Note also that the analysis translators that import the results data into the database can transform results. Check with the appropriate translator guide.
Scale Factor	This scale factor has the effect of simply scaling the results up or down by the specified amount. This will have an effect on the fringe plot but will not effect the spectrum range values.
Filter Values	By specifying a filter value, a gate will be used to keep values below a maximum, above a minimum, between a certain range, or at the exclusion of certain values. The default is none. If filtering is used, only those elements which pass the filter gate will display a fringe plot on them. In other words, the fringe plot will not be displayed on elements that don't pass the filter test completely.
Averaging Domain	For element based result quantities that must be displayed at nodes, an averaging domain must be used since more than one result will exist for each node. There is a contribution from each element attached to any particular node. By default all entities which contribute are used. Alternatively, you can tell the Results application to only average results from those elements that share the same material or element property, are from the same target entities, or have the same element type. For more details, see **Averaging** (p. 292).

Option	Description
Averaging Method	The method in which certain results are determined can make a difference to the actual displayed plot. This is important when derived results from element based tensor or vector results are used such as von Mises stress or Magnitude displacements, For instance, if you average at the nodes first and then derive the desired quantity, you may get a different answer than if you derive first and then average. It is left up to the user to decide which is correct. For more detail see **Averaging** (p. 292).
Extrapolation Method	Many times, element based results that are to be displayed at nodes exist at locations other than the nodes such as at integration points. Various methods are available to the user to extrapolate these results out to the nodes. For more details, see **Extrapolation** (p. 299).
Complex No. as	The Real component of a complex number is the default by which results will be postprocessed. To force the postprocessor to use a different quantity such as Magnitude, Imaginary, Phase, or Angle, set this option pull down menu. This option will only be available if a complex result has been selected. It is not recommended to calculate invariants (e.g., von Mises) from complex results because the phase is not accounted for.
Use PCL Expression	The results can be modified with a user defined PCL expression.
Define PCL Expression	For more details, see **PCL Expressions** (p. 195).
Existing Fringe Plots	This listbox displays all existing fringe plots. You may select one of these plots from the listbox and all settings of that plot including display attributes, target entities, option, and selected results will be restored. This is an easy mechanism to help make many plots with the settings of an existing plot without modifying the selected plot. When the Action is set to Modify, this listbox appears under the Select Results display of the Results application form also.
Save Fringe Plot As	Fringe plots can be saved by name and recalled later for graphical display. Multiple fringe plots can be saved in the database and displayed simultaneously. Be aware that when multiple fringe plots are displayed simultaneously there could be some display problems if they are displayed on the same entities. These fringe plots can be posted/unposted and deleted as explained in **Post/Unpost** (p. 25) and **Delete** (p. 27) respectively. Once a plot has been created and named it retains all results, attributes, target entities, and options assigned to it. If no plot name is specified a default is created called `default_Fringe`. As long as no plot name is specified, the `default_Fringe` will be overwritten each time a plot is created or modified.

Important: Once plot options have been selected, they will remain in effect for the fringe plot until the user physically changes them.

4.5 Examples of Usage

The following are some typical scenarios for usage of the Fringe plot tool. These instructions assume that the Action is set to Create and the Object is set to Fringe unless otherwise specified.

Create a Simple Static Fringe Plot

1. From the Select Results form (left most icon) select the Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.

2. Select the Fringe Result from the next listbox.
3. (Optional) If the selected result has more than one layer associated with it, select the layer using the layer Position button.

4. (Optional) If the result is a vector or tensor, select a resolved scalar results value from the results Quantity pull-down menu (such as von Mises stress or Magnitude displacement).

5. Press the Apply button with the Animate toggle OFF.

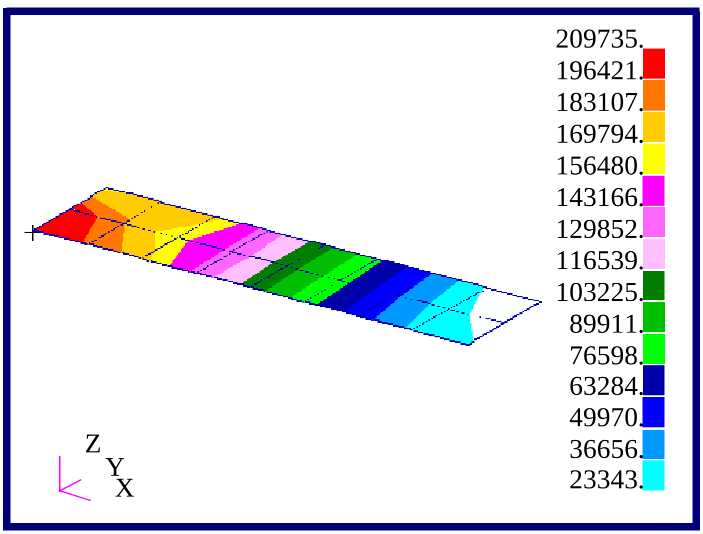


Figure 4-1 Fringe Plot of von Mises Stress on Cantilever Plate.

Animate a Static Fringe Plot

1. From the Select Results form (left most icon) select the Result Case from the first listbox.

2. Select the Fringe Result from the next listbox.

3. (Optional) If the selected result has more than one layer associated with it, select the layer using the result Position button.

Position...(at Z1)

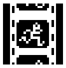
4. (Optional) If the result is a vector or tensor, select a resolved scalar results value from the results Quantity pull-down menu (such as von Mises stress or Magnitude displacement).

Quantity: von Mises

5. Turn the Animate toggle ON.

Animate

6. Press the Animation Options icon button.



7. Change the Animate By option pulldown menu to Ramp.

Animate by: Ramp

8. Press the Apply button.

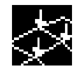
Apply

Put a Fringe on a Deformed Plot

1. Create a deformation plot and make sure it is posted to the current viewport. See **Deformation Plots** (p. 55) for an explanation of deformation plot creation.

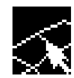
2. Now make a fringe plot as explained in **Create a Simple Static Fringe Plot** (p. 82), but do not press the Apply button.

3. Press the Display Attributes icon button.

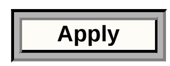


4. Turn ON the Show on Deformed toggle.

5. (Optional) If more than one deformation plot exists and is posted to the viewport, you can optionally specify on which deformation plots to display the fringe plot. This is done under the Target Entities option. Otherwise the Fringe plot will show on all Deformation plots posted.



6. Press the Apply button with the Animate toggle OFF.



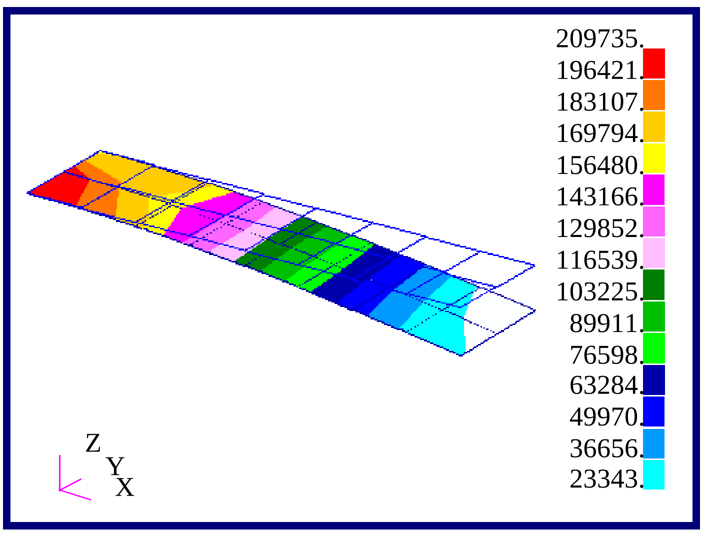
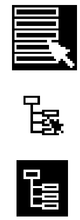


Figure 4-2 Fringe Plot of von Mises Stress on Deformed Cantilever Plate.

Display a Transient Fringe Animation

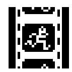


1. From the Select Results form (left most icon) select the Result Cases (time steps) from the first listbox that you wish to include in the transient animation. You must select more than one. Use the mouse and the control key to select discontinuous selections or the shift key to select continuous selection. You can also use the Select button when the Result Cases are being displayed in abbreviated form to filter and select the subcases (time steps) you want. In abbreviated form the Result Case name will only appear in the listbox as the name with number of subcases (time steps) selected, (i.e., Load Case 1, 6 of 41 subcases).
2. Select the Fringe Result from the next listbox.
3. (Optional) If the selected result has more than one layer associated with it, select the layer from the result Position pull-down menu.

4. (Optional) If the result is a vector or tensor, select a resolved scalar results value from the results Quantity pulldown menu (such as von Mises stress or Magnitude displacement).

5. Turn the Animate toggle ON. (If the Animate toggle is OFF then the resulting plot will simply plot up a maximum plot of all selected Result Cases and will not animate.)

6. Press the Animation Options icon button.



7. Select a Global Variable (time) to Animate By. Make any other optional modifications you want.

8. Press the Apply button. This method also works with load steps, frequency steps, or simply with multiple load cases that you may wish to animate.

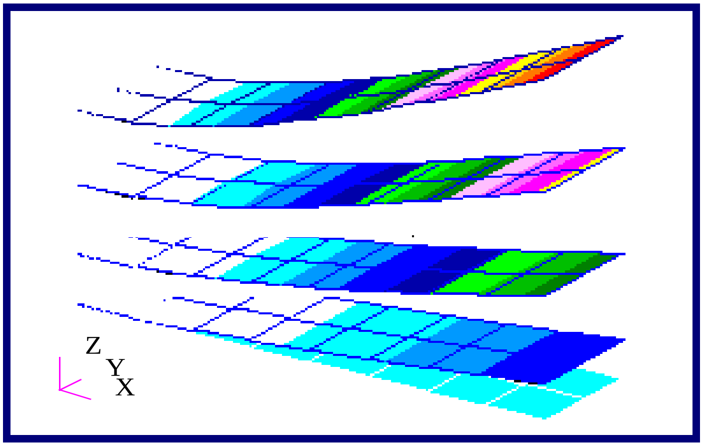


Figure 4-3 Exaggerated Transient Animation of Displacements with Fringe Superimposed on Deformed Shape.

Save a Fringe Plot

1. Set up the fringe plot as explained in **Create a Simple Static Fringe Plot** (p. 82) but do not press the Apply button.
2. Before pressing the Apply button to create a plot or animation, press the Plot Options icon button.

3. Type a name in the Save Fringe Plot As databox.

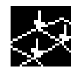
Save Fringe Plot As:
myFringe

4. Then press the Apply button. The plot is now saved under a specific name which can be recalled (posted / unposted) graphically when desired.

Apply

Display Multiple Fringe Plots in the Current Viewport

1. Set up the first fringe plot as explained in the previous examples, but do not press the Apply button.
2. (Optional) You will most likely have to select target entities so as not to overlap fringe plots.



3. Save the fringe plot as explained in **Save a Fringe Plot** (p. 86).
4. Repeat this process for as many fringe plots as necessary.

   To put up multiple fringe plots in different viewports, simply make the viewport of interest active by clicking the mouse in its border and following the same procedure. An active viewport has a red border around the graphics.

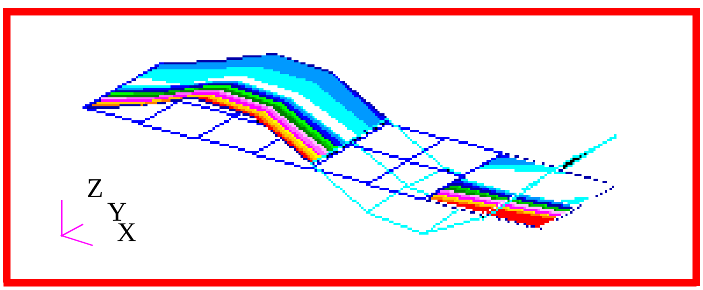
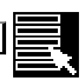


Figure 4-4 Two Fringe Plots Targeting Certain Elements, One on the Undeformed Shape and One on the Deformed Shape.

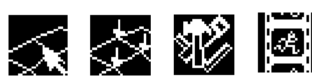
Modify a Fringe Plot or Animation

1. Set the Action to Modify with the Object set to Fringe.
2. Select an existing fringe plot using the Existing Fringe Plots button.

Existing Fringe Plots...



3. Change results, target entities, display attributes, plot or animation options as required.

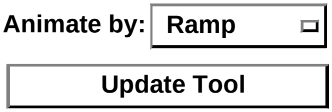


4. Press the Apply button at any time to see the results of your modifications.

Apply

Display Multiple Animations in the Current Viewport

1. First create and save the fringe plots (transient or static) as explained in **Display Multiple Fringe Plots in the Current Viewport** (p. 86), but do not animate them. Do not turn ON the Animate toggle.
2. (Optional) Set the Action to Post and the Object to Plots. Post the plots to the viewports that you want if they are not already posted. They can be any plot type that supports animation.
3. Set the Action to Create and the Object to Animation. One by one, select the posted plots that you wish to animate from the top listbox and modify their animation method if necessary. This is done by pressing the Update Tool button.



4. Press the Apply button. See **Animation** (p. 153) for more details on this method.

CHAPTER

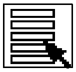
5

Contour Line Plots

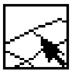
- Overview
- Target Entities
- Display Attributes
- Plot Options
- Contour Plot Example

5.1 Overview

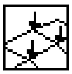
For an overview of how the Results Application works, please see **Introduction to Results Postprocessing** (p. 1). To specifically make or modify a contour line plot, select Create or Modify from the Action pull-down menu on the Results application form and select Contour from the Object pull-down menu.



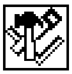
Selecting Results (p. 16)



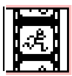
Target Entities (p. 92).



Display Attributes (p. 93).



Plot Options (p. 95).



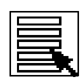
Animation Options (p. 156).

The lines rendered in a contour line plot show the locations within the model where a specific result value exists. Contour line plots can be rendered with and without labels. Characters (e.g. a, b, c, etc.) are used for the contour line labels. When labeling is turned off, you must use the contour line color to determine the numerical value each line represents by matching the color with the viewport's spectrum range. Since there can be only one spectrum for all plots shown within a Patran viewport, you will see that the contour line values are indicated with characters placed at the mid-location of the spectrum's numerical intervals. The contour line value is the average of its interval's maximum and minimum range values. If a Fringe plot is also shown in the viewport, its color bands relate to the range of values represented by each color band.

Only the two options, Create and Modify, exist for the Action method of the tool. The functional difference between these is as follows. Create must be used to derive a new contour plot whereas Modify is used to change an attribute of an existing contour plot. If you choose to Modify an existing plot, you can identify the existing plot by pressing the Existing Contour Plot button on the top of the form.

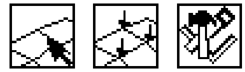
After you set the Action to either Create or Modify you will need to perform the following steps to completely define your contour plot.:

1. Select a Result Case or Cases from the Select Results Case(s) listbox. See **Selecting Results** (p. 16) and **Filtering Results** (p. 19). for a detailed description of this step.

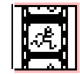


2. Next, select a contour result type from the Select Contour Result listbox.
3. If the result is associated to positions within your simulation model (e.g. layer, shell, or beam section positions), you should select the Position button and specify the position for the result type.
4. A result Quantity can be optionally selected. This will cause Patran to calculate this quantity from the result type you have selected in step 2 above.

5. After performing steps 1-4 you have now defined the numerical value the contour plot will represent. To specify which portion of the model plot will be rendered (i.e. plot target), its graphical display attributes, or to further specify various numerical operations that you would like to apply to the result data you will need to select the second through fourth icons respectively.



6. If a contour animation is desired, select the Animate button. For a detailed description of the animation options, see **Animation Options** (p. 156) and **Animation Control** (p. 158)



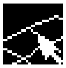
7. Press the Apply button when you are ready to create the contour plot.

When creating a contour plot you can specify its name so you can define multiple contour plots with varying definitions. The plot name is entered within the Plot Options sub form (i.e. the fourth icon described in step 5). If you do not specify a name the default name, `default_contour`, will be used. Multiple contour plots can only be created and posted by defining separate names. Multiple contour plots can be posted to the same viewport or to separate viewports. Each viewport can have multiple plot types posted within it.

The following sections discuss in detail the various options that are found in the plot sub forms that correspond to the steps shown above.

5.2 Target Entities

Contour plots can be displayed on various model entities. By default, Contour plots are displayed on all free faces of all elements displayed in the current viewport. To change the target entity selection for your Contour plot, press the Target Entities selection button with the Object set to Contour.



Toggles the form to select target display entities for Contour plots.

The following table describes in detail to which entities Contour plots can be targeted upon. Contour plots can only be rendered on element entities.

Entity	Description
Current Viewport	By default, Contour plots are displayed on all finite element entities displayed within the current viewport.
Elements	Individual elements may be selected for the Contour plot. You may enter the element ids manually or select them graphically. Be sure to include the word Elem in front of the ids you enter (e.g. Elem 1, 5, 55, 100:102 etc.). To select all elements use the following syntax. “Elem 1:#.”
Groups	Contour plots can be rendered on selected groups. The selected group(s) must contain elements.
Materials	Contour plots can be rendered upon finite elements which have a specific associated material type.
Properties	Contour plots can be rendered upon finite elements which have a specific associated element property set.
Element Types	Contour plots can be rendered upon specific element types (e.g. Quad4, Hex8, etc.).

5.3 Display Attributes

The graphical elements of a Contour plot can be modified by changing its various display attributes. Display attributes for Contour plots are accessed by pressing the Display Attributes selection icon.

Toggles the form to show the available display attributes for the Contour plot.

The following table describes the Contour display attributes which can be modified:

Attribute	Description
Spectrum / Constant	The radio buttons, Spectrum and Constant, allow you display contour lines with either a spectrum of colors relative to their numerical value or as a single color respectively.
Show Spectrum	This toggle will remove or display the spectrum bar within the current viewport. If you set the toggle off and spectrum is still displayed, check if you have other plots posted in the viewport. If there are other plots, you will need to set this toggle off for all plots posted in the viewport.
Show Viewport Legend	This toggle will remove or display the legend that is shown in the lower right hand corner of the viewport. This legend displays the plot name and which entity contains the maximum and minimum result values. If you set the toggle off and the legend is still displayed, check if you have other plots posted in the viewport. If there are other plots, you will need to set this toggle off for all plots posted in the viewport.
Spectrum	This button will activate the Spectrum form. This form allows you to select an existing spectrum to be associated to the current viewport. It also allows you to define a new spectrum. Since this form is independent of the Results application, you will not need to press the Apply button on the Results application to effect changes to the spectrum. To learn more about changing the spectrum see **Overview** (p. 90) of the *MSC.Patran Reference Manual, Part 6: Results Postprocessing.*
Range	This button will activate the Range sub form. This form allows you to assign an existing range to the viewport's spectrum. See **Spectrum/Range Control** (p. 29) for details. The form also allows you to modify an existing range or create a new range. To learn more about ranges see **Overview** (p. 90) of the *MSC.Patran Reference Manual, Part 6: Results Postprocessing.*
Element Edges	This button allows you to change the color that the element edges are rendered with.
Display (Element Edges)	This pop down button allows you to select which element edges are rendered. The selection includes Free Edges and Element Edges which cause either the elements free edges or all element edges to be rendered respectively.

Attribute	Description
Style (Element Edges)	The Style pop down button allows you to control the line type that is used to render the element edges. The selections are solid, dashed, dotted, and dot-dash line styles.
Width (Element Edges)	This pop down button allows you to control the thickness of the rendered element edges.
Contour Style	The Contour Style pop down button allows you to control the line type that is used to render the contour lines. The selections are solid, dashed, dotted, and dot-dash line styles.
Contour Width	This pop down button allows you to control thickness of the contour lines.
Title Editor	Selecting this button opens a form that allows the contour plot title to be edited. See **Results Title Editor** (p. 31).
Show Title	If this toggle is ON, then a title for the contour plot is displayed. Otherwise no title is displayed.
Lock Title	If this toggle is ON, then the title for the contour plot is not modified by results form selections. Otherwise some results form selections modify the title.
Show Max/Min Label	If this toggle is ON, the maximum and minimum value of the selected result type will be displayed. Their value labels will be rendered in a color different than all other value labels.
Label Frequency	As you adjust the Label Frequency slider from 0 to Max the spacing between the contour line labels will decrease therefore increasing the label density. To turn off contour labeling adjust the slider to zero. Zero label frequency is the default frequency value.
Label Style	Pressing the Label Style button causes the Contour Label Style form to appear. The form allows you to modify the label color, format (e.g. fixed, exponential, and integer), and significant figures of the plot's range and maximum / minimum labels.
Show on Deformed	If this toggle is set ON, the Contour plot will be targeted on the model's posted deformed shape. By default, the Contour plot will display on all deformation plots posted in viewport. If this is not desired, see **Target Entities** (p. 92) for further options.

5.4 Plot Options

The Plot Options form includes numerical operations that can be applied to the result type you chose previously in the Select Results form. The options that appear on this form are relative to the selected result type's data type (i.e. scalar, vector, or tensor). The following table describes the various operations that can be applied.

Toggles the form to select numerical plot options for Contour plots.

Option	Description
Coordinate Transformation	Vector and tensor results can be transformed into any of the following coordinate systems. The AsIs coordinate system, any user defined local system (CID), the projection of any CID, the MSC.Patran global system, material, element IJK, or the default coordinate system. See **Coordinate Systems** (p. 305) for the description of each coordinate system type. AsIs is the default option.
Scale Factor	The scale factor is a numerical multiplier that is applied to the data after all other numerical operations have transformed the raw analysis data to its final form.
Filter Values	By specifying a filter value, you can create a gate that will be used to include values below a maximum, above a minimum, between a certain range, or exclude a range of values. The default option is none. If filtering is used, contour lines will only be rendered on elements that contain values allowed by the filter.
Averaging Domain	For element based result quantities that must be displayed at node locations an Averaging Domain must be specified. As the element's analysis result is moved to the element's node locations multiple values can occur at nodes that are shared by adjacent elements. The Averaging Domain options describe to Patran which of the multiple result values you want to resolved to a single value. For more details, see **Averaging** (p. 292).
Averaging Method	The Averaging Method describes to Patran the order of operations that will be applied to the selected results data and the operation type that will be used to resolve multiple values to a single value at the model's node locations. For example, the Derive / Average option will cause the user specified quantity derivation (e.g. Principal stress, extract the x-component of translational displacement) to occur first, before a simple average is applied to resolve multiple results that occur at the shared element's node locations. For more details see **Averaging** (p. 292).
Extrapolation Method	Many times, element based results must be moved from their initial element location to another location (e.g. element integration points to element node locations). The Extrapolation Method defines which algorithm will be used for this operation. For more details, see **Extrapolation** (p. 299).

Option	Description
Complex No. as	If the result value you have chosen to post process is complex, then an option button will appear that allows you to show the results real, complex, magnitude, or phase representation.
Use PCL Expression	This toggle button will activate the Define PCL Expression button
Define PCL Expression	The sub form that will appear when you press this button will allow you to further modify the selected result data by applying a user defined PCL expression. For more details, see **PCL Expressions** (p. 195).
Existing Contour Plots	This listbox displays all existing Contour plots. If you select an existing plot from the listbox, the plot's definition will be used to define each sub formsub form's various options.
Save Contour Plot As	Contour plots can be saved by name and recalled later for editing. Saved Contour plots can be posted, unposted and deleted as shown in **Post/Unpost** (p. 25) and **Delete** (p. 27) respectively. Once a plot has been named, its definition is automatically persisted in the Patran database when you hit the Apply button to create the plot.

5.5 Contour Plot Example

The following is an example of creating a contour plot superimposed on a deformed shape plot.

Create a Contour Plot

1. After setting the Action / Object / Method to Create / Contour / Lines, select a Result Case from the Select Result Case(s) list box.

2. Next, select the Stress Tensor result type from the Select 2DContour Result list box.

3. (Optional) If the selected result has more than one layer associated with it select the layer using the layer Position button.

4. Select the Y Component from the Quantity pull down option menu.

5. Press the Apply button with the Animate toggle OFF to create the plot.

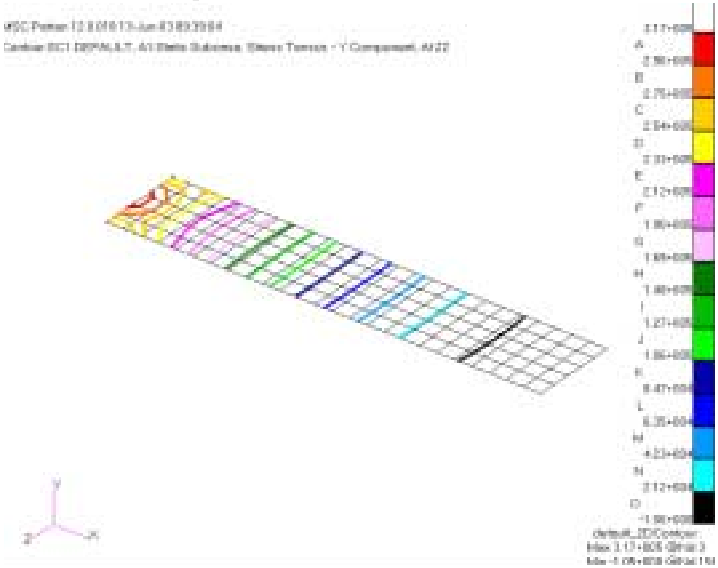
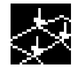


Figure 5-1 Contour Plot of the Y-Component of Stress on Cantilever Plate.

Create a Contour on a Deformed Plot

1. Create a deformation plot and make sure it is posted to the current viewport. See **Deformation Plots** (p. 55) for an explanation of deformation plot creation.
2. Now create a Contour plot as explained in **Create a Contour Plot** (p. 97), but do not press the Apply button.
3. Press the Display Attributes icon button.



4. Turn ON the Show on Deformed toggle.

Show on Deformed

5. Press the Apply button with the Animate toggle OFF.

Apply

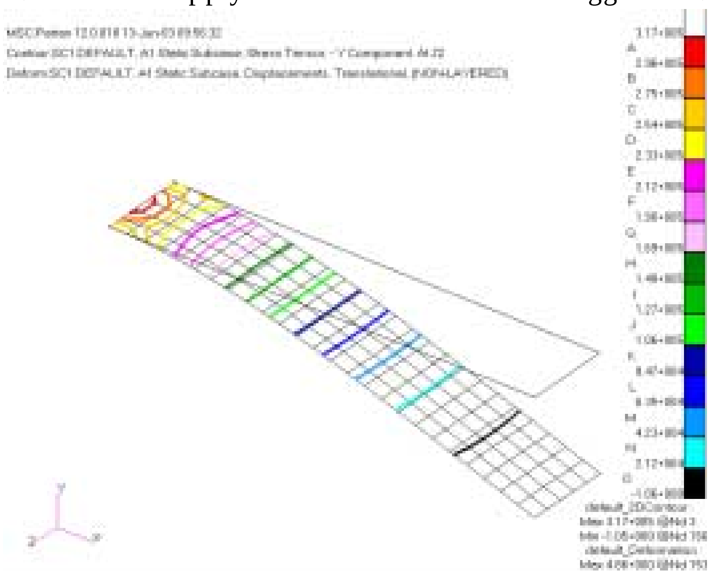


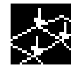
Figure 5-2 Contour Plot of the Y-Component of Stress on Deformed Cantilever Plate.

Create a Contour plot with contour labels on a Deformed Plot

1. Similar the previous example, create a deformation plot and make sure it is posted to the current viewport. See **Deformation Plots** (p. 55) for an explanation of deformation plot creation.

2. Next, create a Contour plot as explained in **Create a Contour Plot** (p. 97), but do not press the Apply button.

3. Press the Display Attributes icon button.



4. Slide the Label Frequency to the right until a frequency of 10 is achieved.

5. Turn ON the Show on Deformed toggle.

Show on Deformed

6. Press the Apply button with the Animate toggle OFF.

Apply

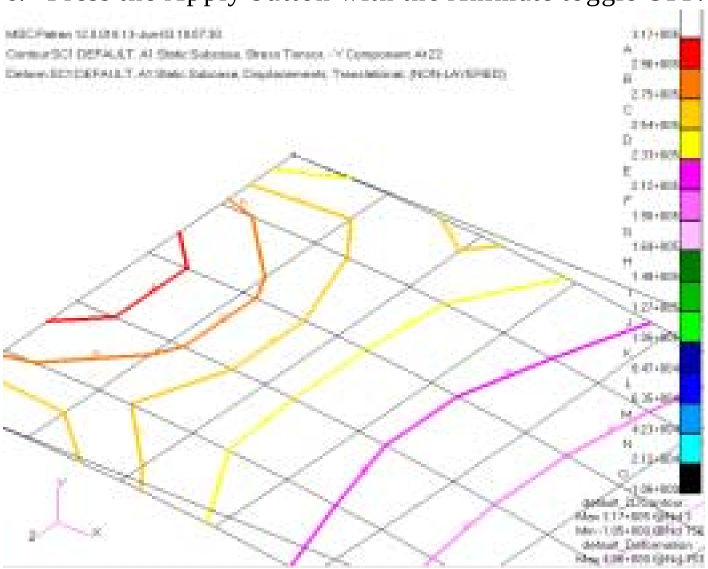


Figure 5-3 Contour Plot of the Y-Component of Stress on Deformed Cantilever Plate.

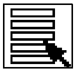
CHAPTER 6

Marker Plots

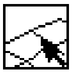
- Overview
- Target Entities
- Display Attributes
- Plot Options
- Examples of Usage

6.1 Overview

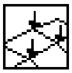
Marker plots are scalar, vector and tensor plots. For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). To specifically make or modify a marker plot select Create or Modify from the Action pull-down menu on the Results Display form; and select Marker from the Object pull-down menu. Then set the marker type to Scalar, Vector, or Tensor from the Method pull-down menu.



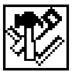
Selecting Results (p. 16)



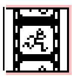
Target Entities (p. 105).



Display Attributes (p. 107).



Plot Options (p. 111).



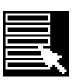
Animation Options (p. 156).

Scalar plots are scalable colored markers of various forms, e.g. triangles, squares, diamonds which represent the selected scalar result quantities displayed at nodes and elements.

Vector plots are similar to deformation plots except that instead the representation of the results is displayed as three arrow (vectors) at right angles to one another representing each coordinate contribution. The component vectors can also be resolved into a resultant vector. Any deformation plot can also be represented as a vector plot. There are of course other applications for vector plots such as viewing principal stresses.

Tensor results are similar to vector results except they have six components associated with them (the upper triangular portion of a 3x3 matrix). Typically results for tensor plots are component stress.

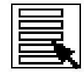
There is only a slight difference between Create and Modify. The main difference is that Create must be used to make a new marker plot and Modify is used to change an existing one. If you try to modify an existing plot with Create you will be asked for overwrite permission whereas Modify assumes that the action is desired, so no overwrite permission is requested.



Toggles the form to select results for marker plots. This is the default mode of the Marker plot form.

For both Modify and Create the same basic operations and options are available. To create or modify a marker plot the following basic steps must be followed after setting the marker type (Scalar, Vector, or Tensor):

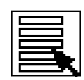
1. Set the Action to Create or Modify, the Object to Marker and the Method to Scalar, Vector, or Tensor.
2. Select a Result Case or Cases from the Select Results Case(s) listbox. See **Selecting Results** (p. 16) for a detailed explanation of this process as well as **Filtering Results** (p. 19).



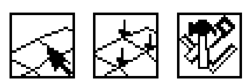
3. Select a result from the Select Scalar / Vector / Tensor Result listbox.
4. If more than one layer is associated with the results, select the layer (using the Position button) you wish to plot. These can be top or bottom results of shell elements, beam locations or laminate layers.
5. Optionally change the result Quantity. This is only possible if the selected result allows for this. If either a tensor or vector result is selected for a scalar marker plot then it must be resolved to a scalar value. If a tensor result has been selected for a vector marker plot, it must be resolved to a vector value. The various resolutions are:

Tensor to Scalar	von Mises, X, Y, Z, XY, YZ, ZX, XY Engineering, ZX Engineering, ZX Engineering, Maximum Principal, Minimum Principal, Middle Principal, Hydrostatic, 1st Invariant, 2nd Invariant, 3rd Invariant, Tresca, Maximum Shear, Octahedral, Maximum Principal 2D, Minimum Principal 2D, Tresca 2D, Maximum Shear 2D.
Vector to Scalar	Magnitude, X, Y or Z Component.
Tensor to Vector:	Minimum Principal, Middle (intermediate) Principal, Maximum Principal, Component, Minimum Principal 2D, Maximum Principal 2D.

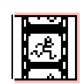
6. Optionally, for tensor or vector marker plots, change the form in which to display the tensor or vector: as components, resultants or principal values. This is done in the Show As pull-down menu at the bottom of the main Select Results form.



7. Optionally select the target entities, change display attributes, or invoke other plot options by changing these settings using the three middle icons at the top of the form. These are described in detail later in this chapter.



8. If animations are desired, turn the Animate button ON in the main form where results are selected and optionally change animation attributes with the right most icon at the top of the screen. For detailed explanations of animation options see **Animation Options** (p. 156) and **Animation Control** (p. 158)



9. Press the Apply button when ready to create the marker plot.

To modify an existing marker plot, simply follow the above procedure with the Action set to Modify. However, you must first select an existing plot using the Existing Scalar / Vector / Tensor Plots button on the main form where results are selected. When an existing marker plot is selected, all results, attributes, and options in the various widgets associated with that plot are updated to reflect that plot's settings. You may then proceed to modify the plot.

By default a marker plot with the name `default_Scalar, default_Vector or default_Tensor` will be created unless the user specifically gives a different name. Multiple marker plots can only be created and posted by giving separate names. Multiple marker plots can be posted to the same viewport or to separate viewports. Each viewport can have its own set of marker plots or other plot types posted. This is also true for animation of these plots. Only animation in the same viewport will be synchronized.

Each plot can have its own attributes. Each plot can also target or be displayed on separate entities and have its own associated options. These are detailed in the next sections.

Component selection is represented as XX, YY, and ZZ for the three component directions of any coordinate system. These translate into r, θ, z and r, ϕ, θ for cylindrical and spherical coordinate systems respectively.

Tensor Notes. By default tensors are displayed as vectors without arrow heads, each with its own color code and result label (value). Traditional tensors with a box and vectors with arrow heads can also be created by changing display attributes. For two dimensional tensors, the default plots are know as "Crow's Feet" and display two axial and one shear (coupled) component or the two principals (maximum and minimum). This display is supported for running loads, moments, stresses, and strains. The results can be rotated to any direction desired.

The tensor results for element strains are displayed at the mid-plane and outer fiber locations in two different manners.

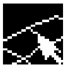
1. Three legged tensor display indicating axial and shear directions with the appropriate result quantities labeled at the ends of the tensor.
2. From one to n number of axial strain values in directions relative to the requested orientation.

For anisotropic elements, the strain tensor directions will default to 0, 90, ±45°. For elements that reference a laminate, the default strain tensor directions will be 0, 90 and those directions corresponding to each lamina's orientation. The default angles can be modified and additional angles can be specified. The principal results can also be displayed as principal components at the element centroid with the same axes display.

Tensors can also be treated as two dimensional or three dimensional. The default is three dimensional, however the generic method will look at the data to determine whether to use the 2D or the 3D calculations. The 2D calculation method will ignore the YY, ZZ, and ZX components. The 3D method will treat zero component values as significant.

6.2 Target Entities

Marker plots can be displayed on various model entities. By default marker plots are displayed on all nodes displayed in the current viewport regardless of the results data type (elemental or nodal based). To change target entity selection for marker plots, press the Target Entities selection button with the Object set to Marker and the appropriate Method (Scalar, Vector, or Tensor) selected.



Toggles the form to select target display entities for marker plots.

The following table describes in detail which entities marker plots can be targeted.

Entity	Description
Current Viewport	By default all marker plots are displayed on all finite element nodes displayed in the currently active viewport. Elemental based results are extrapolated out to the nodes and averaged.
Nodes	Individual nodes may be selected on which to display the marker plot. You may type in any node numbers manually or by selecting them graphically from the screen. Be sure to include the word Node in front of the IDs you type in manually, (i.e., Node 1 5 55 100 etc.). To select all elements use the syntax "Node 1:#." Elemental based results are extrapolated to the nodes and averaged.
Elements	Individual elements may be selected on which to display the marker plot. You may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.). To select all elements use the syntax "Elem 1:#." Results are plotted at the element centroid and summed and averaged if necessary (for nodal results).
Groups	Marker plots can be limited to only selected groups. A selected group or groups must have elements or nodes in them otherwise the plot will not appear. A listbox allows selection of the group(s) to which the marker plot will be applied. This is handy in that the same finite element entities can belong to multiple groups. Only those groups selected will be displayed with a marker plot while all other non-selected groups will remain unaffected and retain their own display attributes.
Materials	Marker plots can be targeted at only those finite elements which have certain material properties assigned to them. A listbox appears allowing selection of the materials for whose elements will be targeted for a marker display.
Properties	Marker plots can be targeted at only those finite elements which have certain element properties assigned to them. A listbox appears allowing selection of the properties for whose elements will be targeted for a marker display.
Element Types	Marker plots can be limited to only certain element types also.

In addition to targeting the entities, the marker plot can also be isolated to attributes of the entities as described in the following table. When nodes and elements are specifically targeted for the plot these choices are not available. However for any other target entities (groups, materials, properties, current viewport, element types), these choices are available:

Attribute	Description
Nodes	By default all marker plots are posted on the nodes of the target entities, including elemental based data. Elemental based data are extrapolated to the nodes, summed and averaged.
Free Faces	Marker plots can be posted on to all free faces of the target entities. The markers are posted at nodes on the free faces only. Elemental based data are extrapolated and averaged.
Free Edges	This will display a maker plot on the edges that only have one common element. The markers are posted at nodes on the free edges only. Elemental based data are extrapolated and averaged.
Elements	Marker plots may be posted at element centroids. Data are summed and averaged for each element if necessary to produce a single data set for each element.
Corners	This will display markers only at corners of the model on nodes only. Elemental based data are extrapolated and averaged.
Target Deformations	By default, marker plots that are to be displayed on the deformed shape will be displayed on all deformation plots posted. You can select which deformation plots to target the marker plot to by selecting deformation plots form this listbox. This listbox will only appear when more than one deformation plot exists and is posted.

Important:	Once a target entity has been selected, it will remain the target entity for the marker plot until the user physically changes it.

6.3 Display Attributes

Marker plots can be displayed in various forms. Display attributes for marker plots are accessible by pressing the Display Attributes selection button on the Results application form with the Object set to Marker and the appropriate Method set (Scalar, Vector, or Tensor).

Toggles the form to change display attributes for marker plots with the method set to the type of desired marker (scalar, vector, or tensor).

Table 6-1 describes in detail the marker plot attributes which can be modified. They differ for each type of marker.

Table 6-1 Common Marker Display Attributes

Attribute	Description
Spectrum / Constant	This controls whether the marker colors (the vector arrows or other markers) are color coded according to the spectrum bar and value ranges or whether they are simply assigned different constant colors. If Constant is chosen you can select the colors for the resultant or components for vectors and tensors.
Show Spectrum	This toggle will remove or display the spectrum bar on the right of the current viewport. If the spectrum does not disappear from the graphics screen when the toggle is OFF, make sure other plots also do not have the spectrum turned ON.
Spectrum	This button will bring up the Spectrums form from which you can set the spectrum for the current viewport or create new spectrums. This form operates independently of the Results application. Therefore you do not need to press the Apply button in the Results application to effect changes in the spectrum. To learn more about changing the spectrum see **Display>Spectrums** (p. 306) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.
Range	This button will bring up a subordinate form for reassigning the range to the current spectrum in the current viewport. See **Display>Ranges** (p. 307) in the *MSC.Patran Reference Manual, Part 1: Basic Functions* for details. Also from this form you can invoke the Ranges form for creating and modifying new ranges. To learn more about changing ranges see **Display>Ranges** (p. 307) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.
Title Editor	Selecting this button opens a form that allows the marker plot title to be edited. See **Results Title Editor** (p. 31).
Show Title	If this toggle is ON, then a title for the marker plot is displayed. Otherwise no title is displayed.
Lock Title	If this toggle is ON, then the title for the marker plot is not modified by results form selections. Otherwise some results form selections modify the title.

Table 6-1 Common Marker Display Attributes (continued)

Attribute	Description
Show Max/Min Label Display	If this toggle is ON, then the maximum and minimum values of the selected results set are displayed in the viewport. Otherwise they are not displayed.
Show Scalar/Vector/ Tensor Label	If this toggle is ON, then the marker label (value) will be displayed at each marker.
Label Style	You can change the style of the numbers displayed on the marker label from the subordinate form that appears when you press this button. Styles can be integer, fixed, or exponential with specification of the number of significant digits. Also the label color is controlled from this form.
Show on Deformed	The marker plot will be displayed on the deformed shape of the model if a deformation plot has also been posted to the current viewport. The marker plot will display on all deformation plots if more than one is posted to the viewport. If this is not desired, see Target Entities (p. 105).

Table 6-2 Scalar Display Attributes

Attribute	Description
Constant - Colors	If Constant is selected as the display attribute for the scalars, all markers will be drawn with the same color.
Scalar Scale	The size of the marker can be constant or relative. Screen Constant scale will keep all markers the same size based on a percentage of the screen size set by the Scale Factor. This means markers will stay the same size no matter whether you zoom in on the model or not. Model Constant scale will keep all markers the same size based on a percentage of the model size set by the Scale Factor. Zooming in on the model will make the markers appear larger. Screen Scaled and Model Scaled will give variable size markers based on the value of the scalar result and scale each relative to the screen size or the model size set by the Scale Factor.
Scale Factor	This scale factor scales the size of the marker as described above.
Scale Style	Scale marker styles are: (1) triangle, (2) filled triangle, (3) square, (4) filled square, (5) diamond, (6) filled diamond, (7) hourglass, (8) filled hourglass, (9) cross, (10) filled cross, (11) circle, (12) filled circle, (13) dot, (14) sphere, (15) shaded sphere, (16) brick, (17) shaded brick

Table 6-3 Vector Display Attributes

Attribute	Description
Constant - Colors	If Constant is selected as the display attribute for the vectors you will be presented with color selection icons for the different components or for a single resultant. This is dependent on whether you have chosen to display the individual components or only the resultant from the main Select Results form. Component selection is represented as XX, YY, and ZZ for the three component directions of any coordinate system. These translate into , θ, z and r, ϕ, θ for cylindrical and spherical coordinate systems respectively.
Vector Length	The length of the vector can be constant or relative. Screen Constant vector length will keep all vector lengths the same size based on a percentage of the screen size set by the Scale Factor. This means vectors will stay the same length no matter whether you zoom in on the model or not. Model Constant vector length will keep all vector length the same size based on a percentage of the model size set by the Scale Factor. Zooming in on the model will make the vectors appear larger. Screen Scaled and Model Scaled will give variable length vectors based on the magnitude of each component (the actual result associated with each vector) and scale each relative to the screen size or the model size set by the Scale Factor.
Scale Factor	This scale factor scales the size of the actual vector length as described above.
Anchor Style	The vectors can be anchored either at the tip of the head or at the base of the arrow.
Head Style	One, two or no head can be placed on the end of a vector.
Line Style	Line styles can be either a line or a cylinder.

Table 6-4 Tensor Display Attributes

Attribute	Description
Constant - Colors	If Constant is selected as the display attribute for the tensor vectors you will be presented with color selection icons for the different components or for the principals. This is dependent on whether you have chosen to display the individual components or only the principals from the main Select Results form. Component selection is represented as XX, YY, ZZ, XY, YZ, and ZX for the six component directions of any coordinate system. These translate into , θ, z and r, ϕ, θ for cylindrical and spherical coordinate systems respectively. For 2D components, only XX, YY, and XY are presented and for 2D principals on the maximum and minimum principals are presented. For 3D display all components or principals are presented.
Vector Length	See **Scalar Display Attributes** (p. 108) description of vector length.
Scale Factor	See **Scalar Display Attributes** (p. 108) description of scale factor.
Head Style	See **Scalar Display Attributes** (p. 108) description of head style.
Line Style	See **Scalar Display Attributes** (p. 108) description of line style.
Show Tensor Box - Color	This toggle allows for the display of the tensor box around which are shown the components of the tensor. The color of this box is controllable.
Box Style	The box can appear as either wireframe (see through) or filled (opaque).
Box Scale	The same description for the length of the vectors applies for the size of the box as displayed on the model. See **Scalar Display Attributes** (p. 108) description of Vector Length.
Box Scale Factor	This scale factor controls the size of the tensor box.

Important:	Vectors and tensors will only animate properly if you set the Vector Length to Scaled. Constant vector lengths will not animate. Only the labels and sign changes will change during an animation. Also, once a target entity has been selected, it will remain the target entity for the particular marker plot until the user physically changes it.

6.4 Plot Options

Marker plots have various options. Plot options for marker plots are accessible by pressing the Plot Options selection button.

Toggles the form to select plot options for marker plots.

The following table describes in detail the marker tool options which can be modified:

Option	Description
Coordinate Transformation	Vector and tensor results to display marker plots can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran global system, a material coordinate system, element IJK coordinate system or the nodal (analysis) coordinate system depending on the type of result (vector, or tensor). See **Coordinate Systems** (p. 305) for a definition of each coordinate system. The default is no transformation, which will plot data in the coordinate frame as stored in the database. Typically the solver code will calculate results at nodes in the analysis coordinate system specified by the user. These can vary from node to node. Element data can be stored from the analysis code in any coordinate system. Note also that the analysis translators that import the results data into the database can transform results. Check with the appropriate translator guide.
Scale Factor	This scale factor has the effect of simply scaling the results up or down by the specified amount. This will have an effect on the fringe plot but will not effect the spectrum range values.
Filter Values	By specifying a filter value, a gate will be used to keep values below a maximum, above a minimum, between a certain range, or at the exclusion of certain values. The default is none. If filtering is used, only elements which pass the filter gate will display a marker plot on them.
Averaging Domain	For element based result quantities that must be displayed at nodes, an averaging domain must be used since more than one result will exist for each node. There is a contribution from each element attached to any particular node. By default all entities which contribute are used. Alternatively you can tell the Results application to only average results from those elements that share the same material or element property, are from the same target entities, or have the same element type. For more detail see **Averaging** (p. 292).

Option	Description
Averaging Method	The method in which certain results are determined can make a difference to the actual displayed plot. This is important when derived results from element based tensor or vector results are used such as von Mises stress or Magnitude displacements. For instance if you average at the nodes first and then derive the desired quantity you may get a different answer than if you derive first and then average. It is left up to the user to decide which is correct. For more detail see **Averaging** (p. 292).
Extrapolation Method	Many times element based results that are to be displayed at nodes exist at locations other than the nodes such as at integration points. Various methods are available to the user to extrapolate these results out to the nodes. For more detail see **Extrapolation** (p. 299).
Complex No. as	The Real component of a complex number is the default by which results will be postprocessed. To force the postprocessor to use a different quantity such as Magnitude, Imaginary, Phase, or Angle, set this option pull down menu. This option will only be available if a complex result has been selected. It is not recommended to calculate invariants (e.g., von Mises) from complex results because the phase is not accounted for.
Existing Scalar/Vector/ Tensor Plots	This listbox displays all existing marker plots. You may select one of these plots from the listbox and all settings of that plot including display attributes, target entities, option, and selected results will be restored. This is an easy mechanism to help make many plots with the settings of an existing plot without modifying the selected plot. When the Action is set to Modify, this listbox appears under the Select Results display of the Results application form.
Save Scalar/Vector/Tensor Plot As	Marker plots can be saved by name and recalled later for graphical display. Multiple marker plots can be saved in the database and displayed simultaneously. Be aware that when multiple marker plots are displayed simultaneously there could be some display problems if they are displayed on the same entities. These marker plots can be posted / unposted and deleted as explained in **Post/Unpost** (p. 25) and **Delete** (p. 27) respectively. Once a plot has been created and named it retains all results, attributes, target entities, and options assigned to it. If no plot name is specified a default is created called `default_Scalar, default_Vector` or `default_Tensor`. As long as no plot name is specified, the default name will be overwritten each time a plot is created or modified.

Important:	Once plot options have been selected, they will remain in effect for the marker plot until the user physically changes them.

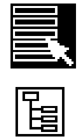
6.5 Examples of Usage

The following are some typical scenarios for usage of the Marker plot tool. These instructions assume that the Action is set to Create and the Object is set to Marker unless otherwise specified.

Create a Scalar Plot

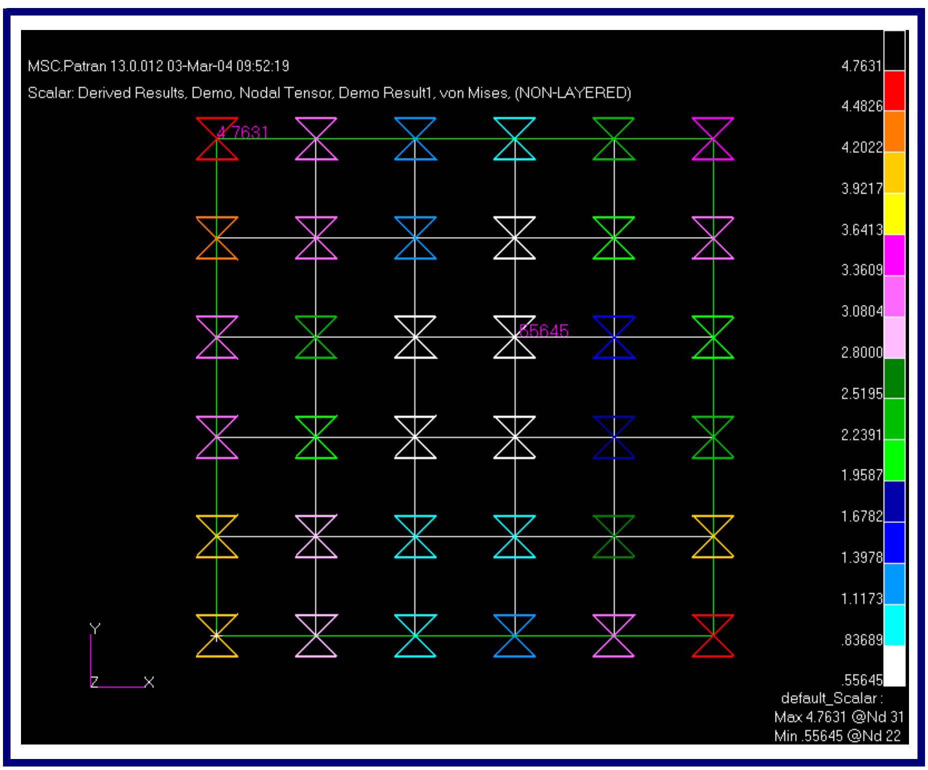
1. Set the Method to Scalar.

2. From the Select Results form (left most icon) select the Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.



3. Select the Scalar Result from the next listbox.
4. (Optional) If the result selected in the above listbox is a vector or tensor then select the scalar quantity to be derived from the selected vector or tensor result by selecting an item from the Quantity pull-down.

5. Press the Apply button with the Animate toggle OFF.

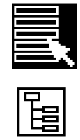


Create a Vector Plot of Displacement Data

1. Set the Method to Vector.

   Method: Vector

2. From the Select Results form (left most icon) select the Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.

3. Select the Vector Result (displacement) from the next listbox.

4. (Optional) Decide whether you want to display the vector plot as a resultant (single value) or as components (three values) with the Show As pull-down.

   Show As: Resultant

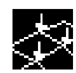
5. (Optional) If components are selected, toggle ON or OFF the components that you wish to display.

   XX YY ZZ

6. Press the Apply button with the Animate toggle OFF.

   Apply

   It is generally best to turn the vector labels OFF when plotting on the entire model. The display can get very cluttered with labels. This is done by pressing the Display Attributes icon and turning OFF the Show Vector Label toggle.

   ShowVectorLabe

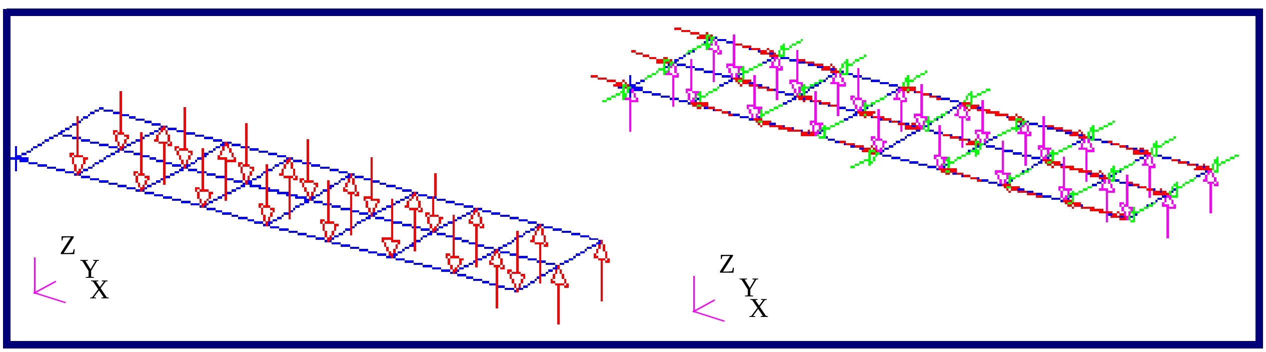


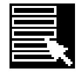
Figure 6-1 Resultant and Component Vector Plots of Displacement on Cantilever Plate (constant vector lengths).

Create a Tensor Plot of Principal Stresses

1. Set the Method to Tensor.

   Method: Tensor

2. From the Select Results form (left most icon) select the Result Case from the first listbox.

3. Select the Tensor Result (component stresses) from the next listbox.

4. (Optional) If more than one layer exists for these results, select the layer using the result Position button.

   Position...(at Z1)

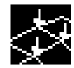
5. Decide which principals you want to display in the tensor plot with the Show As pull-down, and turning ON or OFF the desired components.

   Show As: Principal
   Max Mid Min

6. Press the Apply button with the Animate toggle OFF.

   Apply

   It is generally best to turn the Tensor Labels OFF when doing this on the entire mode to reduce label display clutter. This is done by pressing the Display Attributes icon and turning OFF the Show Tensor Label toggle.

   ShowTensorLabe

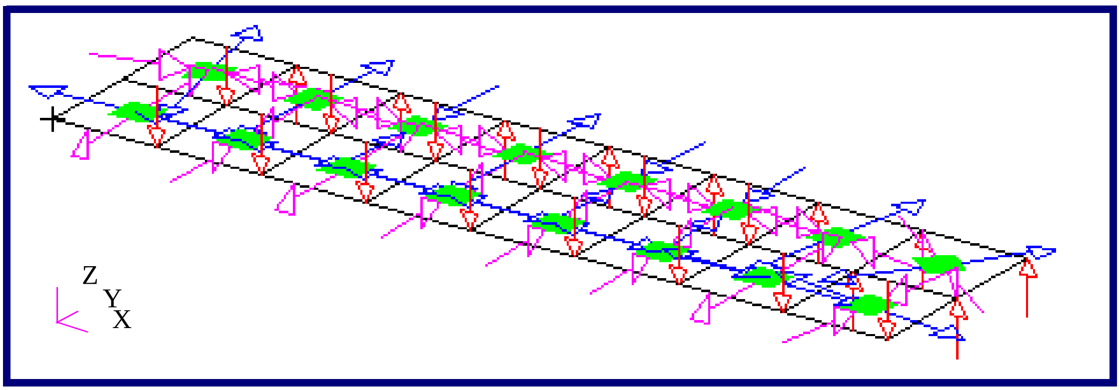
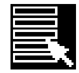


Figure 6-2 Principal Tensor Plot of Stresses on Cantilever Plate (constant vector lengths).

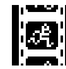
Animate a Static Scalar, Vector or Tensor Plot

1. Follow the general instructions to **Create a Scalar Plot** (p. 113) **Create a Vector Plot of Displacement Data** (p. 114) or **Create a Tensor Plot of Principal Stresses** (p. 115) for a vector or tensor plot respectively regardless of the actual result to be plotted, but do not press the Apply button.

2. Turn the Animate toggle ON on the main mode of the form.

   Animate 

3. (Optional) Press the Animation Options button icon.

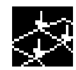
4. (Optional) A modal style animation will result by default. If a ramped style is preferred then change the Animation By pulldown to Ramp.

   Animate by: Ramp

5. Press the Apply button.

   Apply

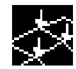
   Animation will proceed and will oscillate from the minimum (zero) to the maximum display which is based on the scale factor for either the constant or scaled vector lengths. Change the vector length settings in Display Attributes. Constant will make the animation appear as if only the labels and / or colors are changing unless the vectors change sign. Scaled will make the vectors appear to animate by scaling the actual size of the arrow based on result values.

   Length: Model Scaled

Put a Scalar, Vector or Tensor on a Deformed Plot

1. Create a deformation plot and make sure it is posted to the current viewport. See **Deformation Plots** (p. 55) for an explanation of deformation plot creation.
2. Now make a scalar, vector or tensor plot as explained in any of the examples above but don't press the Apply button.
3. Press the Display Attributes icon button.



4. Turn ON the Show on Deformed toggle.

5. Press the Apply button with the Animate toggle OFF.

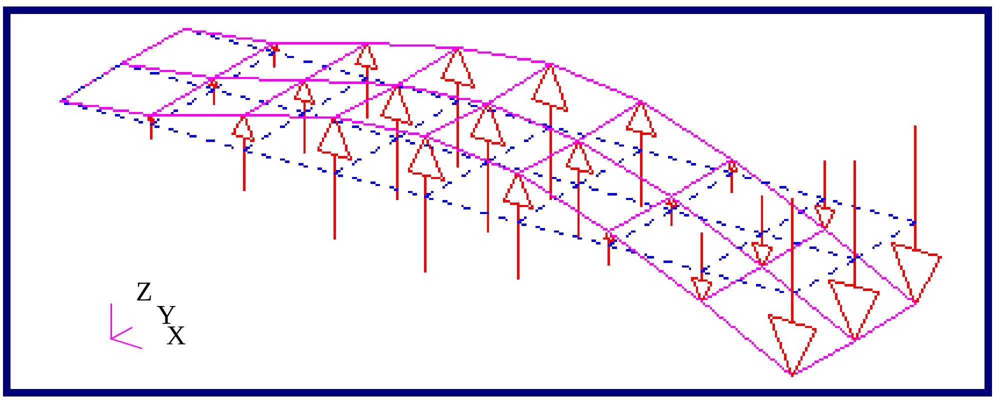
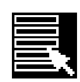


Figure 6-3 Vector Plot on a Deformed Shape (scaled vector lengths).

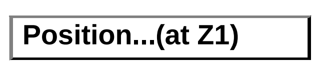
Display a Transient Scalar, Vector or Tensor Animation

1. Set the Method to Scalar, Vector or Tensor as desired.

2. From the Select Results form (left most icon) select the Result Cases (time steps), from the first listbox, that you wish to include in the transient animation. You must select more than one. Use the mouse and the control key to select discontinuous selections or the shift key to select continuous selections. You can also use the Select button, when the Result Cases are being displayed in abbreviated form, to filter and select the subcases (time steps) you want. In abbreviated form the Result Case name will only appear in the listbox as the name with number of subcases (time steps) selected, (i.e., Load Case 1, 6 of 41 subcases).



3. Select the Marker result from the next listbox.
4. (Optional) If the result quantity has more than one layer associated with it, select the layer from the result Position pull-down menu.



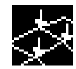
5. (Optional) If the results quantity is a tensor result and the Method is set to Vector, select a resolved vector quantity from the result Quantity pull-down menu (such as Maximum Principal stress).

   Quantity: Max Principal

6. Turn the Animate toggle ON. (If the Animate toggle is OFF then the resulting plot will simply be a maximum plot of all selected Result Cases and will not animate.)

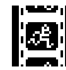
   Animate

7. (Optional) Set the Vector Length to Model Scaled or Screen Scaled under Display Attributes. This will result in the vector lengths being animated. Constant vector lengths will not animate and only the vector labels will change during the animation.

   Length: Model Scaled

8. Press the Animation Options icon button.

9. Select a global variable (time). Make any other optional modifications you want.

   Animate by: Global Variable

10. Press the Apply button. This method also works with load steps, frequency steps, or simply with multiple load cases that you may wish to animate.

   Apply

Save a Scalar, Vector or Tensor Plot

1. Set up the marker plot as explained in previous examples, but do not press the Apply button.
2. Before pressing the Apply button to create a plot or animation, press the Plot Options icon button.

3. Type a name in the Save Scalar / Vector / Tensor Plot As databox.

   Save Vector Plot As: myVector

4. Then press the Apply button.

   Apply

Display Multiple Marker Plots

1. Set up the first marker plot as explained in previous examples, but do not press the Apply button.
2. Save the marker plot as explained in **Save a Scalar, Vector or Tensor Plot** (p. 118).
3. (Optional) Make a different viewport the active current viewport if desired. This is done by placing the cursor at the edge of the viewport and clicking with the mouse. The current viewport will have a red border around it. This is the area where you click the mouse.

4. Repeat this process for as many marker plots as necessary and change the current viewport each time if desired.

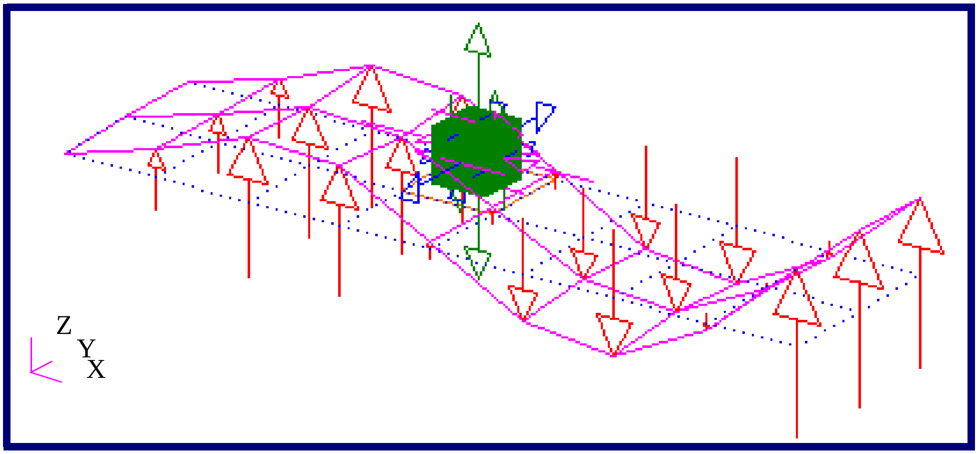


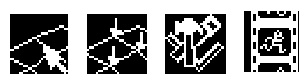
Figure 6-4 Display of a Vector and a Tensor Plot in the Same Viewport.

Modify a Marker Plot or Animation

1. Set the Action to Modify with the Object set to Marker.
2. Set the Method to Scalar, Vector or Tensor as desired.

3. Select an existing marker plot from the Existing Scalar/Vector/Tensor Plots listbox.

4. Change results, target entities, display attributes, plot or animation options as required.



5. Press the Apply button at any time to see the results of your modifications.

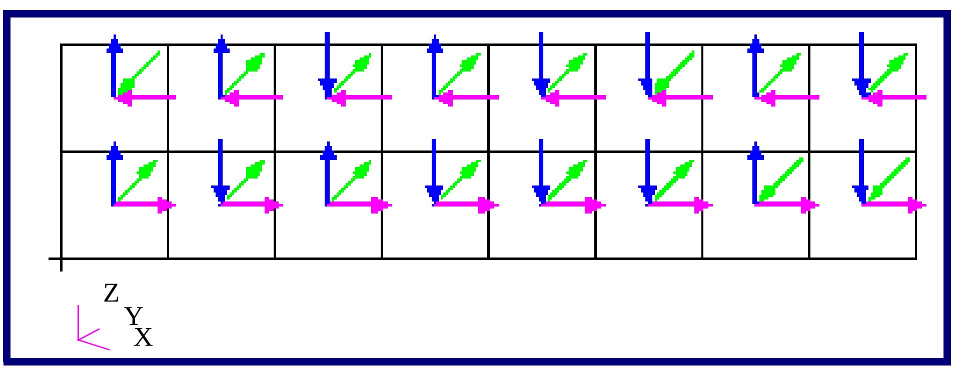
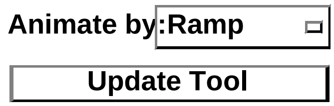


Figure 6-5 A Tensor Plot on the Cantilever Beam Modified to Display in Crow's Feet Form.

Display Multiple Animations in the Current Viewport

1. First create and save the marker plots (transient or static) as explained in **Display Multiple Marker Plots** (p. 118), but do not animate them. Do not turn ON the Animate toggle.
2. (Optional) Set the Action to Post and the Object to Plots. Post the plots to the viewports that you want if they are not already posted. They can be any plot type that supports animation.
3. Set the Action to Create and the Object to Animation. One by one, select the posted plots that you wish to animate from the top listbox and modify their animation method if necessary. This is done by pressing the Update Tool button.



4. Press the Apply button. See **Animation** (p. 153) for more details on this method.

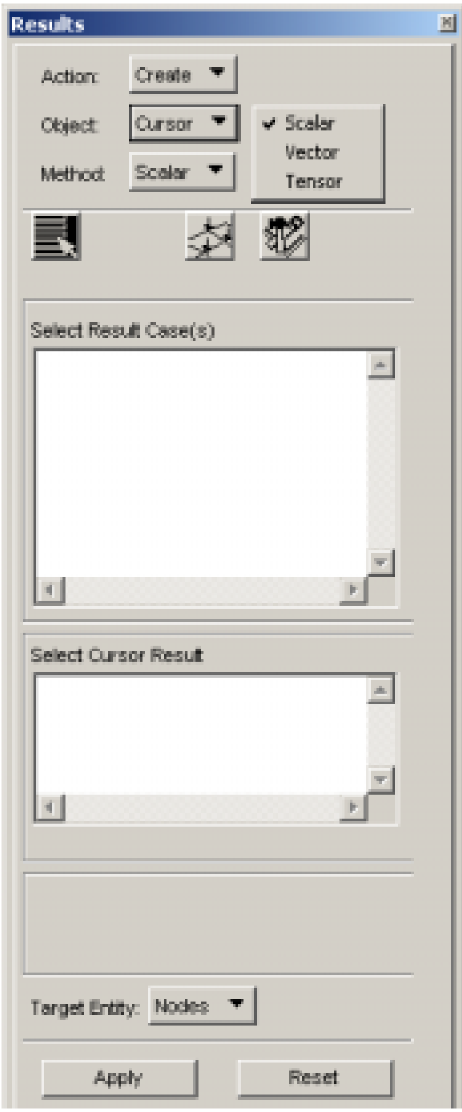
CHAPTER
7

Cursor Plots

7.1 Overview

For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). Cursor plots interactively display labels of either scalar, vector or tensor results on the model. To specifically make or modify a cursor plot select Create or Modify from the Action pull-down menu on the Results Display form; and select Cursor from the Object pull-down menu. Then set the Cursor type to Scalar, Vector or Tensor from the Method pull-down menu.

Application Form

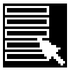


Method

Scalar cursor plots display labels representing the scalar result value at the selected entities. Vector cursor plots display three labels, one for each of the vector components, in a column vector format. Tensor cursor plots display six labels in the lower triangular portion of a 3x3 matrix. Typically results for tensor cursor plots are stress components.

Attributes and Options Toggles

- Select Results — This default setting establishes the application form to select the results used for the cursor plot.



- Display Attributes — Changes the application form to shows display attributes available.

- Plot Options — Changes the application form to show the plot options available.

Select Result Case(s) Specifies a results case to use for the cursor plot.

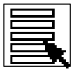
Select Cursor Result Sets the result from the available scalar, vector, or tensor results list.

Creating and Modifying a Cursor Plot

There is only a slight difference between Create and Modify. The main difference is that Create must be used to make a new Cursor plot and Modify is used to change an existing one. If you try to modify an existing plot with Create you will be asked for overwrite permission whereas Modify assumes that the action is desired, so no overwrite permission is requested.

To create a Cursor plot the following basic steps must be followed:

1. Set the Action to Create or Modify, the Object to Cursor and the Method to Scalar, Vector or Tensor.
2. Select a Result Case or Cases from the Select Results Case(s) listbox.

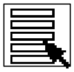


3. Select a result from the Select Scalar, Vector or Tensor Result listbox.
4. If more than one layer is associated with the results, select the layer (using the Position button) you wish to plot. These can be top or bottom results of shell elements, beam locations or laminate layers.
5. Optionally change the result Quantity. This is only possible if the selected result allows for this. If a tensor result has been selected for a vector Cursor plot, it must be resolved to a vector value. If a tensor or vector result has been selected for a scalar Cursor plot, it must be resolved to a scalar value. The various resolutions are:

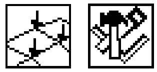
Tensor to Vector:	Minimum Principal, Middle (intermediate) Principal, Maximum Principal, Component, Minimum Principal 2D, Maximum Principal 2D
Vector to Scalar:	Magnitude, X component, Y comp., Z comp.
Tensor to Scalar:	von Mises, XX, YY, ZZ, XY, YZ, XZ, Maximum Principal, Middle Principal, Minimum Principal, Hydrostatic, 1st Invariant, 2nd Invariant, 3rd Invariant, Tresca, Max Shear, Octahedral. See hypertextparatextDefault Fontp.\

Vector to Scalar:	Magnitude, X Component, Y Component, Z Component
Tensor to Scalar:	von Mises, XX, YY, ZZ, XY, YZ, ZX, Maximum Principal, Middle Principal, Minimum Principal, Hydrostatic, 1st Invariant, 2nd Invariant, 3rd Invariant, Tresca, Max Shear, Octahedral. See hypertextparatextDefault ¶ Fontp.\ ).

6. Optionally, change the target entity type. This is done in the Target Entity pull-down menu at the bottom of the main Select Results form.



7. Optionally change display attributes, or invoke other plot options by changing these settings using the two other icons at the top of the form. These are described in detail later in this chapter.



8. Press the Apply button when ready to create the Cursor plot.
9. The Apply button causes the Cursor Data form to be created and displayed. The user may now select entities at which results values are desired. The result values are recorded in the spreadsheet on the Cursor Data form.

To modify an existing Cursor plot, simply follow the above procedure with the Action set to Modify. However, you must first select an existing plot using the Existing Cursor Plots button on the main form where results are selected. When an existing Cursor plot is selected, all results, attributes, and options in the various widgets associated with that plot are updated to reflect that plot's settings. You may then proceed to modify the plot.

By default a Cursor plot with the name

default_Cursor will be created unless the user specifically gives a different name. Multiple Cursor plots can be created by giving cursor plots unique names. Only one Cursor plot can be posted to the viewport. Each viewport can have its own Cursor plot or other plot types posted.

Each plot can have its own attributes. Each plot can also target or be displayed on separate entities and have its own associated options. These are detailed in the next sections.

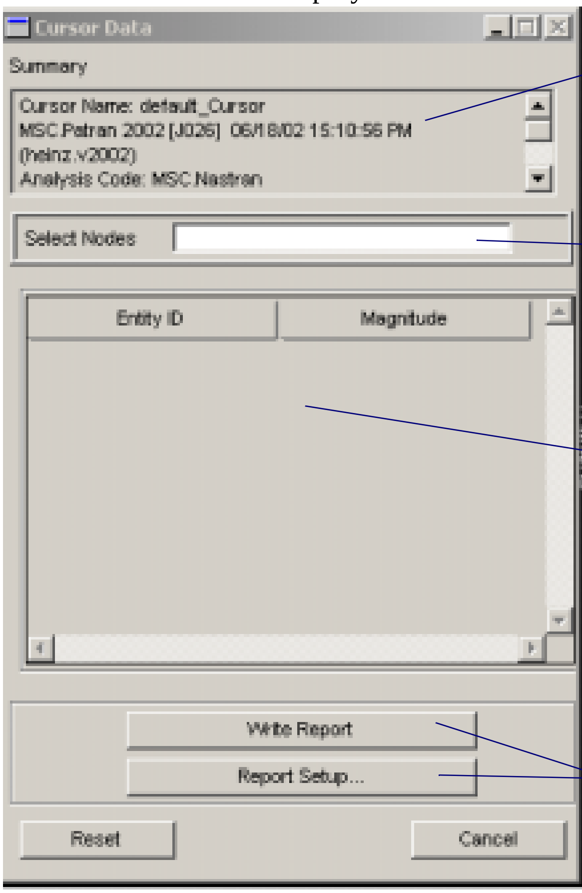
Component selection is represented as XX, YY, and ZZ for the three component directions of any coordinate system. These translate into r, ?, z and r, ?, ? for cylindrical and spherical coordinate systems respectively.

Cursor Data Form

To display this form go to the Results Application

1. Set AOM menus to Create / Cursor / Scalar
2. Select a Result Case, select a cursor result and hit the apply button. (Note: There must be a results model and results in the database.

The cursor plot differs from the other graphical results plots, in that there is not an immediate graphical feedback when the Apply button is pushed on the main form The Cursor Data form is displayed instead. The user may then graphically select desired entities at which to display the result value labels. This form must be displayed for the cursor tool in order to the allow the selection of the targeted entities type. Once this form is exited the cursor tool will no longer be active. If the user wishes to use select more entities for an existing tool then the user must post the tool which will re-display this form.



The Summary textbox displays the title information for the Cursor Plot including cursor name, selected result cases, selected result type, target entity type and minimum and maximum values for selected result.

In this example the target entity type is set to nodes. This select databox allows the user to interactively select nodes at which the result values will be displayed. The user may also manually type desired nodes into the select databox

The result values for the selected entities are displayed in this spreadsheet. The first column of data is always the Entity ID. Scalar, vector and tensor results have one, three and six column(s), respectively of additional data, A maximum of four visible columns are displayed in this spreadsheet. Tensor results require the user to use the horizontal scroll bar to see the last three columns of data.

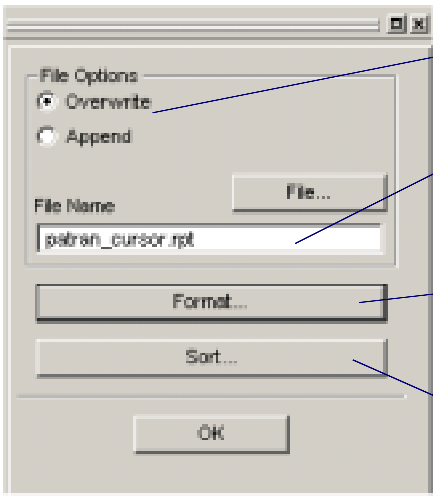
The data collected in the spreadsheet may be output to a file by pressing the Write Report Button. The Report Setup button allows the user to customize the output of the report.

Cursor Report Setup

The Cursor Report Setup form is used to change the default report setup for the Cursor plot. The cursor reports can be formatted in a variety of ways and with many options.

To display this form go to the Results Application

1. Set AOM menus to Create / Cursor / Scalar
2. Select a Result Case, select a cursor result and hit the apply button. (Note: There must be a results model and results in the database.)
3. Hit the Report Setup button on the Cursor Data form.



The cursor data may be output by either overwriting or appending data to a file.

A file name can be specified in the File Name databox. A file browser is also available if you press the File button to select an existing file. The default cursor report filename is `cursor_patran.prt.`

A subordinate form will appear to allow for report formatting. This is explained below in **Cursor Report Format**.

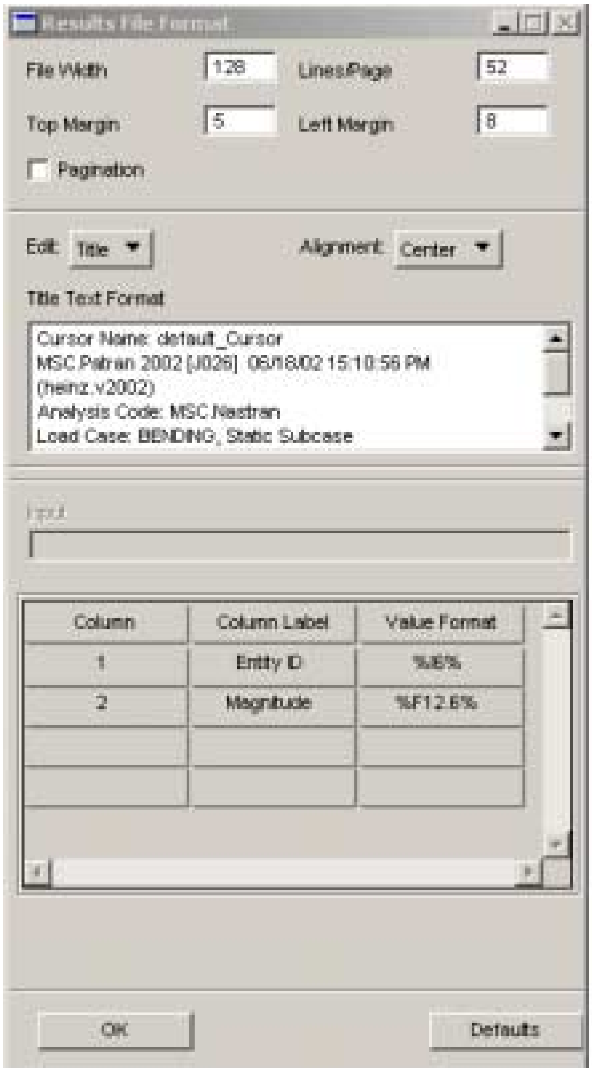
A subordinate form will appear to allow for selection of sorting options. This is described below in **Sorting Options** (p. 87)

Cursor Report Format

The Cursor Report Format is used to set the specific format of the data output to file including title text and alignment, pagination, column labels and the output format of the data. The formatting options are described in Cursor Report Format Options table.

[To display this form go to the Results Application

1. Set AOM menus to Create/Cursor/Scalar
2. Select a Result Case, select a cursor result and hit the apply button. (Note: There must be a results model and results in the database.)
3. Press the Report Setup button on the Cursor Data form.



Cursor Report Format. Below are descriptions for all the fields and settings for formatting cursor reports.

Table 7-1 Cursor Report Format Options

Item	Description
File Width	Sets the number of characters that can fit in the width of a page including spaces. The default is 128 characters.
Lines/Page	Sets the number of lines per page. The default is 52 lines per page.
Top Margin	Sets the number of lines used to form a top margin. The bottom margin is set by the number of Lines / Page.
Left Margin	Sets the number of characters used to make a left margin. The right margin is set by the File Width.
Pagination	If you wish to use pagination turn this toggle ON. The Page Number setting will appear to set the beginning page number. No footer or header information will be printed.
Page Number	Set the beginning page number with this option. This databox only appears if Pagination has been turned ON.
Edit	This is an option menu for editing the Title, Footer, or Header. No Footer or Header is allowed if pagination is OFF. This text box below this menu will update to allow for editing of the selected text.
Alignment	Alignment of the report can be from the left margin, right margin or the report can be centered.
Title/Header/Footer Text Format	This textbox allows for inclusion and modification of a Title, Header or Footer. Which is set for editing is determined by the Edit option menu above this text box. A %I% may be placed in any of these text boxes to include the page number if Pagination has been turned ON. A %rN% may also be included for adding additional blank lines. These formatting characters are explained below.
Input Column/ Label/Format	This is a databox that becomes active to allow for changes in the actual Column Labels or Value Formats. Simply click on a cell in the spreadsheet that appears below this databox and the databox will become active to allow you to change the cells contents.
Column	This is the column number with its associated label and value format.

Column Label	This is the label that appears above the column of results. By default it is the same as results quantities selected. To change a label, select the cell and then change the value in the databox above. Press the Enter or Return key to effect the change.
Value Format	Results formats are listed in this column. They specify how the actual results values will be formatted in the report. They consist of the format characters surrounded by percentage signs. To change one of these formats, click on the cell that contains the format to change and enter your changes in the databox above the spreadsheet. Press the Enter of Return key to effect the change. The different characters and combinations acceptable for these formats is explained in .

Format Strings. The format string is a description of how to convert integer, real, and string data to a formatted output. It is necessary to use a format string in the Value Format column in the spreadsheet to specify how to format the results values in the report. Some of these formats can also be used in the Title, Header and Footer. The format string is a simple character string which contains both raw text to output, and format specifiers, enclosed by a set of percent characters, which control how data items are formatted and output. Upper case letters (I, F, E, etc.) are interpreted literally and lower case letters are to be substituted with the appropriate values.

To change a value format simply click the mouse button with the cursor in the cell whose format you wish to change. Then in the Input databox above the spreadsheet change the value format to what you want and then press the Return or Enter key.

Table 7-2 Value Format Strings for Formatting Text Report Numbers

Format	Description
%%	The simplest form of format specifier is a double percent to produce a single percent in the final output. Used if you want a percent character in the Title, Header or Footer.
%Im%	Integer (I) specifier. This format specifier takes an integer value such as a node or element (entity) ID or other integer result for formatting. The value of “m” is the minimum number of characters to produce from the format. If “m” is omitted, then the exact number of characters necessary to hold the integer is used. The exact format produced is an optional minus sign followed by one or more digits. The default for integer data is %I6%.
%Fm.n%	Fixed (F) float specifier. This format specifier takes a real results value for formatting in fixed point notation. The value of m is the minimum number of characters to produce from the format. If m is omitted, then the exact number of characters necessary to hold the conversion is used. The value of n is the number of decimal digits to produce. If omitted, then all significant digits will be produced. The exact format produced is an optional minus sign followed by zero or more digits, a decimal point, and zero or more digits. At least one digit will precede or follow the decimal point. The default for real data is %F12.6%.

%Em.n.p%	Exponential (E) float specifier. This format specifier takes a real value for formatting in scientific exponential notation. The value of m is the minimum number of characters to produce from the format. If m is omitted, then the exact number of characters necessary to hold the conversion is used. The value of n is the number of decimal digits to produce. If omitted, then all significant digits will be produced. The value of p is the number of digits to display before the decimal point, and defaults to one. If zero is specified, then a single zero precedes the decimal point. The exact format produced is an optional minus sign followed by zero or more digits, a decimal point, zero or more digits, a capital E, a plus or minus sign, and two decimal digits. At least one digit will precede or follow the decimal point. The default value for read data is the F format.
%Gm.n.p%	General (G) float specifier. This format specifier takes a real value for formatting in either F or E format. The format used depends on the value of the number to convert. In general, if the exponent is small, the F format will be used, otherwise the E format is used. See the descriptions of the F and E formats.
%Sm%	String (S) specifier. This format specifier takes the next string value from the character data array for formatting. The value of m is the minimum number of characters to produce from the format. If m is omitted, then the exact number of characters in the string is used. The default value for string data is %S32%.
%rN%	New (N) line specifier. This format specifier causes a new line to be started. The previous line is output as is, and formatting starts at column one of the new line. The value of r is a repeat count for skipping multiple lines. If output is to a string, then new line characters will be written to the string. This is used in the Title, Header and Footer text.

Variables. Variables can be placed in titles, footers, or headers of reports. The variables available are shown in the table below. Be sure to place the $ symbol in front of the variable otherwise it will not be recognized as a variable.

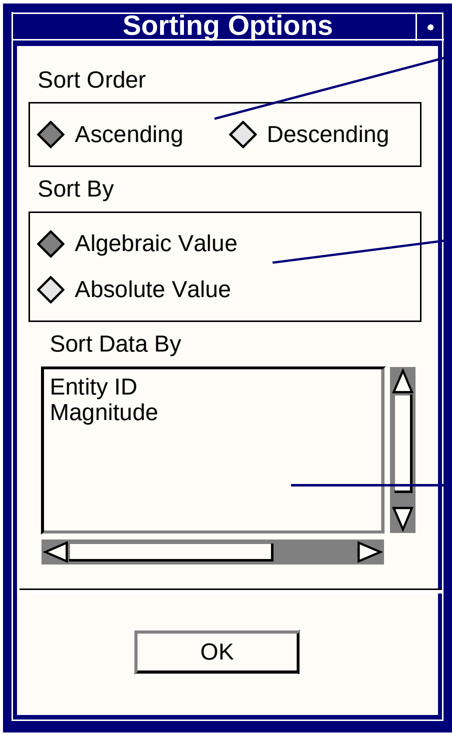
Table 7-3 Value Format Strings for Formatting Text Report Numbers

Format	Description
$LC_NAME	This is the Result Case (load case) name.
$SC_NAME	This is the subcase name.
$PRES_NAME	This is the primary result name.
$SRES_NAME	This is the secondary result name.
$LYR_NAME	This is the result layer name.
$DATE	The current date and time in the format dd-mmm-yy hh:mm:ss.
$PAGE	The current report page number.
$NNODES	The number of nodes in the report. Variable is printed in I9 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$MAXNOD	The highest ID of a node in the file. Variable is printed in I9 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$DEFMAX	The maximum value encountered within the file. Variable is printed in E15.6 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$NDMAX	The ID of the node with the maximum value. Variable is printed in I9 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$NWIDTH	The number of columns in the file. This will be the number of results quantities output to the report. Note that the Entity Id which is the first column of most reports by default is not included in NWIDTH. It is actually the number of columns of real, floating point data. Typically this is used to create MSC.Patran nodal (nod) and elemental (els) result files.
$DATA_TITLE	The register title. You must use the built in function res_data_title() to set a title for your register. Once this title is set, then it will show up when you use $DATA_TITLE.
$PRODUCT	The MSC.Patran product/version.
$DB_NAME	The name of the current database.
$JOB_NAME	The name of the analysis job.
$CODE_NAME	The name of the analysis code as set under Preferences/Analysis.

$GV:<name>	The name and value of an associated global variable such as time, frequency, eigenvector, etc. If a global variable is one word then all that is needed is to specify that global variable after the colon, i.e., $GV:Time. However, if a global variable name has a space in it or, that is, consists of more than one word, you must surround the name with single quotes, i.e., $GV:'Design Cycle.' Failing to do this will results in the variable picking up only the first word and will not find the correct global variable and will report garbage. Using this variable in the header and footer when multiple results cases (multiple GVs) will only use the first global variable encountered.
$LEFT	Aligns the current line of text to the left, overriding the global page alignment.
$MIDDLE	Aligns the current line of text to the middle, overriding the global page alignment.
$RIGHT	Aligns the current line of text to the right, overriding the global page alignment.

Sorting Options

Results can be sorted in a report and sorting is controlled via this form which is available from the Cursor Report Setup form by pressing the Sort button.



Results can be sorted in Ascending (from smallest to largest) or Descending order. Ascending order is the default.

Sorting can be done by comparing Algebraic Values which considers the sign of a value. A negative value will be treated as less than a positive number. Or the Absolute Value of the results can be used where sign is ignored, and the relative size in magnitude is considered in the sort.

This listbox displays the result entity by which the sort is based. By default sorting is always done and is based on the Entity ID. This would be a node or element number in general.

7.2 Target Entities

Cursor plots can be displayed on various model entities. By default Cursor plots are targeted on all nodes displayed in the current viewport regardless of the results data type (elemental or nodal based). To change target entity selection for Cursor plots, select an entity type from the Target Entity option menu on the Select Results form.

The following table describes in detail which entities Cursor plots can be targeted.

Entity	Description
Nodes	Individual nodes may be selected on which to display the Cursor plot. You may type in any node numbers manually or by selecting them graphically from the screen. Be sure to include the word Node in front of the IDs you type in manually, (i.e., Node 1 5 55 100 etc.). Elemental based results are extrapolated to the nodes and averaged.
Elements	Individual elements may be selected on which to display the Cursor plot. You may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.).Results are plotted at the element centroid and summed and averaged if necessary (for nodal results).
Isosurfaces	
Streamline	
Path	

Important: Once a target entity type has been selected, it will remain the target entity type for the Cursor plot until the user physically changes it.

7.3 Display Attributes

Cursor plots can be displayed in various forms. Display attributes for Cursor plots are accessible by pressing the Display Attributes selection button on the Results application form with the Object set to Cursor and the appropriate Method set (Scalar, Vector or Tensor).

Toggles the form to change display attributes for Cursor plots with the method set to the type of desired Cursor (scalar, vector or tensor).

Table 7-4 Common Cursor Display Attributes

Attribute	Description
Title Editor	Selecting this button opens a form that allows the cursor plot title to be edited. See **Results Title Editor** (p. 31).
Show Title	If this toggle is ON, then a title for the cursor plot is displayed. Otherwise no title is displayed.
Lock Title	If this toggle is ON, then the title for the cursor plot is not modified by results form selections. Otherwise some results form selections modify the title.
Label Style	You can change the style of the numbers displayed on the Cursor label from the subordinate form that appears when you press this button. Styles can be integer, fixed, or exponential with specification of the number of significant digits. Also the label color is controlled from this form.
Show on Deformed	The Cursor plot will be displayed on the deformed shape of the model if a deformation plot has also been posted to the current viewport. The

7.4 Plot Options

Cursor plots have various options. Plot options for Cursor plots are accessible by pressing the Plot Options selection button.

Toggles the form to select plot options for Cursor plots.

The following table describes in detail the Cursor tool options which can be modified:

Option	Description
Coordinate Transformation	Vector and tensor results to display Cursor plots can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran global system, a material coordinate system, element IJK coordinate system or the nodal (analysis) coordinate system depending on the type of result (vector, or tensor). The default is no transformation, which will plot data in the coordinate frame as stored in the database. Typically the solver code will calculate results at nodes in the analysis coordinate system specified by the user. These can vary from node to node. Element data can be stored from the analysis code in any coordinate system. Note also that the analysis translators that import the results data into the database can transform results. Check with the appropriate translator guide.
Scale Factor	This scale factor has the effect of simply scaling the results up or down by the specified amount.
Averaging Domain	For element based result quantities that must be displayed at nodes, an averaging domain must be used since more than one result will exist for each node. There is a contribution from each element attached to any particular node. By default all entities which contribute are used. Alternatively you can tell the Results application to only average results from those elements that share the same material or element property, are from the same target entities, or have the same element type.
Averaging Method	The method in which certain results are determined can make a difference to the actual displayed result value. This is important when derived results from element based tensor or vector results are used such as von Mises stress or Magnitude displacements. For instance if you average at the nodes first and then derive the desired quantity you may get a different answer than if you derive first and then average. It is left up to the user to decide which is correct.
Extrapolation Method	Many times element based results that are to be displayed at nodes exist at locations other than the nodes such as at integration points. Various methods are available to the user to extrapolate these results out to the nodes.

Existing Cursor Plots	This listbox displays all existing Cursor plots. You may select one of these plots from the listbox and all settings of that plot including display attributes, target entities, option, and selected results will be restored. This is an easy mechanism to help make many plots with the settings of an existing plot without modifying the selected plot. When the Action is set to Modify, this listbox appears under the Select Results display of the Results application form.
Save Cursor Plot As	Cursor plots can be saved by name and recalled later for graphical display. Multiple Cursor plots can be saved in the database. These Cursor plots can be posted / unposted and deleted respectively. Once a plot has been created and named it retains all results, attributes, target entities, and options assigned to it. If no plot name is specified a default is created called default_Cursor. As long as no plot name is specified, the default name will be overwritten each time a plot is created or modified.

Important: Once plot options have been selected, they will remain in effect for the Cursor plot until the user physically changes them.

7.5 Examples of Usage

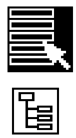
The following are some typical scenarios for usage of the Cursor plot tool. These instructions assume that the Action is set to Create and the Object is set to Cursor unless otherwise specified.

Create a Cursor Plot of von Mises Stress

1. Set the Method to Scalar.

Method: Scalar

2. From the Select Results form (left most icon) select the Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.



3. Select the Stress Tensor from the Select Cursor Result listbox.
4. Select von Mises from the result Quantity menu.

Quantity: von Mises

5. Set the entity type from the Target Entity menu to Elements.

Target Entity: Elements

6. Press the Apply button.

Apply

7. Once the Cursor Data Form is displayed the user may graphically select the desired entities. The von Mises value will be displayed on the model.

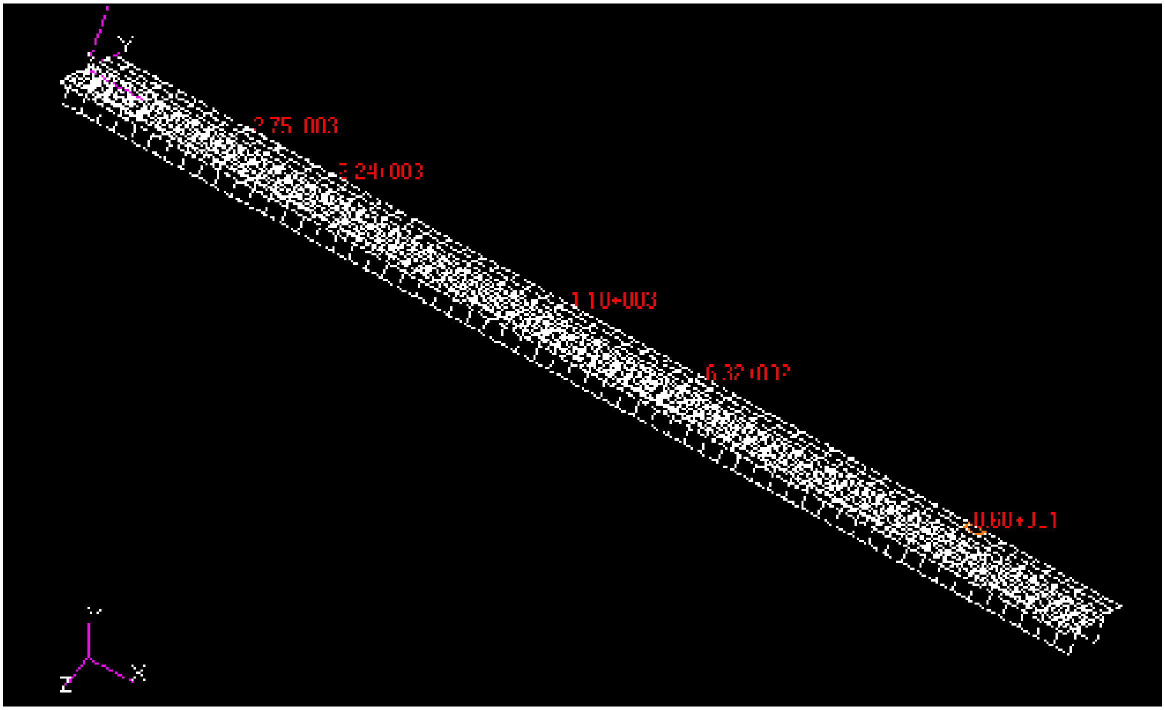


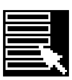
Figure 7-1 Cursor Plot of von Mises stress.

Create a Cursor Plot of Displacement Data

1. Set the Method to Vector.

Method: **Vector**

2. From the Select Results form (left most icon) select the Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.



3. Select the Displacements from the Cursor Result listbox.
4. Set the entity type from the Target Entity menu to Nodes.

Target Entity: **Nodes**

5. Press the Apply button.

6. Once the Cursor Data Form is displayed the user may graphically select the desired nodes. The displacement components will be displayed on the model.

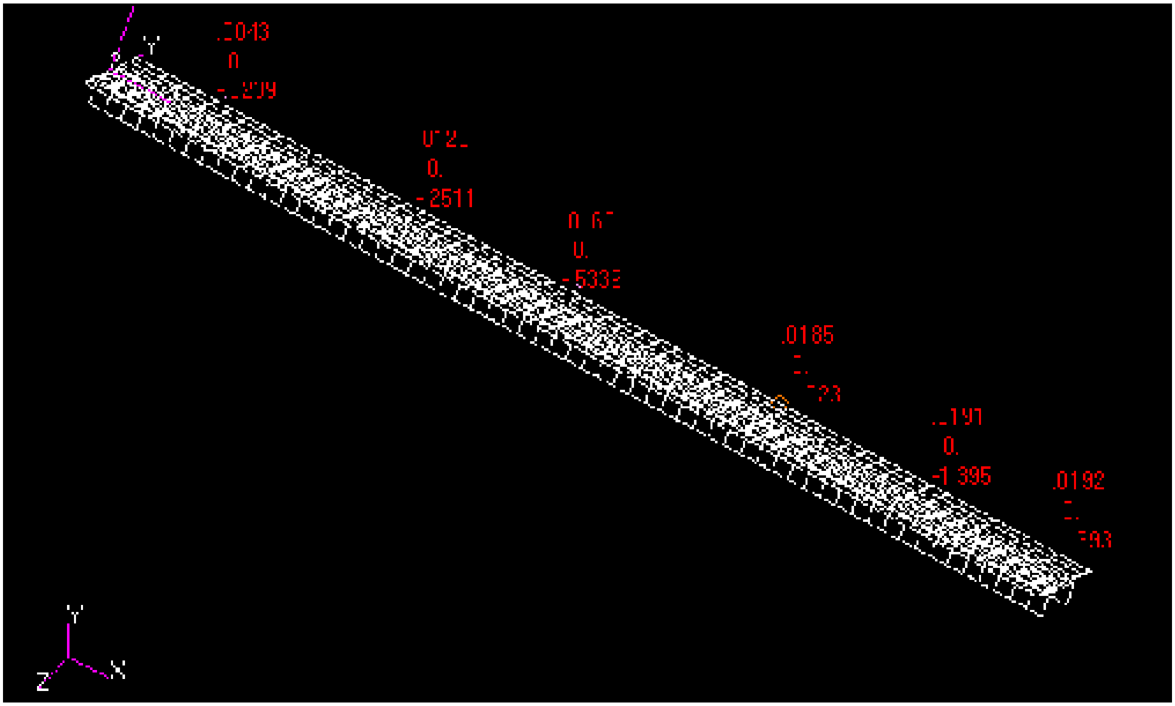
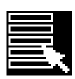


Figure 7-2 Cursor Plot of Vector Components of Displacement.

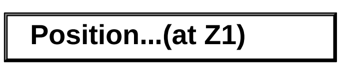
Create a Tensor Plot of Stresses

1. Set the Method to Tensor.

2. From the Select Results form (left most icon) select the Result Case from the first listbox.



3. Select the Stress Tensor from the Cursor Result listbox.
4. (Optional) If more than one layer exists for these results, select the layer using the result Position button.



5. Set the entity type from the Target Entity menu to Elements.

6. Press the Apply button.

7. Once the Cursor Data Form is displayed the user may graphically select the desired elements. The stress components will be displayed on the model.

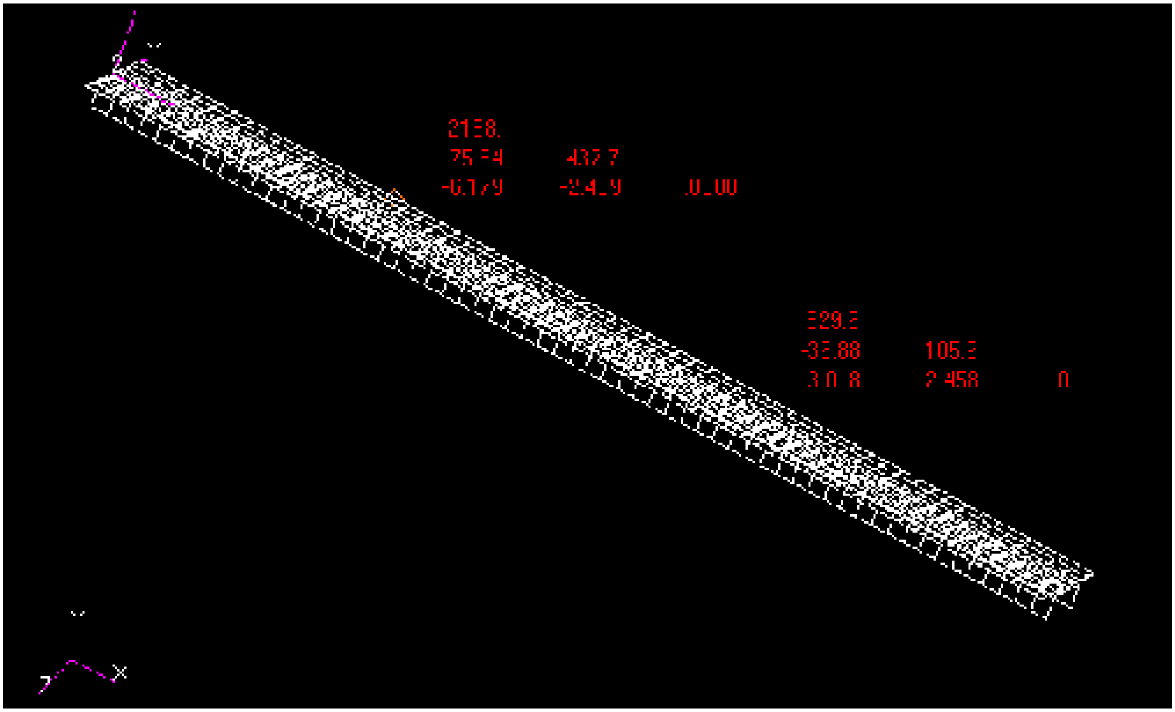


Figure 7-3 Stress Components.

Put a Cursor on a Deformed Plot:

1. Create a deformation plot and make sure it is posted to the current viewport. See for an explanation of deformation plot creation.
2. Now make a cursor plot as explained in any of the examples above but don't press the Apply button.
3. Press the Display Attributes icon button.

4. Turn ON the Show on Deformed toggle.

■ **Show on Deformed**

5. Press the Apply button.

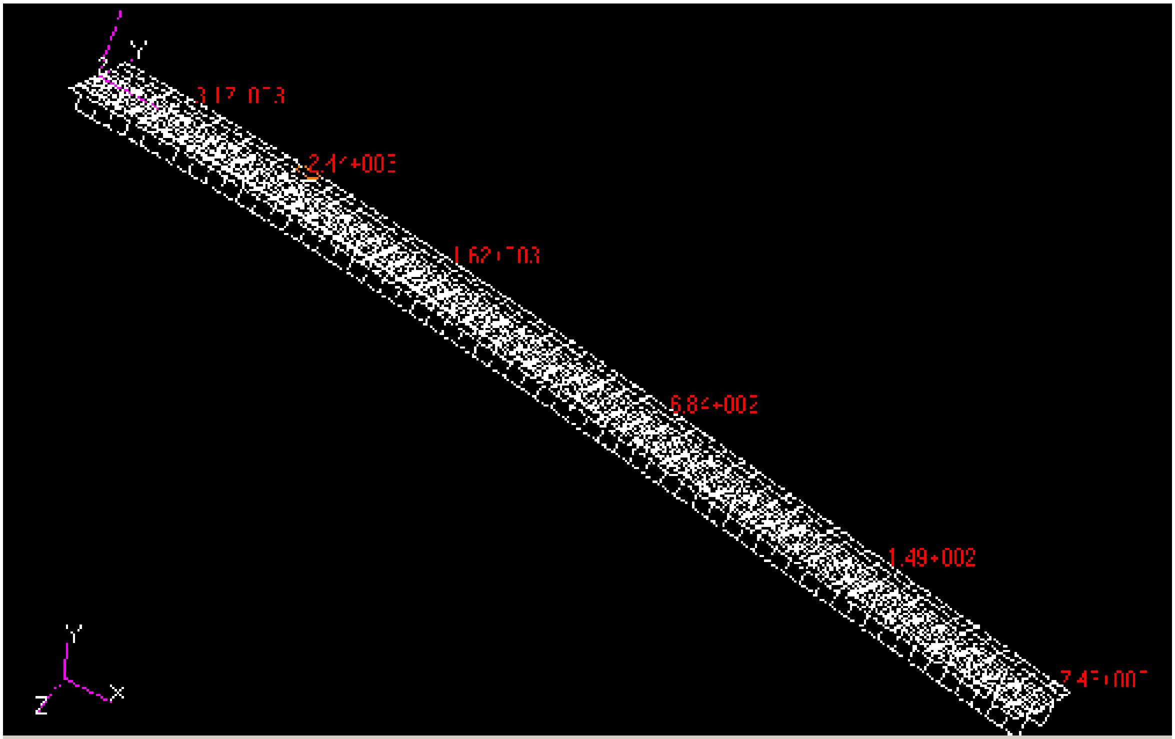


Figure 7-4 Cursor Plot on a Deformed Shape .

Save a Cursor Plot

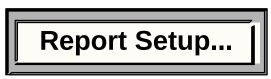
1. Set up the Cursor plot as explained in previous examples, but do not press the Apply button.
2. Before pressing the Apply button to create a plot, press the Plot Options icon button.

3. Type a name in the Save Cursor Plot As databox.

4. Then press the Apply button.

Report of Cursor Data

1. After creating a cursor plot and populating the Cursor Data form spreadsheet with the desired values, the data in spreadsheet may be output to a file.
2. By default, a new file will be created called `patran_cursor.prt`
3. If the user wishes to customize the report, formatting and sorting options are available from the Report Setup form. To access these, press the Report Setup button on the Cursor Data form.



4. Press the Write Report button to generate a report file.

The report should look somewhat similar to this:

Figure 7-5 Report of Displacement Data with Summary.

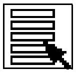
CHAPTER 8

Graph (XY) Plots

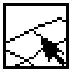
- Overview
- Target Entities
- Display Attributes
- Plot Options
- Examples of Usage

8.1 Overview

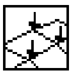
For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). To specifically make or modify a graph plot select Create or Modify from the Action pull-down menu on the Results Display form; and select Graph from the Object pull-down menu. Presently only Y vs X plots may be created.



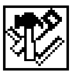
Selecting Results (p. 15)



Target Entities (p. 137).



Display Attributes (p. 140).



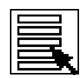
Plot Options (p. 142).

There is only a slight difference between Create and Modify. The main difference is that Create must be used to make a new graph plot and Modify is used to change an existing one. If you try to modify an existing plot with Create you will be asked for overwrite permission whereas Modify assumes that the action is desired, so no overwrite permission is requested.

Toggles the form to select results for graph plots. This is the default mode of the Graph form.

For both Modify and Create the same basic operations and options are available. To create or modify a graph plot the following basic steps must be followed:

1. Set the Action to Create or Modify and the Object to Graph.
2. Select a Result Case or Cases from the Select Results Case(s) listbox. See **Selecting Results** (p. 15) for a detailed explanation of this process as well as **Filtering Results** (p. 18).



3. Set the Y-axis value to either a Results value or a Global Variable if multiple Result Cases are selected. The Global Variables available depend on the result type but are generally time, frequency or load step. If a Result value is chosen, then select the result from the Select Y Result listbox otherwise select a Global Variable. Global variables will only be available if multiple Result Cases have been selected. Skip to step 5 if a global variable has been set for the Y-axis.
4. For Result Y-axis values, if more than one layer is associated with the result, select the layer (using the Position button) you wish to plot.

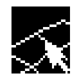
5. For Result Y-axis values, optionally change the results Quantity. This is only possible if the selected result allows for this. If a vector or tensor result has been selected for a graph plot, it must be resolved to a scalar value. The various resolutions are:

ector to Scalar: Magnitude, X Component, Y Component, Z Component.

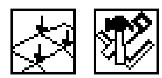
ensor to Scalar: von Mises, XX, YY, ZZ, XY, YZ, XZ, Minor, Intermediate, Major, Hydrostatic, 1st Invariant, 2nd Invariant, 3rd, Invariant, Tresca, Max Shear, Octahedral. See **Derivations** (p. 286).

6. Set the X-axis value to either to Result, Global Variable, Coordinate Axis, Path Length or Beams and select the necessary items as was done for the Y-axis values. More detailed explanations of the X- and Y-axis types are given below. Path Lengths (points, curves and element edges) and Beams are selected as target entities.

7. Select the target entities. For most plot types this is an optional activity but for a Graph plot it is required unless the graph is a global variable versus another global variable. To do this press the **Target Entities** (p. 137) button icon and select the nodes, elements, beams, curves, or path for which you wish to create an Graph plot.



8. Optionally change any display attributes, or invoke other plot options by changing these settings using the other two icon buttons at the top of the form. These are described in detail later in this chapter. See **Display Attributes** (p. 140) and **Plot Options** (p. 142).



9. Press the Apply button when ready to create the graph.

To Modify an existing graph, follow the above procedure with the exception that you must first select an existing plot using the Existing Graph Plots button on the main form where results are selected. When an existing graph is selected, all results, attributes, and options in the various widgets associated with that plot are updated to reflect that plot's settings. You may then proceed to modify the plot.

By default a graph plot named `default_Graph` will be created unless the user specifically gives a different name. Multiple graph plots can only be created and posted by giving separate names. Multiple graph plots can be posted to the same XY window or to separate XY windows. Each XY window can have its own set of graphs posted.

Each plot can have its own attributes. Each plot can also target or be associated with separate entities and have its own associated options. These are detailed in the next sections.

The Results application, when producing a graph, actually is driving the XY Plot application by creating XY windows and curves and setting display attributes. It is important to understand the interaction between the XY Plot application and the Results application. Many more display attributes and other options are available and modifiable from the XY Plot application than can be controlled from the Results application. Although a Graph plot can be posted/unposted and modified from the Results application, more versatile controls are found under XY Plot.

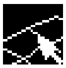
Care should also be taken when naming Graph plots in the Results application since each graph can create a new XY window with its own name. If two Graph plots share the same XY window, things may happen that do not quite make sense until you understand the interaction between Results and XY Plot. See **Overview of the XY Plot Application** (Ch. 1) in the *MSC.Patran Reference Manual, Part 7: XY Plotting*.

X and Y Axis Values. The following table explains the different X and Y axis values and what each can be plotted against.

X or Y Value	Description
Result	A result value on the Y-axis can be plotted against another results value for the selected target entities or it may be plotted against a global variable such as time, frequency, or load step. A Y-axis result value may also be plotted against coordinate locations, or locations along a curve or beam. To select a result you simply select the result from the listbox presented whether it be for the X or the Y axis. A subordinate form will appear for the X-axis result selection but functions identically to that for the Y-axis. Since only scale values can be plotted, you must select resolved values if a tensor or a vector is chosen. If multiple layers are associated with a result you must also select the desired layer such as top or bottom stresses of a plate element.
Global Variable	A global variable is a single value associated with a particular Result Case such as the time of a time step, the frequency of a mode shape, or the load step number of a non-linear analysis. These values can be plotted against result values (Y-axis) to give you transient type graphs or they can be plotted against other global variables (X-axis).
Coordinate	Result values (Y-axis) can be plotted against coordinate locations (X-axis). To choose the X-axis coordinate, use the Select Coordinate Axis databox to graphically select the coordinate frame and the desired direction (vector component- Z, Y, Z; R, θ, Z; R, Φ, θ) or points that define a vector direction. This is useful for plots such as a stress gradient as a function of distance from a hole.
Path Length / Beam	You can define a geometric curve from which a graph may be generated where the X-axis is defined as the distance along the curve. Use the Target Entities option to specify which curve, points, or edges/beams will actually make up the path. See **Target Entities** (p. 137).

8.2 Target Entities

Graphs can be displayed for various model entities. There is no practical default for graph plots and some sort of target entity must be selected, be they nodes, elements, or paths. To change target entity selection for graph plots, press the Target Entities icon with the Object set to Graph.



Toggles the form to select target display entities for graph plots.

The following table describes in detail to which entities graph plots can be targeted. The entity types and their entity attributes are of course dependent on the type of graph you want and what you have selected as the Y-axis or X-axis quantity. Some combinations may not make any sense. For instance a global variable plotted against another global variable does not need the specification of target entities. However, results quantities versus anything will need target entities specified.

Entity	Description
Nodes	Individual nodes may be selected from which to create a graph. Nodes are selected graphically from the screen and fill the databox. However, you may type in any node numbers manually. Be sure to include the word Node in front of the IDs you type in manually, (i.e., Node 1 5 55 100 etc.). Elemental based results are extrapolated to the nodes and averaged.
Elements	Individual elements may be selected from which to create a graph. Elements are selected graphically from the screen and fill the databox. However, you may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.). With elemental data, values will be extrapolated or averaged to the element centroid for reporting purposes.
Groups	Graphs can be limited to entities (nodes or elements) within a selected group. A selected group or groups must have elements or nodes in them otherwise the plot will not appear. A listbox allows selection of the group(s). Since groups can contain either nodes or element it is therefore also necessary to identify the entity attributes (nodes, element centroids, etc.) as explained in the following table.
Materials	As with groups, elements with certain material properties can be selected. As with elemental data, values will be extrapolated or averaged to the element centroid for reporting purposes.

Entity	Description
Properties	As with groups, elements with certain element properties can be selected. As with elemental data, values will be extrapolated or averaged to the element centroid for reporting purposes.
Path	A path can be defined in a variety of ways. The graph will be created by extrapolating results from the elements that touch the path to points that lie on that path. The path can be defined as a collection of points (geometric grids or nodes), curves, or edges of surfaces and elements. A select listbox will appear to allow you to type in individual beams/curves/etc. or select them graphically from the screen. When typing in individual beam element IDs make sure the word "Elem" is in front of all the IDs, (e.g., "Elem 1 3 5 10 20:40 100:150:10."). When a path is selected as the target entity and that path is defined by curves or beams/element edges, you need to also specify in an accompanying databox, the number of result locations to extract data from for the resulting plot. This is referred to as Points Per Segment in the Target Entities form.

In addition to targeting the above entities for a graph plot, the graph must be isolated to attributes of the entities as described in the following table. When nodes or elements are specifically targeted for the plot these choices are not available. However, for group or path target entities, the following choices are available.

Attribute	Description
Nodes	Specifies the use of the nodes of the target entities for extraction of the results value. Elemental based data are extrapolated to the nodes and averaged.
Element Centroids	Specifies that results be extracted at the element centroids for the plot. Nodal based data are summed and averaged at the centroid.
Points Per Segment	This is the number of points to be created for a graph using a Path as its Target Entity. The number of points generated in between selected nodes or curve end points is the number specified less one. A point will always be created at node points and end points of curves or element edges. The number of points can be one (1) to any positive number.
Points	When a path is the target entity, Points specifies that results be extracted at geometric points along that path. You can select these points graphically which can be made up of nodes, points, intersections, screen positions, etc. If the Number of Segments is one (1), a point on the graph will be created for each node. If two (2) is specified then an intermediate point will be created between each node. The number of points per segment less one will be created (interpolated) between points if greater than one (1). The path through the points will be piece wise linear (PWL). See notes below.

Attribute	Description
Curves	When a path is the target entity, specifies that path by the selection of geometric curves. These curves can be lines or edges of surfaces and elements. The select mechanism gives you control for defining all of these choices. If the Number of Segments is one (1), a point on the graph will be created for each end point of selected curves. If two (2) is specified then an intermediate point will be created between each end point. The number of points per segment less one will be created (interpolated) between end points and evaluated at equal arc lengths along each curve if greater than one (1). See notes below.
Edges/Beams	When a path is the target entity, specifies that path as beams or element edges. The number of points per segment less one (1) will be generated along each element edge. Midside nodes are not used by necessity. The points are generated at equal parametric locations along the edge using the order/geometry of the element to determine the locations. See notes below.

Notes on Path Target Entities:

1. The Point and Curve types have a dependency on the current group. This is important if you are working in small sections of a large model. The current group will limit the scope of the search for potential elements to contain the XYZ points generated. Thus selecting points that are not in the current group will result in no plot.
2. The Edges/Beam type is the most efficient interpolator. Since the elements are specified in the input, interpolation of results is a direct operation without any global searching for point mapping of global to parametric space.
3. When plotting versus Path Length, distances are calculated as straight line distances between interpolated points. If curves or edges are contiguous, the matching end points appear only once. If disjointed paths are specified, the distance value is reset to zero and the plot will zig-zag back to zero for the start of each disjointed section.

Important:	Once a target entity has been selected, it will remain the target entity for any graph until the user physically changes it.

8.3 Display Attributes

Graph plots can be displayed in various forms. Display attributes for graphs are accessible by pressing the Display Attributes icon on the Results application form with the Object set to Graph.

Toggles the form to change display attributes for graph plots.

This section describes the graph plot attributes which can be modified. Only a limited number of display attributes are actually available under the Results application for graph plots. Once a graph plot has been created, you may make attribute changes to the curves, axis and other entities in the XY Window with the XY Plot application (see **Overview of the XY Plot Application** (Ch. 1) in the *MSC.Patran Reference Manual*, *Part 7: XY Plotting*).

Attribute	Description
Curve Fit	Curve fit options are Linear, Scatter, Spline, and Least Squares. Linear is the default and will connect adjacent points with a linear line between the two. Scatter does not put any line on the plot at all but leaves just the points. Spline will connect the points with a curved and continuous line. This results in a smooth line with no abrupt changes is direction. Least Squares will do a least squares fit through all points to create one linear best fit line.
Curve Style	Curves can be solid, dotted, dashed, or dot-dashed.
Show Symbol	This toggle turns ON or OFF the display of symbols. The symbol used is the default symbol of the XY Plot application (a yellow round dot).
Show X/Y Axis Label	This toggle will display the X- or Y-axis label if ON.
X/Y Axis Label	This databox allows for specification of the X- and Y- axis label.
X/Y Axis Scale	The X- and Y-axis scales can be set to linear or log with this switch.
Label Style	This button brings up a subordinate form to change the label style of numerical text such as the numbers on the X- and Y-axes. The numerical format can be changed to integer, fixed, or exponential with specifications of color, size, and significant digits.

Attribute	Description
XY Window Name	A XY Window name must be supplied in order to create a graph. This will be the name of the separate graphical viewport for the graph that will appear on creation. Care should be taken when creating multiple graphs not to become confused. Even though you can save a graph plot with at specific name, (see Plot Options (p. 142)), multiple plots can reference the same XY window. Multiple plots will therefore appear in the same XY window even though they are actually separate Graph plot tools. The default XY window name is `Results Graph`.
Append Curves in XY WIndow	If this toggle is ON, then additional curves that may be created in the same XY Window and associated to the same Graph plot as opposed to being overwritten. For instance, you may put up a transient displacement plot of Node 101 and then decide it would be nice to overlay the same results from Node 345 in the same plot. However, care should be taken in that multiple graph plots can reference the same XY window and multiple curves in the same XY window of different data and magnitude can significantly change the axis scales and make plots unreadable.

Important:	Once plot attributes have been selected, they will remain in effect for any graph plot until the user physically changes them.

8.4 Plot Options

Graph plots have various options. Plot options for graphs are accessible by pressing the Plot Options button icon on the Results Display form.

Toggles the form to select plot options for graph plots.

The following table describes the graph options which can be modified:

Option	Description
Coordinate Transformations	Vector and tensor results for displaying graph plots can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran global system, a material coordinate system, element IJK coordinate system or the nodal (analysis) coordinate system depending on the type of result (vector, or tensor). See **Coordinate Systems** (p. 305) for a definition of each of these coordinate systems. The default is no transformation, which will extract data in the coordinate frame as stored in the database. Typically the solver code will calculate results at nodes in the analysis coordinate system specified by the user. These can vary from node to node. Element data can be stored from the analysis code in any coordinate system.
Scale Factor	This scale factor has the effect of simply scaling the results up or down by the specified amount. For results data this will scale both the x and y axes.
Filter Values	By specifying a filter value, a gate will be used to keep values below a maximum, above a minimum, between a certain range, or at the exclusion of certain values. The default is none. If filtering is used, only those results which pass the filter gate will be used in the graph plot. The filter values apply only to the Y-axis data.
Averaging Domain	For element based result quantities that must be extracted at nodes, an averaging domain must be used since more than one result will exist for each node. There is a contribution from each element attached to any particular node. By default all entities which contribute are used. Alternatively you can tell the Results application to only average results from those elements that share the same material or element property, are from the same target entities, or have the same element type. For more detail see **Averaging** (p. 292).
Averaging Method	The method in which certain results are determined can make a difference to the actual displayed plot. This is important when derived results from element based tensor or vector results are used such as VonMises stress or Magnitude displacements. For instance if you average at the nodes first and then derive the desired quantity you may get a different answer than if you derive first and then average. It is left up to the user to decide which is correct. For more detail see **Averaging** (p. 292).

Option	Description
Extrapolation Method	Many times element based results that are to be extracted from the nodes exist at locations other than the nodes such as at integration points. Various methods are available to the user to extrapolate these results out to the nodes. For mode detail see **Extrapolation** (p. 299).
Complex No. as	The Real component of a complex number is the default by which results will be postprocessed. To force the postprocessor to use a different quantity such as Magnitude, Imaginary, Phase, or Angle, set this option pull down menu. This option will only be available if a complex result has been selected. It is not recommended to calculate invariants (e.g., von Mises) from complex results because the phase is not accounted for.
Existing Graph Plots	This listbox displays all existing graph plots. You may select one of these plots from the listbox and all settings of that plot including display attributes, target entities, option, and selected results will be restored. This is an easy mechanism to help make many plots with the settings of an existing plot without modifying the selected plot. When the Action is set to Modify, this listbox appears under the Select Results display of the Results application form also.
Save Graph Plot As	Graph plots can be saved by name and recalled later for graphical display. Multiple graph plots can be saved in the database and displayed simultaneously. These graph plots can be posted / unposted and deleted as explained in **Post/Unpost** (p. 24) and **Delete** (p. 26) respectively. Once a plot has been created and named it retains all results, attributes, target entities, and options assigned to it. If no plot name is specified a default is created called `default_Graph`. As long as no plot name is specified, the `default_Graph` will be overwritten each time a plot is created or modified.

Important: Once plot options have been selected, they will remain in effect for any subsequent graph plot until the user physically changes them. Also for graph plots be aware that although the name is saved in the database and all attributes attached to it, the user can inadvertently delete or modify XY Windows, Curves, and other attributes associated with a graph plot in the XY Plot application. This is because each graph plot creates a XY Window and Curves which then become available in the XY Plot application. If you delete a XY Window in the XY Plot application you will affect any graph plot referencing that XY Window. See **Overview** (p. 134).

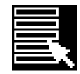
8.5 Examples of Usage

The following are some typical scenarios for usage of the Graph plot tool. These instruction assume that the Action is set to Create and the Object is set to Graph unless otherwise specified.

Graph of Results Versus Coordinate Distance (Nodal)

Displacement results are used in this example.

1. From the Select Results form (left most icon) select the Result Case from the first listbox. If more than one subcase exists for a Result Case, turn the Abbreviate Subcases toggle OFF and then select the Result Case.



2. The Y-axis should be set to Result values.

Y: Result

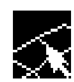
3. Select the Y-axis result quantity (Displacements) from the Select Y Result listbox.
4. (Optional) Set the result Quantity to Magnitude for displacement results.

Quantity: Magnitude

5. Set the X-axis to Coordinate and then either select the coordinate from the graphics screen or type in the coordinate value (Coord 0.1). Make sure you are selecting the correct coordinate axis. Use the select mechanism if necessary (Coord 0.1 indicates coordinate frame ID zero and the 1 or x direction.)

X: Coordinate

6. Use the Target Entities form to select the desired nodes. The nodes used in this example are indicated on the beam shown in **Figure 8-1**.



7. Press the Apply button.

Apply

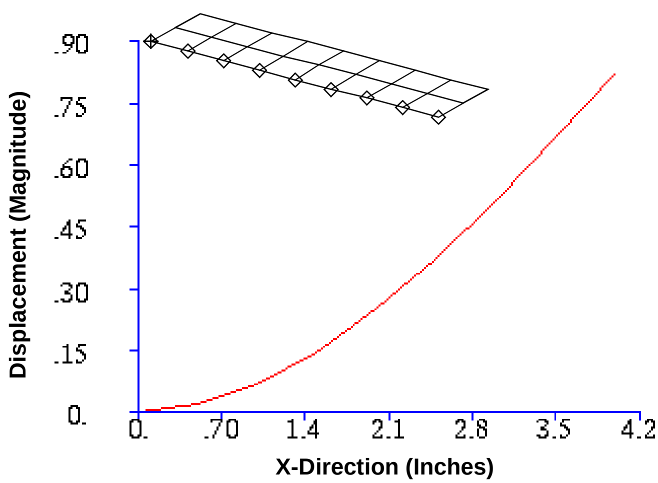
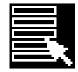


Figure 8-1 Result Value (Displacement) Versus Distance of the Cantilever Beam's Nodes as Shown.

Graph of Results Versus Results (Nodal)

Displacement versus stress results are used in this example.

1. From the Select Results form (left most icon) select the Result Case from the first listbox.



2. The Y-axis should be set to Result values.

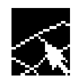
3. Select the Y-axis result quantity (displacement) from the Select Y Result listbox.
4. (Optional) Set the result Quantity if necessary (to Magnitude for this example).

5. (Optional) If more than one layer exists for these results, also select the layer you wish to plot.

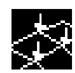
6. Set the X-axis to Result.

7. Similar to the Y-axis, you must select results for the X-axis. This is done in a subordinate form similar to selecting results for the Y-axis. Repeat steps **3.** through **5.** for the X-axis.

8. Use the Target Entities form to select nodes as target entities. Graphically select the nodes from the screen after setting the target entities to nodes.



9. (Optional) Use the Display Attributes form to make any changes to the display attributes of the plot.



10. Press the Apply button.

The order in which the results will be plotted will be the sequence that the nodes were selected in Target Entities.

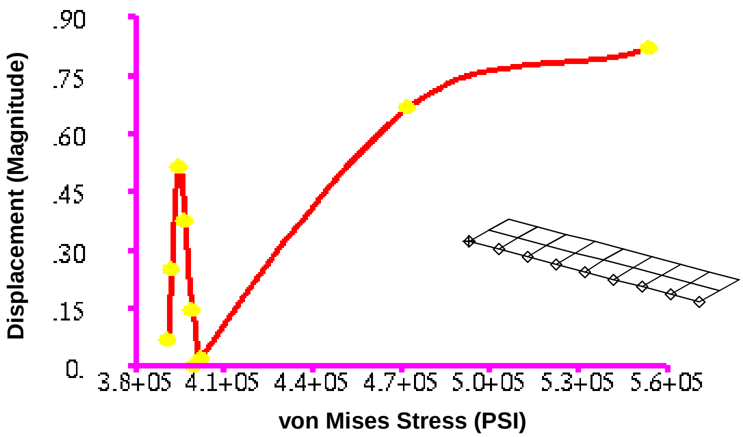
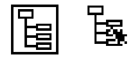


Figure 8-2 Result Value (Displacement) Versus Result Value (Stress) from the Cantilever Beam's Nodes as Shown.

Graph of Results Versus Global Variable (Transient Style)

Displacement versus time results are used in this example.

1. From the Select Results form (left most icon) select the Result Cases from the first listbox. You must select more than one Result Case. Use the Select button if necessary or turn OFF the Abbreviate Subcases toggle.

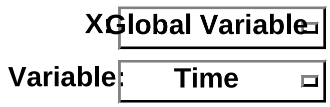


2. The Y-axis should be set to Result values.

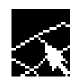
3. Select the Y-axis result quantity (displacement) from the Select Y Result listbox.
4. (Optional) Set the result Quantity if necessary (to Magnitude for this example).

5. (Optional) If more than one layer exists for these results, also select the layer you wish to plot.

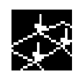
6. Set the X-axis to Global Variable (time). If more than one global variable exists, select the one you want to plot against.



7. Use the Target Entities form to select nodes as target entities. Graphically select the nodes from the screen after setting the target entities to nodes. You can pick as many nodes as you wish or as few as one. Too many will clutter the plot. One curve will result for each target entity selected for transient style plots.



8. (Optional) Use the Display Attributes form to make any changes to the display attributes of the plot.



9. Press the Apply button.

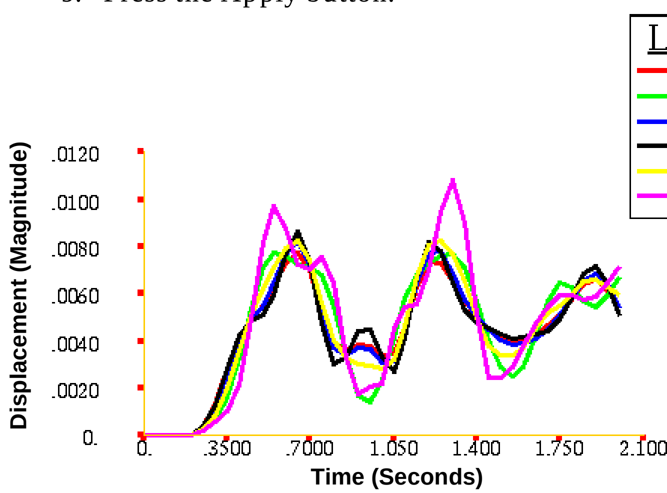
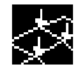


Figure 8-3 Result Value (Displacement) Versus Global Variable (Time) for Various Nodes of a Model (Transient Results).

Add Curve to Existing Graph

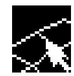
1. Follow any previous example to create a graph. For the sake of this example we will assume that another curve should be added to **Figure 8-3** at another node. So first follow the example **Graph of Results Versus Global Variable (Transient Style)** (p. 146).
2. Go to Display Attributes.



3. Turn ON the Append Curves toggle.

Append Curves in XY Win

4. Go to Target Entities and select a new node to create a transient curve from.



5. Press the Apply button. The new curve will be added to the existing XY Window retaining all previously created curves. All curves will be associated with the same Graph plot. In this case the default_Graph since we did not specifically give it a name.

Apply

Graph of Global Variable Versus Global Variable

This example uses design optimization results to plot design variables versus design iteration.

1. From the Select Results form (left most icon) select the Result Cases from the first listbox. You must select more than one or you will only get one data point for your graph.

2. The Y-axis value should be set to Global Variable.

Y Global Variable

3. Select the Y-axis global variable from the Variable pulldown menu.

Variable: Design Var 1

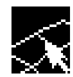
4. Set the X-axis to Global Variable.

X Global Variable

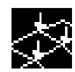
5. Select X-axis global variable from the Variable pulldown menu.

Variable Design Iteration

6. No Target Entities need to be selected for this type of plot.



7. (Optional) Use the Display Attributes form to make any changes to the display attributes of the plot.



8. Press the Apply button.

Apply

The plot below shows four curves. To create this the Append Curves in XY Window toggle was turned ON so as to create only one XY window with multiple curves. See the example **Add Curve to Existing Graph** (p. 147).

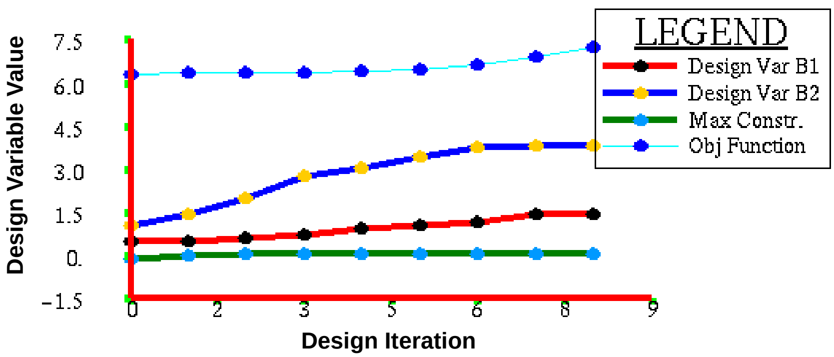


Figure 8-4 Four Plots of Global Variable (Design Variables) Versus Global Variable (Design Iteration).

Graph of Beam or Edge Results

The moment along the length of a beam (or element edges) was used for this example.

1. From the Select Results form (left most icon) select the Result Case from the first listbox.

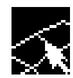
2. The Y-axis should be set to Result values.

3. Select the Y-axis result quantity (Bar or Shell Forces, Translational or Rotational) from the Select Y Result listbox.
4. (Optional) Set the result Quantity if necessary (to Y Component for this example).

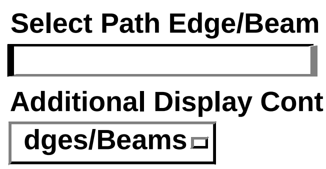
5. If more than one layer exists for these results, also select the layer you wish to plot.

6. Set the X-axis to Beams (It could also be set to Path Length).

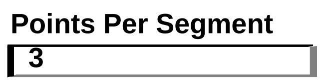
7. Go to Target Entities.



8. Select the beams or element edges as target entities. Graphically select them from the screen. The target entities should be set to Path. The entity display attributes should be Edges/Beams.



9. (Optional) Specify how many points (result locations) should be used per curve segment in the Points Per Segment databox. This will determine how many data points will make up the resulting graph.



10. (Optional) Use the Display Attributes form to make any changes to the display attributes of the plot.

11. Press the Apply button.

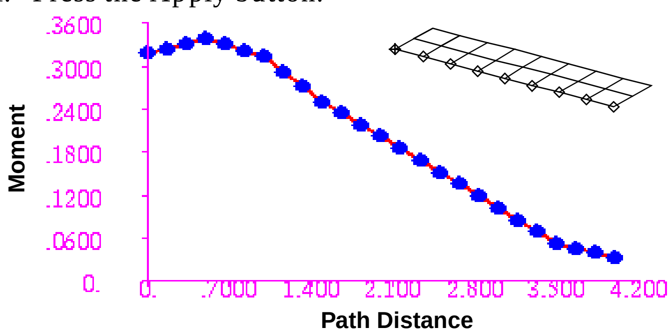


Figure 8-5 Result Value (Moment) Along a Set of Beams.

Graph of a Results Value Along an Arbitrary Path

An arbitrary path was defined by a curve created in the plane of the elements defining a cantilever beam for this example.

1. From the Select Results form (left most icon) select the Result Case from the first listbox.

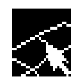
2. The Y-axis should be set to Result values.

3. Select the Y-axis result quantity (Displacements) from the Select Y Result listbox.
4. Set the result Quantity if necessary (to Magnitude for this example).

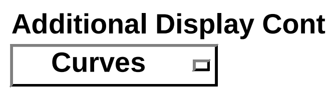
5. If more than one layer exists for these results, also select the layer you wish to plot.

6. Set the X-axis to Path Length.

7. Use the Target Entities form to select the curve or curves as target entities.



8. Graphically select them from the screen. The target entities should be set to Path. The entity display attributes should be Curves.



9. (Optional) Set the Points Per Segment up to a reasonable number to create points in between the end points of the curve(s). If set to one (1), only results at the end points of the curve will be plotted.

10. (Optional) Use the Display Attributes form to make any changes to the display attributes of the plot.

11. Press the Apply button.

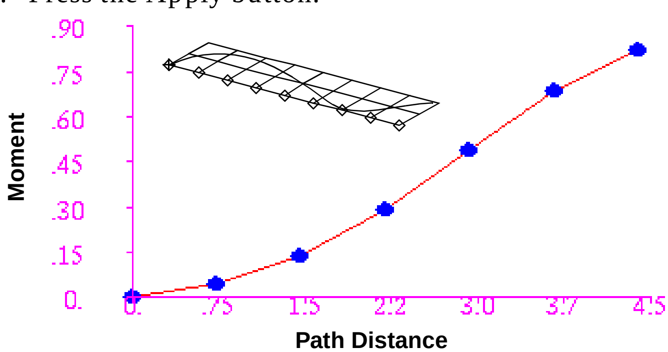
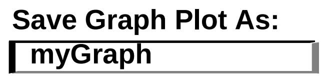


Figure 8-6 Result Value (Displacement) Along an Arbitrary Path (Defined by a Curve) as Shown on the Cantilever Beam.

Save a Graph Plot

1. Set up the graph plot as explained in the above examples but do not press the Apply button.
2. Before pressing the Apply button to create a plot or animation, press the Plot Options icon button (fourth button from the left).

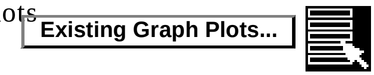
3. Type a name in the Save Graph Plot As databox.



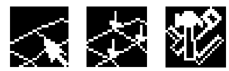
4. Then press the Apply button.

Modify a Plot

1. Set the Action to Modify with the Object set to Graph.
2. Select an existing graph plot from the Existing Graph Plots listbox.



3. Change results, target entities, display attributes, plot options, or other attributes as required.



4. Press the Apply button at any time to see the results of your modifications.

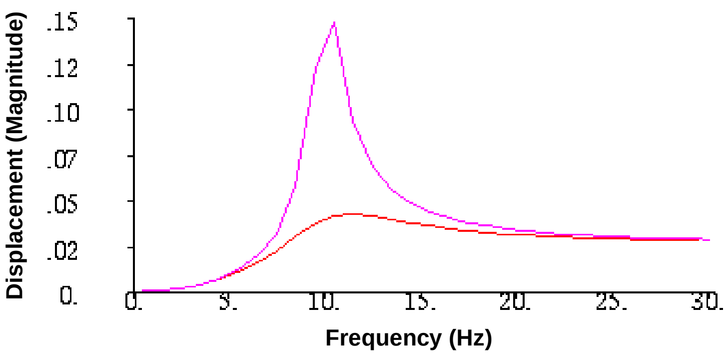


Figure 8-7 Result Value (Displacement) Versus Global Variable (Frequency) of Different Damping Values.

CHAPTER 9

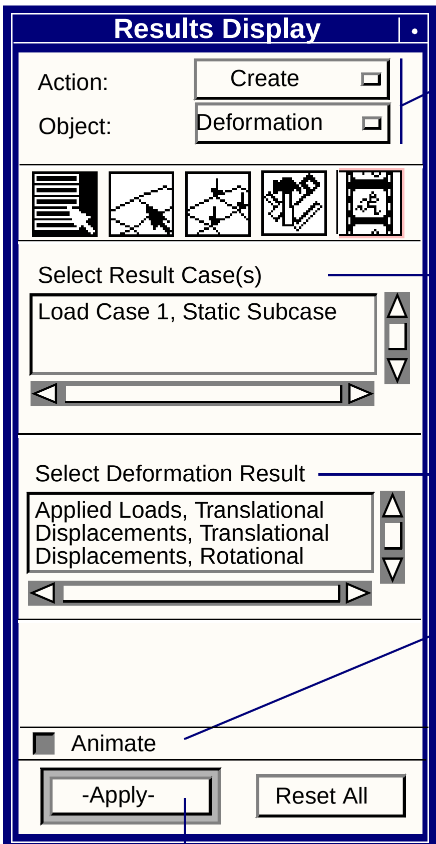
Animation

- Overview
- Animation Options
- Animation Control
- Animating Existing Plots
- Examples of Usage

9.1 Overview

Result animations are an integral part of understanding the behavior of a structure, especially those subjected to dynamic forces. Animations in MSC.Patran are set up in a variety of ways. Most plots, once created, can be animated. Or they can be animated at creation time. To successfully create an animation it is important to first understand how to create a plot. For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1).

To specifically create an animation of a given plot type at creation time, simply turn ON the Animate button with the Object set to the desired plot type. The general steps to take when creating an animation at plot creation time are:



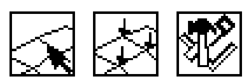
Step 1: Set the Action to Create and select an Object (the plot type) from the Results application form.

Step 2: Select a Result Case from the form for animation of static results. Select multiple Results Case if a transient animation is desired.

Step 3: Select a result type from the form.

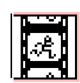
Step 4: Turn ON the Animate toggle.

Step 5: Change any target entities or other attributes and options if desired (optional).

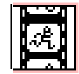


Step 6: Press the Apply button on the bottom of the form. Animation frames will be created and animation will begin.

Note: Animation options are set by pressing this button icon. Detail on animation options and animation control can be found in **Animation Options** (p. 156) and **Animation Control** (p. 158).



There are two aspects of animation: setting up the animation and controlling the animation once the animation frames have been created. For most basic animations, the default settings will be appropriate. However full control is given to the user to change animation options.



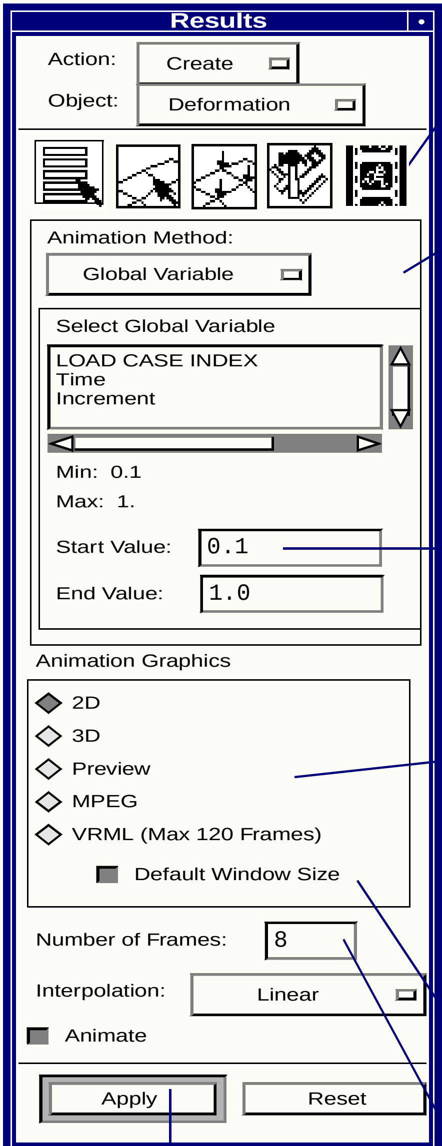
Animation options for any given plot type is controlled from the Animation Options form which is accessible by pressing the right most button icon on the Results Application form when the Object is set to a plot type. These options control things such as the number of frames to be generated, the animation method (modal, ramped, global variable), starting and ending values, 2D or 3D animation and interpolation methods. All settings are saved with a plot when it is created whether animation has been enabled or not. An explanation of these settings is given in **Animation Options** (p. 156).

Once an animation is in progress or the frames are being created, a form appears to control the animation. Control is given over such things as stopping and starting, pausing, stepping, speeding up and slowing down the animation. More detailed explanation of animation control is given in **Animation Control** (p. 158).

It is possible to save any result plots to the database with a specific name. In fact all plots are saved to the database with default names if none is given. Any plot that has already been created and saved to the database can be animated even though it was not animated when it was created. All attributes of that plot have been saved such as the result or results associated with it, its display properties, target display entities, and other options. Any animation attributes given to a plot is only active as long as the plot is posted. They are not saved in the database. Animation of existing plots is done by setting the object in the Results application to Animation and then selecting the plots to animate and those to remain static. This is explained in detail in the next section.

9.2 Animation Options

There are two aspects of animation: selecting animation options before animation actually begins, and controlling the animation once plots are animating. This is the most general display of the Animation Options form.



Animations are initially set up using the Animation Options form which is selectable for most plot types by selecting the right most icon on the Results application form. Intelligent default options have been set making it unnecessary to enter this area unless options need to be changed.

Several animation types are available. Modal animation allows animations from +MAX (positive maximum value) to -MAX (negative maximum value), whereas Ramped animation only ranges from ZERO to +MAX. These are the only options available for animating static results or results from a single results case (deformed plots, mode shapes, etc.). For transient data the Global Variable method is also available. You must select a global variable when animating transient data and you must have selected more than one subcase (time step) from a results case for this method to be available. See **Table 9-1** for more detail.

For transient animation or animation of multiple results cases you must select a global variable and specify a starting value and ending value. For transient data the global variable is generally time. When the number of frames to plot is not the same as the number of time steps then interpolation is used to create the missing frames. The interpolation method can be controlled at the bottom of this form.

2D animation allows for animation frames to be created only in a 2D plane. This simply means that dynamic rotation with the mouse is not possible without recreating all the animation frames again. 3D animation allows for dynamic rotation with the mouse. The advantage of 2D over 3D animation is speed, although this is highly hardware and model size dependent. Preview will step through each frame and quit. This is best used for transient animations. MPEG or VRML allows for output of animations to these standard formats, in addition to the display in the viewport. See **File>Images** (p. 197) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Controls the window size of image output when the MPEG or VMRL outputs are requested.Turning this toggle ON sets the output file window size to an acceptable size for most image view programs.

Press Apply to create the animation. Be sure, however that you have turned on the Animate button on the main form when selecting results. Otherwise no animation will be created. This form is for simply setting animation options.

Enter the number of frames for the animation to build. The default is 7. There is no current imposed limit to the number of frames that may be used. The more frames used, the smoother the animation will appear, however practical limits such as available memory and model size will quickly dictate the limit. If VRML output is requested, then the maximum number of frames allowed is 120.

Animation Interpolation. To create the specified number of frames for any animation, the program will do interpolation of the data because in most cases there will not be a one to one correspondence of frames to number of results cases. The default is to use Linear interpolation.

The interpolation method has no effect on the animation of static data (mode shapes, deformed plots). Each frame is determined using a simple harmonic or linear scale factor multiplied by the given static result.

For transient animation this is not the case. The interpolation method is used to determine how results are calculated for frames which do not have exact results present. During transient animation, the selected global variable is scaled linearly from its starting to ending value, evenly progressing between each animation frame. If the Load Case Index is chosen, then a dummy variable is assigned to each specified loadcase (1.0-n) and it is used as the interpolation basis (the independent variable). Once a global variable value is determined from each frame, the result values are then calculated based on the available results specified for the animation.

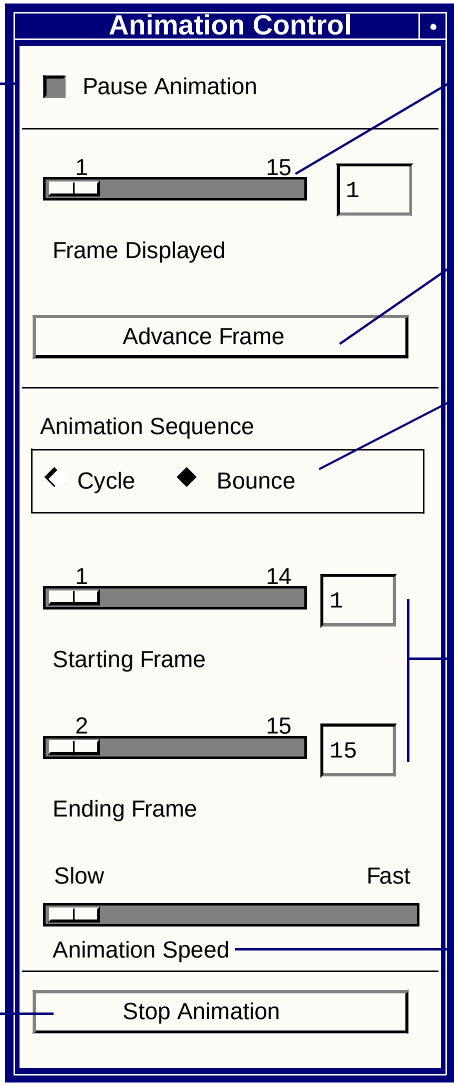
The following is a table of interpolation methods along with their descriptions:

Action	Description
Linear	This is the default interpolation function. The results for a given frame are determined using linear interpolation which performs a weighted average of the two closest results cases to the current frame.
Closest Value	No interpolation is performed. Only results found in the analysis are used. The closest results values associated to the current global variable are used for frame creation.
None	No interpolation is performed. Only results found in the analysis are used. The None interpolator will simply repeat the last usable data to fill the excess frames.

9.3 Animation Control

The Animation Control form automatically appears when the animation toggle has been turned ON after the Apply button has been pressed to create the animation.

Pause/Stop or Restart the animation currently running. The animation must be paused to enable any capability other than the start and end frame values.



Adjust the slidebar or enter a value for the current frame to display a paused animation. Adjusting the slidebar or entering a value in the databox will update the display accordingly.

If an animation is currently paused, click this button to step through the animation frame-by-frame using the current method.

Select a method for displaying the current animation. Cycle will display the animation frames 1:n, 1:n,... Bounce will display the frames 1:n:1:n... Bounce gives a much more continuous looking animation for the ramped method. For the modal method, Cycle and Bounce look much the same. Transient animations are more realistic in Cycle mode.

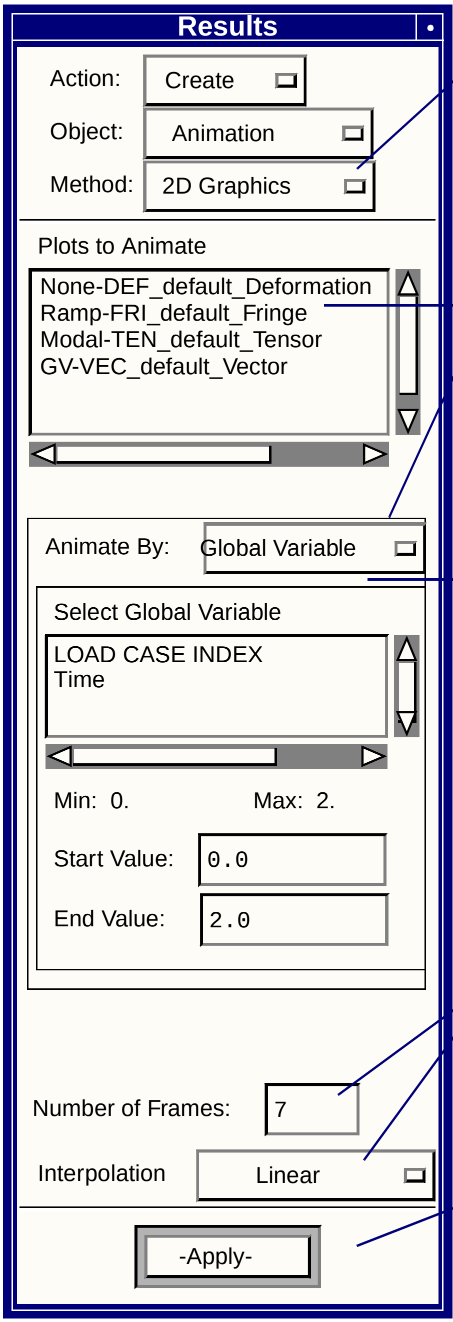
Adjust the slidebars to set a value, or enter a value in the appropriate databox, for the Starting Frame and Ending Frame of the animation currently running. Changing these values will skip the display of all the frames above or below the entered values, respectively.

Adjusts the speed at which the animation frames are played back.

Ends an animation and clears it from the screen. Other methods of stopping an animation are to press the Abort or Cleanup icons (the hand or broom respectively) on the main MSC.Patran form, close the database down or quit from MSC.Patran. The Animation Control form will close down once Stop Animation is pressed or the animation is stopped.

9.4 Animating Existing Plots

The following form and explanation show how to create animations of existing plots. All plots are saved in the database with their corresponding attributes, results, and other options associated with them. It is then a simple matter of turning ON or OFF the animation of these plots via this form. Set the Action to Create and the Object to Animation in the main Results application formIt is possible to record these animations using the new MPEG output (See **MPEG Images Output** (p. 199) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*). The File Images Form can be opened to allow recording of the MPEG file during animation playback.



The Method of animation can be in software (2D) or hardware (3D) mode. The main difference between these two are that hardware allows for dynamic rotation while animating in software does not. Additionally, a Preview method can be used to display each frame successively with no animation.

All posted plots appear in this listbox. One by one you will indicate which to animate and how. This is done by first selecting a plot. Its animation attributes will appear below. The animation attributes are those that were originally set up when creating the plot and can range from None to Ramped to Modal to Global Variable (GV). Note that the animation attribute is indicated next to the plot name for easy reference.

These animation attributes are explained in **Table 9-1**.

Specify the number of frames to be created for the animation. There is no limit, however memory and computer speed may quickly determine the practical limit. Interpolation methods only apply to transient animations when multiple results cases have been selected. They are explained in **Animation Interpolation** (p. 157).

Creates the animation of the specified plots. The animation control form will appear showing the progress of the creation of the animation frames. This form is explained in **Animation Control** (p. 158).

The following table describes in more detail the different animation attributes that can be set when setting up animation of several existing plots.

Table 9-1 Animation Attributes

Option	Description
None	If no animation has been initially set up when the plot was created, this is the animation attribute that it will have. None indicates that this plot will not animate. If you do not wish for a plot to animate, yet it is posted to the viewport, then this attribute must be set to None. The word *None-* will be placed next to the plot name to indicate that no animation will occur for the plot.
Modal	Setting a plot to Modal indicates that you want a modal style animation. This is common for modal analysis results. Modal style animations oscillate from the maximum results value to the negative of the maximum value, or in other words, it is like a fully reversed loading situation that oscillates like a sine function. When specifying a modal animation you can also set an angle offset. This sets up a phase shift in your animation. For example if you want to turn a sine wave oscillation into a cosine oscillation, use an offset angle of 90 degrees. The word *Modal-* will be placed next to the plot name to indicate that no animation will occur for the plot.
Ramped	A ramped style animation is good for animating statically loaded structures. The oscillation will proceed in a ramped form from a scale of zero to the maximum result value over the number of frames indicated. That is it will deform or animate from its position in rest to its fully deformed or stresses state (for deformation plots). The word *Ramp-* will be placed next to the plot name to indicate that no animation will occur for the plot.
Global Variable	This animation attribute can only be assigned to plots that have multiple Result Cases assigned to it in the case of transient animation. In this case you must indicate which global variable to use, and its start and end values. Animation frames will then be created from the results based on an interpolation scheme selected by the user. By default linear interpolation is used. This is the only animation attribute that uses interpolation. All others (Modal and Ramped) simply use linear scaling since only one Result Case is ever involved. See **Animation Interpolation** (p. 157) for more detail. The word *GV-* will be placed next to the plot name to indicate that no animation will occur for the plot.

9.5 Examples of Usage

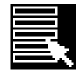
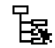
The following are the typical scenarios for creating animations when creating a new plot or from existing plots.

Create an Animation at Plot Creation Time (Static or Transient)

1. Decide what type of plot you wish to animate: deformation, fringe, vector, tensor, etc. Set the Action to Create and Object and / or Method accordingly.

   Object: Fringe

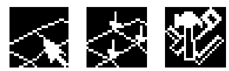
2. Select the Results Case(s) from the first listbox. Select one Results Case for an animation of a static result. Select multiple Result Cases for a transient style animation. You may wish to use the Select Subcases button icon to more easily filter multiple Result Cases.

3. Turn ON the Animate toggle. If an Animate toggle is not present on the main Select Results form for the desired Object, then that plot type does not support animation.

   Animate

4. (Optional) Set any other options such as target entities, display attributes, and plot options by changing the form and subsequent settings with the button icons at the top of the form just below the Action and Object menus.

5. Modal style animations are the default for static, single Result Case data. For transient style animations, press the right most button icon (Animation Options) to select a global variable (time, frequency, etc.). This is necessary for determining proper interpolation of results for each animation frame.

   Animate by: Global Variable

6. Press the Apply button.

   Apply

   If the Animate toggle was not turned ON, then no animation will result. Instead the Results application will simply create a static plot of the selected Result Case or a maximum plot of multiple Result Cases.

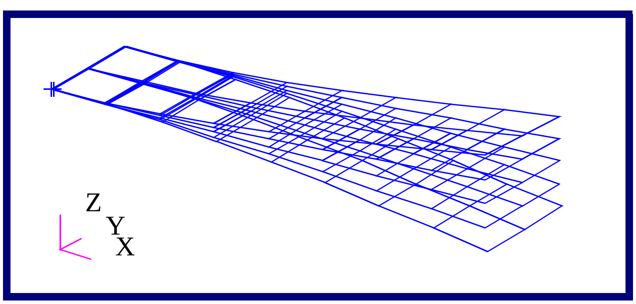
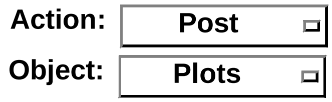


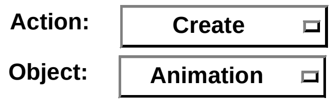
Figure 9-1 Modal Animation of Cantilever Plate

Create an Animation from Existing Plots

1. First make sure that the plots you wish to animate are posted to the current viewport. This is done by setting the Action to Post and the Object to Plots and selecting the desired plots from the listbox and pressing Apply.



2. Set the Action to Create and the Method to Animation.



3. If necessary, set the Method to software (2D) or hardware (3D) mode (hardware mode allows for dynamic rotation while animating in software does not). Speed performance may also differ between the two methods.

4. From the Plots to Animate listbox, select a plot that you wish to animate.
5. Change the Animate By option pulldown menu to the desired animation attribute which can either be None, Modal, Ramped, or by Global Variable. Global Variable is only selectable if the plot has been set up with multiple Result Cases for a transient style animation.

6. (Optional) You may need to change other options such as the Angle Offset for modal animations or global variable information for transient animations.
7. Repeat steps 4. to 6. for each plot you want to animate (to not animate, but still plot, set the attribute to None).
8. (Optional) Set any other animation options on this form as desired such as the number of frames to create and interpolation method for transient animations.

9. Press the Apply button.

   Any combinations of animation types can be combined together. An example of simultaneously animated plots is shown in Figure 9-2.

In general, with the exception of the number of frames or other animation options set on this form, all display attributes and other options associated with the plots to be animated will be retained.

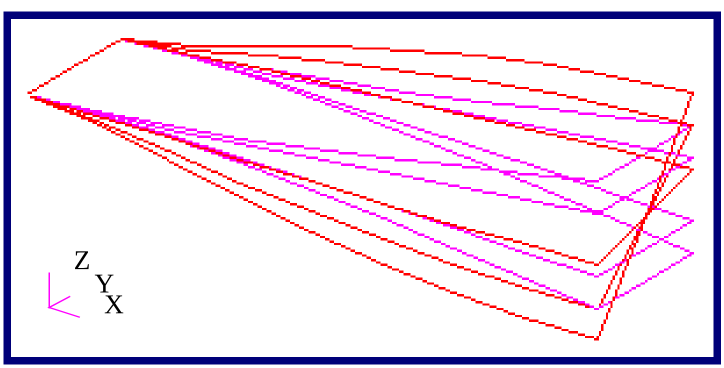


Figure 9-2 Simultaneous Animation of a Bending Mode (Magenta) and a Torsional Mode (Red).

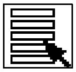
CHAPTER 10

Reports

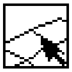
- Overview
- Target Entities
- Display Attributes
- Report Options
- Examples of Usage

10.1 Overview

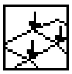
The Report object in the Results application allows for creation of text reports of any result that has been imported into the database or created by any other means such as derivations or combinations from other results quantities.



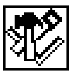
Selecting Results (p. 15)



Target Entities (p. 168).



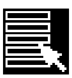
Display Attributes (p. 170) / **Report Format** (p. 170)



Report Options (p. 176).

The Report application works much the same way as any other plot type in the Results application with the only real difference being that a text report will be created as opposed to a graphical display. For an overview of how the Results application works see **Introduction to Results Postprocessing** (p. 1)

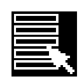
There is only a slight difference between Create and Modify. The main difference is that Create must be used to make a report and Modify is used to change an existing one. If you try to modify an existing report with Create you will be asked for overwrite permission whereas Modify assumes that the action is desired, so no overwrite permission is requested.



Toggles the form to select results for reports. This is the default mode of the Report form.

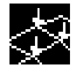
To create or modify a report, follow these general instructions:

1. Set the Action to Create, the Object to Report and the Method to the desired type. This depends on whether the report is to be displayed in the invoking window or written to a file. Preview will display the report in the invoking window, Overwrite File will create a new file or overwrite an existing one by the same name. Append File will append to an existing file.
2. Select a Result Case or multiple Result Cases from the Select Result Case(s) listbox. The results that appear in the listbox can be filtered. See **Filtering Results** (p. 18).



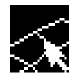
3. Select a result to report from the Select Report Result listbox.
4. If multiple layers exist for the selected result, select the layer positions that you wish to include in the report using the Select Positions listbox.
5. Specify the quantities associated with the selected result from the Select Quantities listbox. Some logical defaults are usually selected such as the components of a stress tensor or displacement vector. For explanations of derived results such as von Mises see **Derivations** (p. 286).

6. At this point you could press the Apply button to create the report. By default the report will be directed to the invoking window with the Method set to Preview. However, you may wish to direct the report to a file and / or change the format of the printed report using the Format button and its subordinate form. See **Display Attributes** (p. 170).



7. If directing the report to a file, make sure to set the Method to the appropriate selection, either Overwrite or Append mode. Specify a filename if not in Preview mode or accept the default. The filename can be changed under Display Attributes. Sorting options are also found there.

8. If you wish to report results for just a portion of the model, change the target entities for which the report is to be written by pressing the Target Entities button icon. See **Target Entities** (p. 168).



9. If other options are necessary to tailor the results report, such as coordinate transformation, averaging and extrapolation methods, sorting options, results filter values, or a scale factor then use the Options button icon. See **Report Options** (p. 176).

10. Press the Apply button to create the report.

Selected Quantities. The following table explains the different quantities that can be included in a report. For a more detailed explanation of derived quantities see **Derivations** (p. 286).

Quantity	Description
Scalar Value	When a scalar result has been selected for a report this is the generic name of the quantity. It is selected by default when a scalar value has been chosen for a report.
NSHAPE	This is a variable that is used with elemental data and outputs an integer value indicating the type of element the result is associated with (Bar=2, Tri=3, Quad=4, Tet=5, Pyr=6, Wed=7, Hex=8). This variable is generally only used when outputting a scalar elemental MSC.Patran results file. See **Create a MSC.Patran .els Formatted File** (p. 184).
Loadcase ID	This is the internal Load Case or Result Case ID that is associated with the result quantity. Generally this information is output with report summaries.
Subcase ID	This is the internal subcase ID associated with the result quantity. Generally this information is output with report summaries.
Layer ID	This is the internal layer ID associated with the result quantity. Generally this information is output with report summaries.
X/Y/Z Location	The X, Y, or Z coordinate location in the MSC.Patran global coordinate system.
CID	The coordinate system ID that the reported results are in.

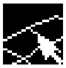
Quantity	Description
Material ID	The internal material ID of the region of interest (zero if none exists).
Material Name	The material name currently assigned to the region of interest. If none exists then “ErrErr” will be reported.
Property ID	The internal property ID of the region of interest (zero if none exists).
Property Name	The property name currently assigned to the region of interest. If none exists then “ErrErr” will be reported.
ACID	The analysis coordinate system attached to the entity ID (node) which is recovered from the entity record in the database. Results from the analysis code are generated in the ACID system.
Magnitude	The magnitude derived from a vector quantity.
X/Y/Z XY/YZ/ZX Component	The X, Y, or Z components of a vector quantity or the X, Y, Z, XY, YZ, or ZX components of a tensor quantity.
von Mises	von Mises stress derived from a stress tensor.
Max/Mid/Min Principal	The maximum, intermediate or minimum principals derived from a tensor.
Hydrostatic	The Hydrostatic stress derived from a stress tensor.
1st, 2nd, 3rd Invariants	The 1st, 2nd, and 3rd invariant stresses derived from a stress tensor.
Tresca	Tresca stress derived from a stress tensor.
Max Shear	Maximum shear stress derived from a stress tensor.
Octahedral	Octahedral stress derived from a stress tensor.

The following quantities are also reported dependent on the type of result selected and target entities.

Quantity	Description
Entity ID	This is the ID of the Node or Element being reported. This Entity ID will always appear. The only control you have over this, is in which column to display. See **Report Format** (p. 170).
Node ID	When element nodal data is being reported this is a node number connected to the element (Entity ID) for which results are being reported. For element nodal data, these node IDs appear automatically so you may distinguish which element result row belongs to which node. Only the column in which the node ID is displayed can be changed. See **Report Format** (p. 170).
Position ID	When element Gaussian data is being reported this is a position ID of the element (Entity ID) for which results are being reported. For element Gaussian data, these position IDs appear automatically so you may distinguish which element result row belongs to which position. Only the column in which the position ID is displayed can be changed. See **Report Format** (p. 170). Element position number are internal IDs which are generally meaningless to users. However -999 signifies centroidal data, and 0, -1, -2, ... signifies internal node (coincident not the nodes) locations. Actual Gauss points will have their own internal IDs.

10.2 Target Entities

Reports can be created for various model entities. By default reports are created for everything displayed in the current viewport. To change target entity selection for results reporting, press the Target Entities icon with the Object set to Report.



Toggles the form to select target entities for report generation.

The following is a description of all target entities and target entity attributes.

Entity	Description
Current Viewport	By default reports are created for all finite element entities displayed in the currently active viewport.
Nodes	Individual nodes may be selected for which to create a report. You may type in any node numbers manually or by selecting them graphically from the screen. Be sure to include the word Node in front of the IDs you type in manually, (i.e., Node 1 5 55 100 etc.). To select all nodes use the syntax “Node 1:#.”
Elements	Individual elements may be selected for which to create a report. You may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.). To select all elements use the syntax “Elem 1:#.”
Groups	Reports can be limited to only selected groups. A listbox will appear allowing selection of the groups for generating a report. Only for entities belonging to those groups selected will a report be generated.
Materials	Reports can be targeted at only those finite elements which have certain material properties assigned to them. A listbox will appear allowing selection of the materials for whose entities will be reported
Properties	Reports can be targeted at only those finite elements which have certain element properties assigned to them. A listbox will appear allowing selection of the properties for whose entities will be reported.
Element Types	Reports can be limited to only certain element types also.

In addition to selecting the target entities from which to generate a report, it is also necessary in some cases to specify the entity attributes to further define what data is to be reported. For target entities in the Current Viewport, Elements, Groups, Materials, Properties, and Element Types it is necessary to specify where to report the values.

Attribute	Description
Nodes	Results data will be reported at the nodes only. This is most appropriate for nodal type data. For elemental type data, the results will be extrapolated to and averaged at the nodes for reporting single values at each node of the target entities.
Element Centroids	Results data will be reported at the element centroids. This is most appropriate for element centroidal type data. For nodal data, the values will be averaged from each contributing node and reported as a single value at the center of each element of the target entities. The same is true for elemental data with more than one value associated with each element (element nodal data or element Gauss data).
Element Nodes	Results data will be reported at the nodes for each element in the target entities. This is most appropriate for element nodal data where results for every element exist at the nodes. This is not appropriate for nodal type data at all. For element centroidal data and other element data (such as at gauss points) the results will be extrapolated to the nodes. Results are therefore reported by element followed by associated nodal results for the given element.
Element All Data	Results data will be reported as is. This is most appropriate for element results at gauss points. The data will be left as is. This is true also for element nodal and element centroidal results but no indication of node numbers will be given, simply element positions as stored internally in the database. This is inappropriate for nodal type data.

Important:	Once a target entity has been selected, it will remain the target entity for the report until the user physically changes it.

10.3 Display Attributes

Result reports can be formatted in a variety of ways and with many options.

Toggles the form to set display attributes and formatting options of reports.

Below are descriptions for all the fields and settings for formatting text reports.

Item	Description
File/File Name	Specify a file name to direct the report to. You can only specify a file name if the Method has been set to Overwrite or Append File. A file browser is also available if you press the File button to select an existing file. The default filename is `patran.prt`. This can be overridden with a `settings.pcl` parameter: `pref_env_set_string(“result_capture_filename”,“patran.prt”)` See **The settings.pcl file** (p. 41) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.
Format...	A subordinate form will appear to allow for report formatting. This is explained below in **Report Format** (p. 170).
Sorting Options	A subordinate form will appear to allow for selection of sorting options. This is described below in **Sorting Options** (p. 175).
Report Type	A full report includes summary and data. Only summary information such as load case and max/min values can be requested as well as data only with no summary information.

Report Format. Below are descriptions for all the fields and settings for formatting text reports.

Table 10-1 Results File Format Options

Item	Description
File Width	Sets the number of characters that can fit in the width of a page including spaces. The default is 128 characters.
Lines/Page	Sets the number of lines per page. The default is 52 lines per page.
Top Margin	Sets the number of lines used to form a top margin. The bottom margin is set by the number of Lines/Page.
Left Margin	Sets the number of characters used to make a left margin. The right margin is set by the File Width.
Pagination	If you wish to use pagination turn this toggle ON. The Page Number setting will appear to set the beginning page number. No footer or header information will be printed.
Page Number	Set the beginning page number with this option. This databox only appears if Pagination has been turned ON.

Table 10-1 Results File Format Options (continued)

Item	Description
Edit	This is an option menu for editing the Title, Footer, or Header. No Footer or Header is allowed if pagination is OFF. This text box below this menu will update to allow for editing of the selected text.
Alignment	Alignment of the report can be from the left margin, right margin or the report can be centered.
Title/Header/Footer Text Format	This textbox allows for inclusion and modification of a Title, Header or Footer. Which is set for editing is determined by the Edit option menu above this text box. You may place a %I% in any of these text boxes to include the page number if Pagination has been turned ON. You may also include a %rN% for including additional lines after. These formatting characters are explained in .
Display Column Labels	This toggle will turn on or off printing of the Column labels. The column labels are the middle column of the spreadsheet shown on the Results File Format form.
Input Column/ Label/Format	This is a databox that becomes active to allow for changes in the actual Column numbers, Column Labels, or Value Formats. Simply click on a cell in the spreadsheet that appears below this databox and the databox will become active to allow you to change the cells contents. If you wish to reorder the columns, change the column numbers using this mechanism and then press the Order Columns button to reorder their appearance in the actual spreadsheet. If no results have been selected before the Format form is opened, the spreadsheet will not appear below this databox.
Column	This is the column number with its associated label and value format. If you wish to change this to a different column, simply click on this cell and enter the column number where you wish this label and its values to appear using the databox above. Press the Enter or Return key to effect the change.
Column Label	This is the label that appears above the column of results. By default it is the same as results quantities selected. To change a label, select the cell and then change the value in the databox above. Press the Enter or Return key to effect the change.
Value Format	Results formats are listed in this column. They specify how the actual results values will be formatted in the report. They consist of the format characters surrounded by percentage signs. To change one of these formats, click on the cell that contains the format to change and enter your changes in the databox above the spreadsheet. Press the Enter of Return key to effect the change. The different characters and combinations acceptable for these formats is explained in .
Order Columns	If you wish to reorder the columns once you have manually changed the column numbers in the spreadsheet then press this button.

Important:	The order in which the quantities are arranged in the spreadsheet of the Format form is dependent on the order in which you selected them from the Select Quantities listbox. Select them from the list box in the order in which you wish them to appear in the report. Use the Control key for discontinuous selections.

Format Strings. gives a description of output format strings used to convert integer, real, and string data to a formatted output. It is necessary to use these strings in the Value Format column in the spreadsheet to specify how to format the results values in the report. Some of these formats can also be used in the Title, Header and Footer. The format string is a simple character string which contains both raw text to output, and format specifiers, enclosed by a set of percent characters, which control how data items are formatted and output. Upper case letters (I, F, E, etc.) are interpreted literally and lower case letters are to be substituted with the appropriate values.

To change a value format simply click the mouse button with the cursor in the cell whose format you wish to change. Then in the Input databox above the spreadsheet change the value format to what you want and then press the Return or Enter key.

Table 10-2 Value Format Strings for Formatting Text Report Numbers

Format	Description
%%	The simplest form of format specifier is a double percent to produce a single percent in the final output. Used if you want a percent character in the Title, Header or Footer.
%Im%	Integer (I) specifier. This format specifier takes an integer value such as a node or element (entity) ID or other integer result for formatting. The value of “m” is the minimum number of characters to produce from the format. If “m” is omitted, then the exact number of characters necessary to hold the integer is used. The exact format produced is an optional minus sign followed by one or more digits. The default for integer data is %I6%.
%Fm.n%	Fixed (F) float specifier. This format specifier takes a real results value for formatting in fixed point notation. The value of m is the minimum number of characters to produce from the format. If m is omitted, then the exact number of characters necessary to hold the conversion is used. The value of n is the number of decimal digits to produce. If omitted, then all significant digits will be produced. The exact format produced is an optional minus sign followed by zero or more digits, a decimal point, and zero or more digits. At least one digit will precede or follow the decimal point. The default for real data is %F12.6%.

Table 10-2 Value Format Strings for Formatting Text Report Numbers (continued)

Format	Description
%Em.n.p%	Exponential (E) float specifier. This format specifier takes a real value for formatting in scientific exponential notation. The value of m is the minimum number of characters to produce from the format. If m is omitted, then the exact number of characters necessary to hold the conversion is used. The value of n is the number of decimal digits to produce. If omitted, then all significant digits will be produced. The value of p is the number of digits to display before the decimal point, and defaults to one. If zero is specified, then a single zero precedes the decimal point. The exact format produced is an optional minus sign followed by zero or more digits, a decimal point, zero or more digits, a capital E, a plus or minus sign, and two decimal digits. At least one digit will precede or follow the decimal point. The default value for read data is the F format.
%Gm.n.p%	General (G) float specifier. This format specifier takes a real value for formatting in either F or E format. The format used depends on the value of the number to convert. In general, if the exponent is small, the F format will be used, otherwise the E format is used. See the descriptions of the F and E formats.
%Sm%	String (S) specifier. This format specifier takes the next string value from the character data array for formatting. The value of m is the minimum number of characters to produce from the format. If m is omitted, then the exact number of characters in the string is used. The default value for string data is %S32%.
%rN%	New (N) line specifier. This format specifier causes a new line to be started. The previous line is output as is, and formatting starts at column one of the new line. The value of r is a repeat count for skipping multiple lines. If output is to a string, then new line characters will be written to the string. This is used in the Title, Header and Footer text.

Variables. Variables can be placed in titles, footers, or headers of reports. The variables available are shown in the table below. Be sure to place the $ symbol in front of the variable otherwise it will not be recognized as a variable.

Table 10-3 Value Format Strings for Formatting Text Report Numbers

Format	Description
$LC_NAME	This is the Result Case (load case) name.
$SC_NAME	This is the subcase name.
$PRES_NAME	This is the primary result name.
$SRES_NAME	This is the secondary result name.
$LYR_NAME	This is the result layer name.
$DATE	The current date and time in the format dd-mmm-yy hh:mm:ss.

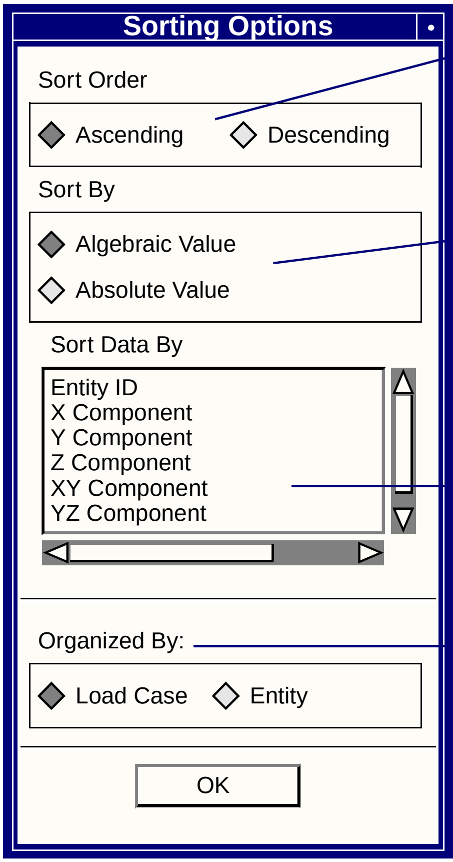
Table 10-3 Value Format Strings for Formatting Text Report Numbers (continued)

Format	Description
$PAGE	The current report page number.
$NNODES	The number of nodes in the report. Variable is printed in I9 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$MAXNOD	The highest ID of a node in the file. Variable is printed in I9 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$DEFMAX	The maximum value encountered within the file. Variable is printed in E15.6 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$NDMAX	The ID of the node with the maximum value. Variable is printed in I9 format if left aligned. Valid for nodal report only, sorted by Result Case. Typically used to create MSC.Patran nodal (nod) result files.
$NWIDTH	The number of columns in the file. This will be the number of results quantities output to the report. Note that the Entity Id which is the first column of most reports by default is not included in NWIDTH. It is actually the number of columns of real, floating point data. Typically this is used to create MSC.Patran nodal (nod) and elemental (els) result files.
$DATA_TITLE	The register title. You must use the built in function res_data_title() to set a title for your register. Once this title is set, then it will show up when you use $DATA_TITLE. See the **Data Register Definition Functions** (p. 1139) in the *PCL Reference Manual, Volume 1: Function Descriptions*.
$PRODUCT	The MSC.Patran product/version.
$DB_NAME	The name of the current database.
$JOB_NAME	The name of the analysis job.
$CODE_NAME	The name of the analysis code as set under Preferences/Analysis.
$GV:<name>	The name and value of an associated global variable such as time, frequency, eigenvector, etc. If a global variable is one word then all that is needed is to specify that global variable after the colon, i.e., $GV:Time. However, if a global variable name has a space in it or, that is, consists of more than one word, you must surround the name with single quotes, i.e., $GV:'Design Cycle.' Failing to do this will results in the variable picking up only the first word and will not find the correct global variable and will report garbage. Using this variable in the header and footer when multiple results cases (multiple GVs) will only use the first global variable encountered.
$LEFT	Aligns the current line of text to the left, overriding the global page alignment.

Table 10-3 Value Format Strings for Formatting Text Report Numbers (continued)

Format	Description
$MIDDLE	Aligns the current line of text to the middle, overriding the global page alignment.
$RIGHT	Aligns the current line of text to the right, overriding the global page alignment.

Sorting Options. Results can be sorted in a report and sorting is controlled via this form which is available from the Report Options form by pressing the Sorting Options button.



Results can be sorted in Ascending (from smallest to largest) or Descending order. Ascending order is the default.

Sorting can be done by comparing Algebraic Values which considers the sign of a value. A negative value will be treated as less than a positive number. Or the Absolute Value of the results can be used where sign is ignored, and the relative size in magnitude is considered in the sort.

This listbox displays the result entity by which the sort is based. By default sorting is always done and is based on the Entity ID. This would be a node or element number in general.

The report can be organized by either the Results Case (Load Case) or by Entity. Load Case organization will report all results quantities for every entity for each load case. This is generally the way static results are reported and is the default. However, for transient type data, it is sometimes easier to view a report in terms of the entities where results for a given Node ID are reported for every time step (load case). Multiple Result Cases must have been selected for this type of organization to be presented meaningfully.

10.4 Report Options

Reports have various options. Report options are accessible by pressing the Report Options selection button.

Toggles the form to select report options.

The following table describes the report plot options which can be modified:

Option	Description
Coordinate Transformations	Vector and tensor results can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran global system, a material coordinate system, element IJK coordinate system or the nodal (analysis) coordinate system depending on the type of result (vector, or tensor). See **Coordinate Systems** (p. 305) for a definition of each of these coordinate systems. The default is no transformation, which will plot data in the coordinate frame as stored in the database. Typically the solver code will calculate results at nodes in the analysis coordinate system specified by the user. These can vary from node to node. Element data can be stored from the analysis code in any coordinate system. Note also that the analysis translators that import the results data into the can transform results. Check with the appropriate translator guide.
Scale Factor	This scale factor has the effect of simply scaling the results up or down by the specified amount.
Filter Values	By specifying a filter value, a gate will be used to keep values below a maximum, above a minimum, between a certain range, or at the exclusion of certain values. The default is none. If filtering is used, only those elements which pass the filter gate will be reported.
Averaging Domain	For element based result quantities that must be displayed at nodes, an averaging domain must be used since more than one result will exist for each node. There is a contribution from each element attached to any particular node. By default all entities which contribute are used. Alternatively you can tell the Results application to only average results from those elements that share the same material or element property, are from the same target entities, or have the same element type. For more detail see **Averaging** (p. 292).
Averaging Method	The method in which certain results are determined can make a difference to the actual displayed plot. This is important when derived results from element based tensor or vector results are used such as VonMises stress or Magnitude displacements, For instance if you average at the nodes first and then derive the desired quantity you may get a different answer than if you derive first and then average. It is left up to the user to decide which is correct. For more detail see **Averaging** (p. 292).

Option	Description
Extrapolation Method	Many times element based results that are to be displayed at nodes exist at locations other than the nodes such as at integration points. Various methods are available to the user to extrapolate these results out to the nodes. For more detail see **Extrapolation** (p. 299).
Complex No. as	The Real component of a complex number is the default by which results will be reported. To force the report to use a different quantity such as Magnitude, Imaginary, Phase, or Angle, set this option pull down menu. This option will only be available if a complex result has been selected. It is not recommended to calculate invariants (e.g., von Mises) from complex results because the phase is not accounted for.
Existing Reports	This listbox displays all existing reports. You may select one of these from the listbox and all settings of that report including format attributes, target entities, option, and selected results will be restored. This is an easy mechanism to help make many reports with the settings of an existing report without modifying the selected report much. When the Action is set to Modify, this listbox appears under the Select Results display of the Results application form.
Save Report As:	Reports can be saved by name and recalled later. These reports can be deleted as explained in **Delete** (p. 26) respectively. Once a report has been created and named it retains all results, attributes, target entities, and options assigned to it. If no name is specified a default is created called `default_Report`. As long as no name is specified, the default name will be overwritten each time a report is created.

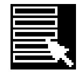
Important: Once report options have been selected, they will remain in effect for the report until the user physically changes them.

10.5 Examples of Usage

The following are some typical scenarios for usage of results reporting. These instructions assume that the Action is set to Create and the Object is set to Report and that results exist in the database unless otherwise specified.

Report of Displacement Data

1. Set the Method to Overwrite file. By default, a new file will be created called `patran.prt.1`.



2. Select a Result Case from the first listbox.

3. Select a translational displacement result from the second listbox.
4. (Optional) Select X Component, Y Component, and Z Component from the Select Quantities listbox. These may already be selected by default.
5. Press the Apply button accepting all format defaults and other option defaults.

The report should look somewhat similar to this:

```
                    MSC.Patran Version x.x 10/01/97 10:18:06 AM
                             Analysis Code: MSC.Nastran
                          Load Case: Load Case 1, Time=0.55
             Result: Displacements, Translational- Layer (NON-LAYERED)
                                Entity: Node Vector
   -Entity ID---X Component---Y Component---Z Component--
           1        0.000251      0.000483      0.005989
           2        0.000258      0.000488      0.006124
           .
        1467        0.000251      0.000455      0.006782
        1468        0.000280      0.000501      0.006957
                                SUMMARY INFORMATION
                             --------------------------
                                  Min/Max Values
    -Source ID--Entity ID--X Component--
   Min:     1        257         -0.001948
   Max:     1         71          0.000882
    -Source ID--Entity ID--Y Component--
   Min:     1        257         -0.002525
   Max:     1         55          0.001091
    -Source ID--Entity ID--Z Component--
   Min:     1        540          0.001625
   Max:     1        257          0.009460
```

Figure 10-1 Report of Displacement Data with Summary

Report of Transient Displacement Data

1. Set the Method to Overwrite file. By default, a new file will be created called `patran.pr1` where `database_name` is the name of the database. Change the name if you wish.

2. Select the Results Cases (time steps) from the first listbox.

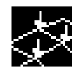
   Method: Overwrite File

3. Select a translational displacement result from the second listbox.

4. Select X Component, Y Component, and Z Component from the Select Quantities listbox. These may already be selected by default.

   You could press the Apply button at this point and create a report similar to the previous example but repeated for each time step. However, it would be better to report the displacements for each time step together grouped together by entity ID.

5. Press the Display Attributes button icon (right most icon).

6. Open the Sorting Options form.

   Sorting Options

7. In the Sorting Options form change the Organized By switch from Load Case to Entity.

   Organized By:
   ◇ Load Case ◆ Entity

8. Press the Apply button accepting all format defaults and other option defaults.

   Apply

The report should look somewhat similar to **Figure 10-2**. Each line corresponds to the next time step for each entity ID. The title contains the names of each Result Case which is given a source ID. From the source ID you can determine from the result data which source it is from.

```
                    MSC.Patran Version x.x  05/13/97 11:46:58 AM
                          Load Case: Load Case 1, Time=1.9
              Result: Displacements, Translational- Layer (NON-LAYERED)
                                Entity: Node Vector
                                (First of 5 Sources)
                                   Result Sources
-Source Id---Loadcase Name---------Subcase Name---------Layer Name----
      1     Load Case 1          Time=1.8             (NON-LAYERED)
      2     Load Case 1          Time=1.85            (NON-LAYERED)
      3     Load Case 1          Time=1.9             (NON-LAYERED)
      4     Load Case 1          Time=1.95            (NON-LAYERED)
      5     Load Case 1          Time=2.              (NON-LAYERED)
-Source ID--Entity ID--X Component---Y Component---Z Component--
   1        1        0.000042      0.000774      0.005799
   2        1        0.000774      0.000774      0.000774
   3        1        0.005799      0.005799      0.005799
   4        1       -0.000011      0.000775      0.005889
   5        2        0.000775      0.000775      0.000775
   .
   1      320       -0.000077      0.000780      0.006008
   2      320        0.000780      0.000780      0.000780
   3      320        0.006008      0.006008      0.006008
   4      320        0.000037      0.001000      0.007320
   5      320        0.000595      0.000028      0.005629
   .
  Continues for every Entity ID
                                SUMMARY INFORMATION
                             _________________________
                                   Min/Max Values
    -Source ID--Entity ID--X Component--
Min:     1        556        -0.001338
Max:     5        556         0.000595
    -Source ID--Entity ID--Y Component--
Min:     5        540        -0.002156
Max:     2        540         0.003511
    -Source ID--Entity ID--Z Component--
Min:     5        540         0.001625
Max:     2        540         0.010750
```

Figure 10-2 Report of Transient Displacement Data.

Create a MSC.Patran .nod Formatted File. A MSC.Patran .nod file is a nodal based ASCII results file. The format of this file is:

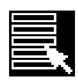
```
Record 1:          TITLE                              (80A1)
Record 2:          NNODES,MAXNOD,DEFMAX,NDMAX,NWIDTH  (2I9 E15.6, 2I9)
Record 3:          SUBTITLE1                          (80A1)
Record 4:          SUBTITLE2                          (80A1)
Record 5 to n+4:   NODID, (DATA(J), J=1, NWIDTH)      (I8, (5E13.7))
```

where:

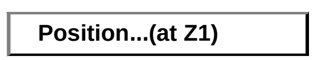
`TITLE`	80A1 title stored in an 80 word real or integer array.
`SUBTITLE1`	Same format as TITLE.
`SUBTITLE2`	Same format as TITLE.
`NNODES`	Number of nodes (integer).
`MAXNOD`	Highest node ID number (integer).
`DEFMAX`	Maximum absolute displacement (real).
`NDMAX`	ID of node where maximum displacement occurs (integer).
`NWIDTH`	Number of columns after NODID for nodal information (integer).
`NODID`	Node ID number (integer).
`DATA`	Result quantities organized by column index (real).

1. Set the Method to Overwrite File. By default, a new file will be created called `patran.prt.1`. The name will be changed later.

2. In Select Results mode, select the Result Case from the first listbox.



3. Select a result from the second listbox.
4. (Optional) If necessary, select a layer. For this example, only one layer can be selected for a proper `.nod` file to be created.



5. Select the Quantities to output to the file. Only real (floating point data) quantities are valid for proper `.nod` files.
6. Go to Display Attributes.

7. Change the file name to `patran.nod`.
8. Press the Format... button.

9. Set the File Width to 80.

10. Clear the Lines / Page, Top Margin, and Left Margin databox or put zeros.

Lines/Page:

11. Turn Pagination OFF.

☐ Paginatio

12. Make the Title Alignment Left, clear the Title textbox and enter these four separate lines for the title:

Alignment: Left

```
TITLE
$NNODES$MAXNOD$DEFMAX$NDMAX$NWIDTH
SUBTITLE1
SUBTITLE2
```

Of course you can substitute anything you want for TITLE, SUBTITLE1 and SUBTITLE2.

13. Turn OFF Display Column Labels.

☐ Display Column Lab

Column	Column Lbl	Value Format
1	Entity ID	%I8%
2	X Component >	%E13.7%

14. Now in the spreadsheet, click on the Value Format cell for Entity ID. Change the format to %I8%. You do this by changing the data in the Input databox above the spreadsheet and pressing the Enter or Return key to accept the data.

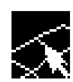
15. For all other results quantities click on their respective Value Format cells and change the formats to %E13.7%. (If you have more results quantities than will properly fit in 80 columns, then put a %E13.7%%1N% in any that extend beyond 80 columns to force it to start on a new line.) Press the OK button on this form.

OK

16. In Display Attributes also set the Report Type to Data Only.

Report Type: Data Only

17. (Optional) Change to Target Entities. Select any specific target entities you may want in the report but more importantly for this example make sure that Entity Attributes (Display Control) is set to Nodes.



Additional Display Con

Nodes

18. (Optional) Since this example requires a fair amount of setup it would be wise to save this report with a specific name if necessary to recall it later for making more .nod files. Do this in Plot Options. Enter a name for the report like *nod_file*.

Save Report As: nod_file

19. Press the Apply button.

Apply

```
TITLE
      810      1468   -2.524668e-03       257     3
SUBTITLE1
SUBTITLE2
       1 2.2122014E-4-2.1767017E-4 5.1733572E-3
       2 2.3673585E-4-2.1802561E-4 5.1421551E-3
       3 2.5836058E-4-2.1974165E-4 5.1010177E-3
       4 2.8729768E-4-2.2396364E-4 5.0501702E-3
       5 3.2454749E-4-2.3185805E-4 4.9901465E-3
       6 1.9807677E-4-1.9593856E-4 5.1675760E-3
       7 2.1360462E-4-1.9616591E-4 5.1363972E-3
       8 2.3523276E-4-1.9754541E-4 5.0952709E-3
       9 2.6418973E-4-2.0135389E-4 5.0444230E-3
      10 3.0147523E-4-2.0885997E-4 4.9843905E-3
      11 1.7493652E-4-1.7423423E-4 5.1618074E-3
      12 1.9049116E-4-1.7429129E-4 5.1305941E-3
       .
       .
       .
```

Figure 10-3 Example of a Formatted MSC.Patran .nod File.

Create a MSC.Patran .els Formatted File. A MSC.Patran .els file is an element based ASCII results file. The format of this file is:

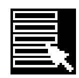
```
Record 1:          TITLE                                                      (80A1)
Record 2:          NWIDTH                                                     (I5)
Record 3:          SUBTITLE1                                                  (80A1)
Record 4:          SUBTITLE2                                                  (80A1)
Record 5 To N+4:   ID, NSHAPE, (DATA(J), J=1,NWIDTH) (2I8, /, (6E13.7))
```

where:

`TITLE`	80A1 Title Stored In An 80 Word Real Or Integer Array.
`SUBTITLE1`	Same format as TITLE.
`SUBTITLE2`	Same format as TITLE.
`NWIDTH`	Number Of Columns Of Data Stored In The File (Integer).
`ID`	Element Identification Number (Integer).
`NSHAPE`	Essential Shape Code (Bar = 2, Tri = 3, Quad = 4, Tet = 5, Pyr = 6, Wedg = 7, Hex = 8; Integer).
`DATA`	Result Quantities Organized By Column Index (Real).

1. Set the Method to Overwrite File. By default, new file will be created called `patran.prt.1`. It will be changed later.

2. In Select Results mode. Select the Result Case from the first listbox.



3. Select a result from the second listbox.
4. (Optional) If necessary, select a layer. For this example, only one layer can be selected for a proper `.els` file to be created.

5. Select the Quantities to output to the file. Select NSHAPE as the first quantity and then any element based results you wish to output to the `.els` file. Only real (floating point data) quantities are valid for proper `.nod` files.
6. Go to Display Attributes.

7. Press the Format... button.

8. Change the file name to `patran.els`.
9. Set the File Width to 80.

10. Clear the Lines / Page, Top Margin, and Left Margin databox or put zeros.

11. Turn Pagination OFF.

☐ Paginatio

12. Make the Title Alignment Left, clear the Title textbox and enter these four separate lines for the title:

Alignment: Left

```
TITLE
$NWIDTH
SUBTITLE1
SUBTITLE2
```

Of course you can substitute anything you want for TITLE, SUBTITLE1 and SUBTITLE2.

13. Turn OFF Display Column Labels.

☐ Display Column Lab

Column	Column Lbl	Value Format
1	Entity ID	%I8%
2	NSHAPE	%I8%%1N%

14. Now in the spreadsheet, click on the Value Format cell for Entity ID. Change the format to %I8%. You do this by changing the data in the Input databox above the spreadsheet and pressing the Enter or Return key to accept the data.

15. Change the Value Format for NSHAPE to %I8%%1N%.

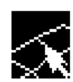
16. For all other results quantities click on their respective Value Format cells and change the formats to %E13.7%. (If you have more results quantities than will properly fit in 80 columns, then put a %E13.7%%1N% in any that extend beyond 80 columns to force it to start on a new line.) Press OK when done.

OK

17. In Display Attributes also set the Report Type to Data Only.

Report Type: Data Only

18. (Optional) Change to Target Entities. Select any specific target entities you may want in the report but more importantly for this example make sure that Entity Attributes (Display Control) is set to Element Centroids since only one value per element can be output to a proper `.els` file.



Additional Display Con

Element Centroids

19. (Optional) Since this example requires a fair amount of setup it would be wise to save this report with a specific name if necessary to recall it later for making more `.els` files. Do this in Plot Options. Enter a name for the report like *els_file*.

Save Report As:

els_file

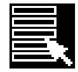
20. Press the Apply button.

Apply

```
    TITLE
        6
    SUBTITLE1
    SUBTITLE2
           1       4
    -8.3064389E-1-9.9953584E-2 0.0000000E+0 1.6953596E-1 0.0000000E+0 0.0000000E+0
           2       4
    -7.7712482E-1 1.5068005E-1 0.0000000E+0 1.2687577E-1 0.0000000E+0 0.0000000E+0
           3       4
    -8.5500902E-1 9.5036231E-2 0.0000000E+0 1.5014736E-3 0.0000000E+0 0.0000000E+0
           4       4
    -5.8506662E-1-3.0595655E-2 0.0000000E+0 1.1098222E-2 0.0000000E+0 0.0000000E+0
           5       4
     1.7681476E-1 1.2351368E-1 0.0000000E+0 1.2829211E-1 0.0000000E+0 0.0000000E+0
           .
           .
           .
```

Figure 10-4 Example of a Formatted MSC.Patran .els File.

View Global Variables in a Report. Global variables are generally single results values associated with an entire Result Case. For instance the load case number is a global variable. Time, frequency, eigenvector, mode number, design variable and design cycle are also global variables. To view such global variables there are two methods.

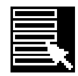


1. Select a single Result Case and result quantities for a report as explained in previous examples.

2. Under Display Attributes, press the Format button and in the Title, Header, or Footer textbox include the variable $GV:<name>, where <name> is the name of the global variable you wish to output. (If you are not sure what the available global variable names are you can always set the Object to Graph, select a few Result Cases and then you can see a list of global variables in the option pull down at the bottom of the form with X or Y value set to Global Variable.)

3. Press the Apply button and your global variables will be reported in the Title, Header, or Footer of your report.

   Or another way is to:



   - Select Result Cases and result quantities as explained previously.

   - Go to Plot Options and set the Report Type to Summary.

   - Press the Apply button. Some global variables are reported in the summary table for each Result Case selected.

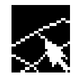
```
                    MSC.Patran Version x.x  05/13/97 11:46:58 AM
                             Analysis Code: MSC.Nastran
                Load Case: DEFAULT, Mode 9:Freq.=56.47 ;Eign=125890.898438
                 Result: Eigenvectors, Translational- Layer (NON-LAYERED)
                                 Entity: Node Vector
                                 SUMMARY INFORMATION
Maximum value - Node ID: 16, value: 0.001634
Minimum value - Node ID: 34, value: -0.009729
```

Figure 10-5 Global Variables Reported in Title for a Single Result Case

Reporting Element Nodal Data. Some results are associated with nodes, others with elements. When results are associated with elements yet you wish to see the contributions at each node due to that element, the report application will do so for you. This will also work if you wish to see the actual values at quadrature points (gauss points) inside the element.

1. Select Result Case and result quantities for a report as explained in previous examples but make sure the results are element based such as a stress tensor (displacement and constraint forces are nodal based).

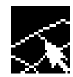
2. Go to Target Entities and make sure that the target entities selected are to be displayed or reported at Element Nodes.



Additional Display Con

Element Nodes

3. Go to Display Attributes. Press the Format button. You will note that in the columns to be reported are the Node ID as well as the Entity ID. The entity ID will be the element number and the Node ID will be the node ID to which a particular row of results is associated for that element ID. If results are at gauss or quadrature points, the Node ID will be reported as Position ID instead which is an internal ID to MSC.Patran. (Target entity display must be set at Element All Data for Position IDs.)



Format

4. Press the Apply button to generate the report.

Apply

```
                    MSC.Patran Version x.x 05/13/97 11:46:58 AM
                            Analysis Code: MSC.Nastran
             Load Case: DEFAULT, Mode 9:Freq.=56.47 ;Eign=125890.898438
              Result: Eigenvectors, Translational- Layer (NON-LAYERED)
                                 Entity: Node Vector
-Entity ID--X Component---Y Component---Z Component---Node ID-
        1    4.469031E+5   1.474780E+5   0.000000E+0       1
        1    4.172904E+5   3.858846E+4   0.000000E+0       2
        1    1.171948E+5  -7.628262E+2   0.000000E+0      11
        1    1.271206E+5   4.194981E+4   0.000000E+0      10
        2    4.172904E+5   3.858846E+4   0.000000E+0       2
        2    4.272350E+5   5.582101E+4   0.000000E+0       3
        2    1.191011E+5  -7.161045E+2   0.000000E+0      12
        2    1.171948E+5  -7.628262E+2   0.000000E+0      11
        3    4.272350E+5   5.582101E+4   0.000000E+0       3
        3    4.236081E+5   5.343995E+4   0.000000E+0       4
        3    1.151038E+5   3.807227E+2   0.000000E+0      13
        3    1.191011E+5  -7.161045E+2   0.000000E+0      12
        .
        .
        .
```

Figure 10-6 Example of Element Nodal Based Report

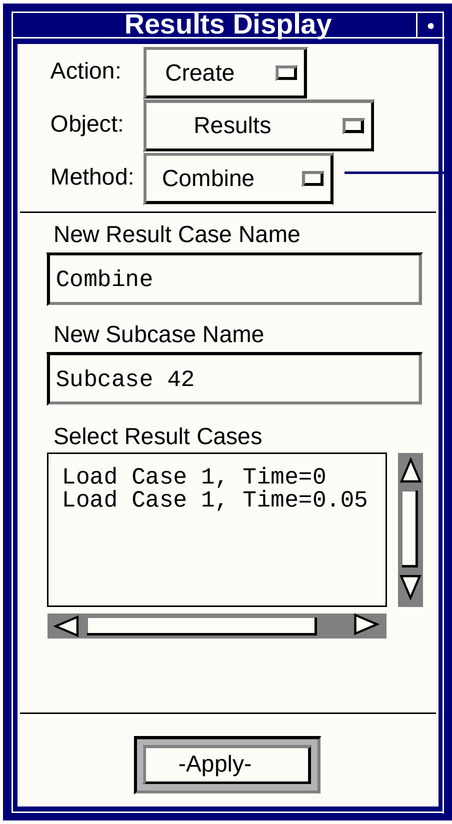
CHAPTER 11

Create Results

- Overview
- Combined Results
- Derived Results
- Demo Results
- Examples of Usage

11.1 Overview

For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). Results can be derived, combined or created in a variety of ways.



Create Results - to create results set the Method to the desired type. Results can be Combined and Derived or fictitious results can be created for demonstration and testing purposes.

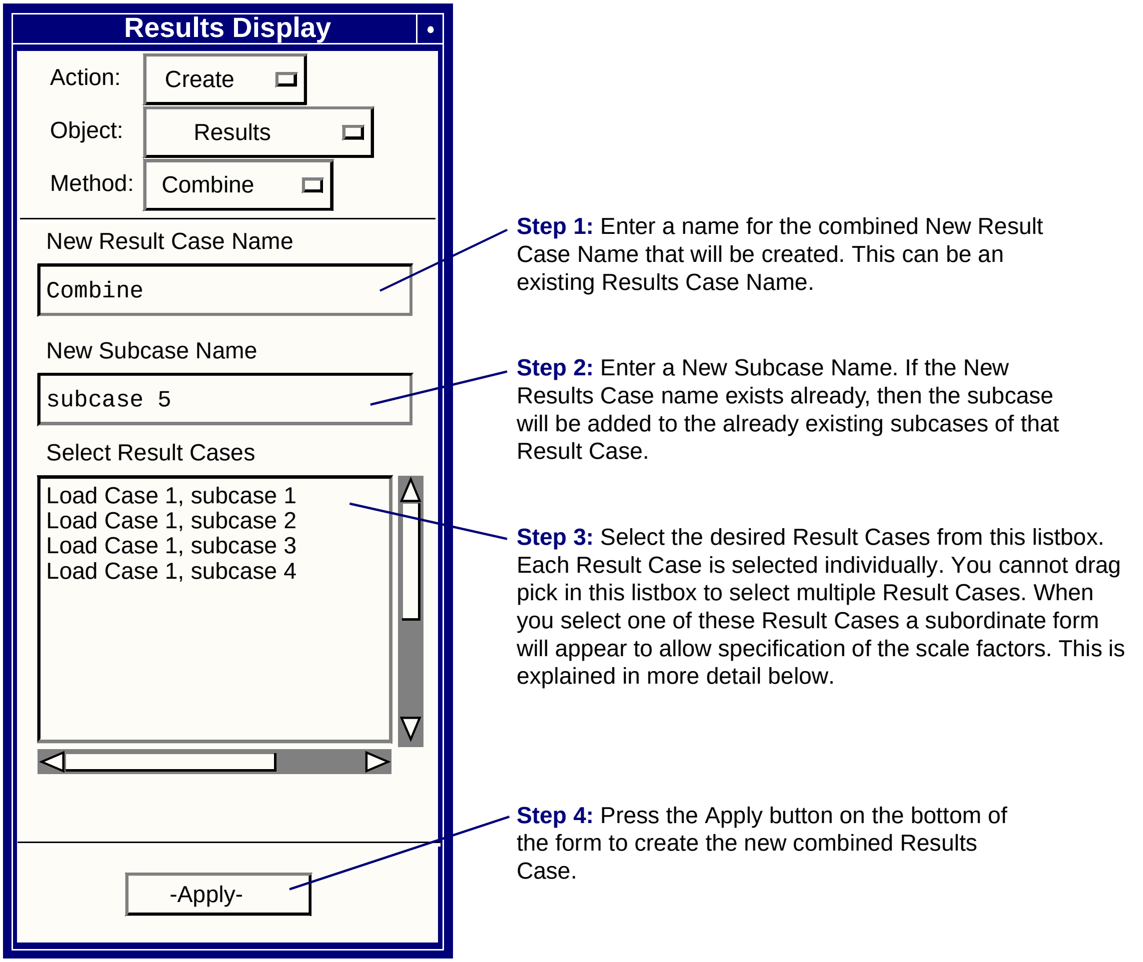
Combined Results: This Method allows for the scaling and combining of selected Result Cases. This is most commonly used for linear superposition of subcases. Although most analysis codes feature subcase superposition, this operation can also be done in MSC.Patran while the analysis code solves for only the individual subcases. Combining results using this method is limited to scaling each selected Result Case and then adding together the individually selected Results Cases to create a new Result Case and subcase. Common results quantities must be selected for the combination and each Result Case must be associated with the same finite element entities. A detailed explanation of this Method is described in **Combined Results** (p. 191).

Derived Results: This Method can be set to Maximum, Minimum, Sum, Average, and PCL Function to allow for derivation of results quantities in a variety of different manners. New results may be derived from maximum / minimum values from selected Result Cases and / or layer positions or results may be used in a PCL expression to derive a new result quantity. Derived results differ from combined results in two ways: (1) results to be derived may belong to different sets of nodes or elements which may or may not be disjointed whereas results to be combined must belong to exactly the same entities; and, (2) derived results are computed by varying procedures whereas combined results are computed by linear combinations (sum of scaled values). A detailed explanation of these Methods are described in **Derived Results** (p. 193).

Demo Results: This capability is more meant for simple creation of results for demonstration and testing purposes when no results are available. See **Demo Results** (p. 202).

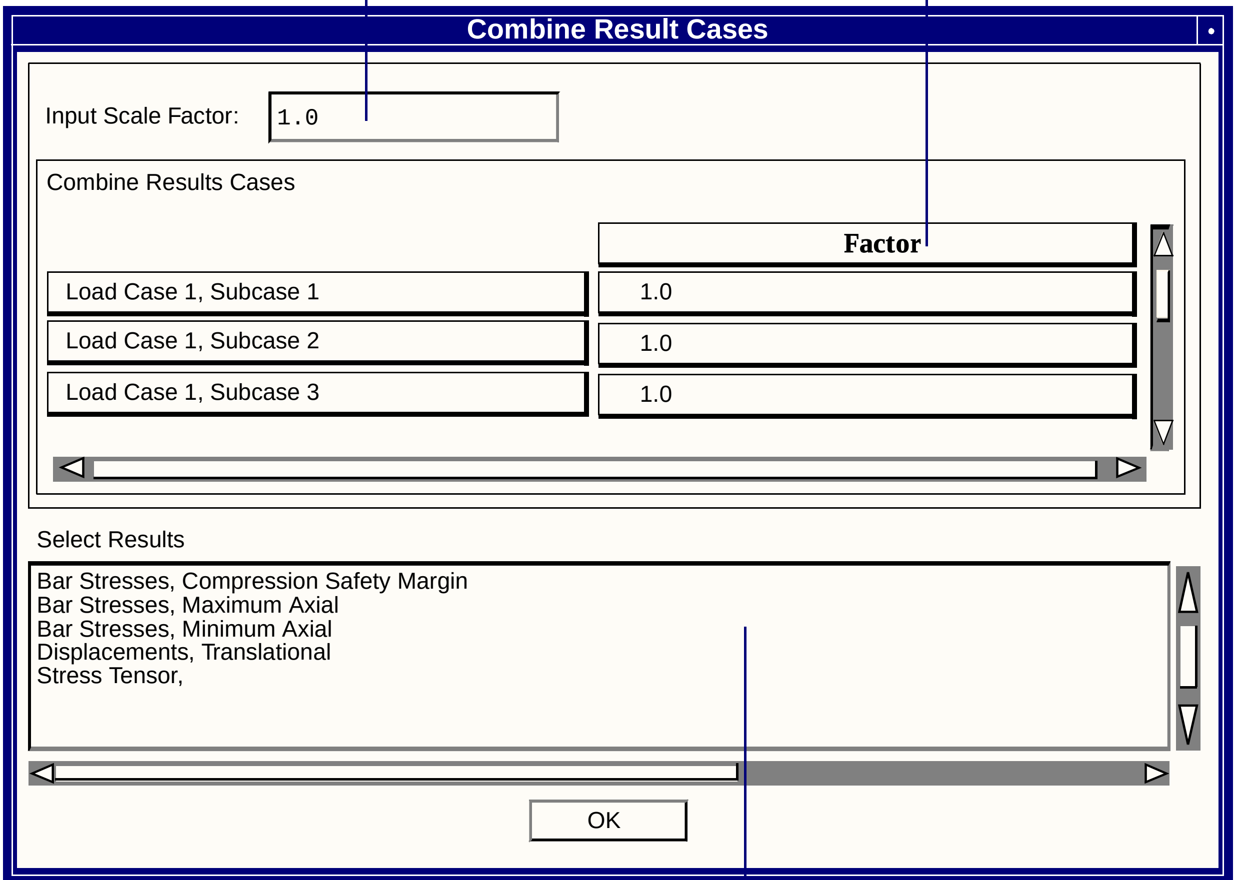
11.2 Combined Results

For performing combinations of Result Cases the form appears as shown below. This is the capability of linear superposition of subcases and / or for simple scaling of results. To combine results follow these general instructions:



When a Result Case is selected in the listbox a subordinate form appears. This is a simple form to specify the scale factors to be applied to each selected Result Case and for specifying which results associated with the Result Cases will be combined into the new Result Case, subcase. Once the Apply button is pressed the new Result Case, subcase will be available to postprocess.

To change the scale factor of one of the Results Cases, first select the Factor cell and then type an new factor in the Input Scale Factor databox. Press the Enter key to acceptance the new scale factor.

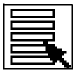


Select the Result or Results associated with these Result Cases that you wish to retain in the new combined Result Case.

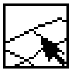
To add or remove Result Cases from this form simply select or deselect the Result Cases from the listbox on the main Results application form. As you do this, rows will be added or deleted from the spreadsheet on this form. Press the OK button when everything is set as desired. Press the Apply button on the main form to create the new results. If you need to change scale factors or the selected results to retain, select or deselect a Result Case and this form will reappear allowing for changes.

11.3 Derived Results

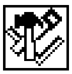
Deriving results is a powerful tool and gives a fair amount of flexibility. In this respect, Derived Results differs from Combined Results significantly. If only a simple global scaling of a Result Case is desired or the scaling and/or addition of one or more like Result Cases (subcases) is desired, then it is advisable to use Combined Results as explained in **Combined Results** (p. 191). When more complex derivations are needed such as extracting the maximum or minimum values from multiple layers and Result Cases or creating a new results quantity, then the derive capability must be used. Derive Results can be as simple as extracting max/min values or as complex as creating PCL expressions to derive new results. There are therefore, five Methods available to derive results: Maximum, Minimum, Sum, Average, and PCL Function.



Selecting Results (p. 15)



Target Entities (p. 200).



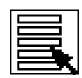
Result Options (p. 201).

Derived results can only be created. There is no Modify function for derived results.

Toggles the form to select results for derivations when the Method is set to Maximum, Minimum, Average, Sum, or PCL Function.

To derive results, these basic operations must be followed:

1. Set the Action to Create, the Object to Results and the Method to the function you wish to perform. Is it the extraction of max/min values, averaging or summing, or a more complicated expression that will have to be define with a PCL function? Set this with the Method pulldown at the top of the form.
2. Select a Result Case or Cases from the Select Result Case(s) listbox. See **Selecting Results** (p. 15) for a detailed explanation of this process as well as **Filtering Results** (p. 18).



3. Define a new name for the derived result. Default names can be accepted and no action is required.
4. Select a Result from the second listbox. It is important to understand that whatever result type is selected, the same result type will be created. That is if the result type is scalar, vector or tensor, then the newly derived result will be either a scalar, vector or tensor, respectively.

5. For Average, Sum, and PCL Function derivations skip to step **6.** Otherwise select the Quantity for comparison purposes, such as von Mises, from the Quantity option pulldown menu. The following quantities are available from vector and tensor data.

   Vector to Scalar. Magnitude, X Component, Y Component, Z Component.

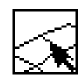
   Tensor to Scalar. von Mises, XX, YY, ZZ, XY, YZ, XZ, Minor, Intermediate, Major, Hydrostatic, 1st Invariant, 2nd Invariant, 3rd Invariant, Tresca, Max Shear, Octahedral. See **Derivations** (p. 286).

   For vector or tensor result types a resolution to the selected scalar quantity is made for comparison purposes only. The resulting result remains a vector or tensor.

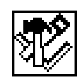
6. If more than one layer is associated with the selected result, select the layer(s) from which you wish to derive results using the Position button.

7. Skip this step unless a PCL Function derivation is requested. If the function is the derivation of results based on a PCL function, then determine what the type of the new derived result will be. That is, will it be a scalar value, a vector, or a tensor? Then define the PCL expression. This is explained below in **PCL Expressions** (p. 195).

8. Optionally you can select target entities from which to derive results if you wish to limit where the new results are calculated. By default all entities associated with the selected Result Cases will be targeted. However you may limit it to only particular selected entities as explained in **Target Entities** (p. 200). Target entities are specified by pressing the second button toggle on the top of the form.



9. Optionally you can set comparison criteria (Algebraic or Absolute) and other optional result manipulations such as scaling and coordinate transformations. This is explained in **Result Options** (p. 201).



10. Press the Apply button when ready to create the derived result.

Max/Min. Once the Apply button is pressed, the following internal operations occur:

1. Any operations under Plot Options are performed. This includes coordinate transformations, scale factors, etc.
2. The scalar Quantity is derived from vector or tensor data. For scalar results, this step is not necessary.
3. The maximum or minimum comparison is done for each target entity based on the derived scalar result Quantity for vector / tensor data or the scalar result itself for scalar data.
4. What is kept from the comparison is not the Quantity itself, but the scalar, vector, or tensor values based on the Quantity comparison for each entity. The new results are stored in the database in the same form as the originally selected results. That is, nodal data remains nodal, element centroidal data remains as element centroidal, and element nodal or element gauss data remain as is. A single new Result Case of maximum or minimum data is created from this operation.

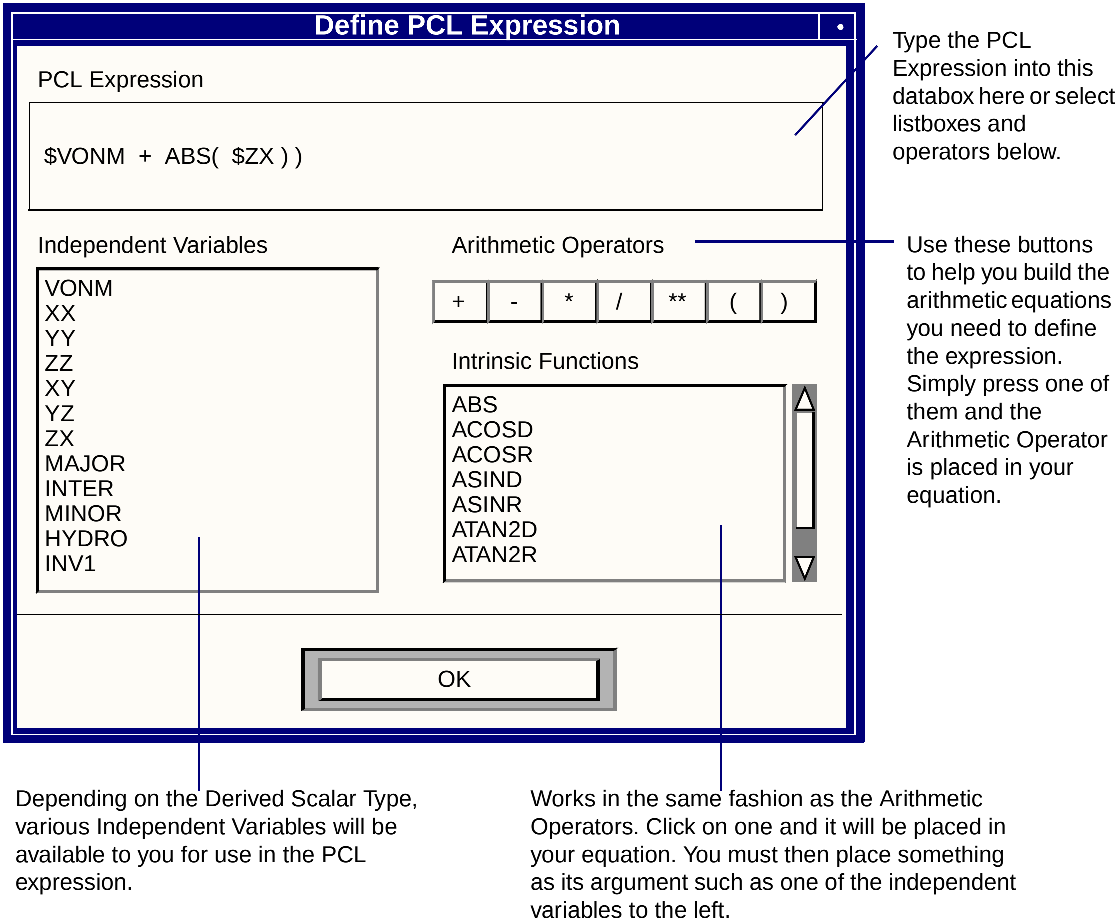
Average/Sum. Once the Apply button is pressed, the following internal operations occur:

1. Any operations under Plot Options are performed. This includes coordinate transformations, scale factors, etc.
2. The individual components of the selected result type are summed separately for all selected Result Cases and layers. For vector data, the X, Y, and Z components are summed separately. For tensor data, the XX, YY, ZZ, XY, YZ, and ZX components are summed separately.
3. For the Average method, the sum of each component is then divided by the number of occurrences. For example, if at a particular node six Result Cases contain results, the average will be computed by adding the six together and dividing by six. It is possible to have a different number of results existing at different target entities.
4. The new results are stored in the database in the same form as the originally selected results. That is, nodal data remains nodal, element centroidal data remains as element centroidal, and element nodal or element gauss data remain as is. A single new Result Case of averaged or summed data is created by this operation.

PCL Expressions. Once a PCL expression has been defined and the Apply button is pressed, the following internal operations occur:

1. Any operations under Plot Options are performed. This includes coordinate transformations, scale factors, etc.
2. The PCL expression is applied to all results in the selected Result Cases and / or layers.
3. The new results are stored in the database in the form as indicated by the New Result Type setting. Note that nodal data remains nodal, element centroidal data remains as element centroidal, and element nodal or element gauss data remain as is. A new Result Case is created for every selected Result Case and a new layer for every selected layer.

The PCL Expression builder form is shown below. You may type in your own PCL equation but for most operations, the PCL expression can be built simply by pressing the arithmetic operator buttons and selecting intrinsic functions and / or independent variables from the listboxes. No typing is required unless you need to reference user defined PCL functions. For detailed information on how to write your own PCL functions see **Understanding PCL** (p. 2) in the *PCL and Customization*.



The variables used in the PCL expression must begin with the $ sign, (i.e., $SCALAR). The variable simply signifies that when the expression is evaluated for a particular entity (say at a node), the results value for that entity will be used in the variable to derive the new result and this will be the case for every entity selected for the derivation.

Vectors require three function definitions. Tensors require six. These function definitions are separated by semi-colons (example: $XX; $YY; $ZZ / 2).

The Independent Variables available for use in the PCL equation are dependent on what results type you have asked to derive. For derivations from scalar values, the only variable available is $SCALAR. However when vector results have been selected you have the choice of these variables:

> Vector to Scalar: $MAG (magnitude), $XX (X component), $YY (Y-component.), $ZZ (Z-component.). See **Derivations** (p. 286).

For tensor results you have the choices of these variables:

Tensor to Scalar: $VONM (von Mises), $XX, $YY, $ZZ, $XY, $YZ, $ZX, $MINOR, $INTER, $MAJOR, $HYDRO, $INV1, $INV2, $INV3, $TRESCA, $MAXSHR, $OCT. See **Derivations** (p. 286).

The intrinsic functions available for scalar derived results are:

Function	Description
ABS(value)	Returns the absolute value of the result argument.
ACOSD(value)	Returns the angle in degrees which corresponds to the trigonometric cosine of the result contained in the argument.
ACOSR(value)	Returns the angle in radians which corresponds to the trigonometric cosine of the result contained in the argument.
ASIND(value)	Returns the angle in degrees which corresponds to the trigonometric sine of the result contained in the argument.
ASINR(value)	Returns the angle in radians which corresponds to the trigonometric sine of the result contained in the argument.
ATAND(value)	Returns the angle in degrees which corresponds to the trigonometric tangent of the result contained in the argument.
ATANR(value)	Returns the angle in radians which corresponds to the trigonometric tangent of the result contained in the argument.
ATAN2D(x, y)	Returns the angle in degrees to the trigonometric tangent represented by the specified x and y components in the argument.
ATAN2R(x, y)	Returns the angle in radians which corresponds to the trigonometric cosine of the result contained in the argument.
COSD(angle)	Returns the trigonometric cosine value of the result argument specified in degrees.
COSR(angle)	Returns the trigonometric cosine value of the result argument specified in radians.
EXP(value)	Returns the power function of natural logarithm base, e to the x, where x is the result value argument.
LN(value)	Returns the natural logarithm of the result argument.
LOG(value)	Returns the common logarithm (base 10) of the result argument.
SIND(angle)	Returns the trigonometric sine value of the result argument specified in degrees.
SINR(angle)	Returns the trigonometric sine value of the result argument specified in radians.
SQRT(value)	Returns the square root of the result argument.
TAND(angle)	Returns the trigonometric tangent value of the result argument specified in degrees.

Function	Description
TANR(angle)	Returns the trigonometric tangent value of the result argument specified in radians.
User Defined	Any user defined function that can accept a scalar, vector, or tensor array may be used. For example you might have function defined in MSC.Patran called my_result_function. You would then type it in manually and enter one, or more of the independent variables separated by commas, i.e., my_result_function($XX, $YY, $ZZ). Inside your function $XX, $YY, and $ZZ are defined as REAL variables. For vector and tensors, the function parameters must be defined as REAL 3x1 and 3x3 dimensioned variables respectively. The return value must be the same as the type of new result being created: scalar, vector, or tensor. See **User Defined PCL** (p. 199).

All intrinsic function accept a single scalar quantities as input, e.g., SIND($XX) except ATAN2D and ATAN2R. Also note that PCL derived results from as-is coordinate systems results in a new result in an undefined coordinate system. The new result can there for not be transformed into any other coordinate system. To avoid this it is suggested to derive the new results in a know, consistent coordinate system such as the Global system.

Examples:

Function	Description
LN($SCALAR)	Would generate the natural log of the variable $SCALAR when deriving a new scalar result.
$XX;$YY;$ZZ	Would generate a vector replicating the input result.
$XX+$YY;$YY+$ZZ;0.0	Would generate some sort of user modified result with the Z component of the vector set to 0.0.
$ZX;$YZ;$XY;$ZZ;$YY;$XX	Would swap the order of the components in a tensor.

User Defined PCL. You may enter your own user defined PCL functions also that have been compiled into MSC.Patran. A user defined PCL function must be set up to accept scalar values as the parameters and return a single scalar value. Any number of scalar parameter inputs may be defined. For example you may enter a function:

```
myFunction( $XX, $YY, $ZZ )
```

The contents of your PCL functions might look something like:

```
FUNCTION myFunction( x, y, z )
     REAL x, y, z          /* these are real scalars */
     RETURN x+y+z          /* must return a scalar value */
END FUNCTION
```

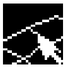
To use this function, enter it as you would a variable (no $ sign however). Examples:

Function	Description
LN(myFunction($XX, $YY, $ZZ))	Would generate the natural log of the returning value form myFunction when deriving a new scalar result.
myFunction($XX, $YY, $ZZ);2.0;2.0	Would generate a vector with the returning value of myFunction as the first component and the second and third set to 2.0.

For more information on how to create and compile PCL functions see **The PATRAN Command Language (PCL) Introduction** (Ch. 2) in the *PCL and Customization*.

Target Entities

Similar to the other graphical plot tools in the Results application, derived results can be limited to various entities via the Target Entities options.



Toggles the form to select target entities for creating derived results.

You may wish to derive results for only a few elements or nodes or those entities associated with a group or group of groups.

The following table describes in detail to which entities derived results can be targeted:

Entity	Description
Entire Model	This is the default option where derivations will be performed for all entities (nodes or elements) associated with the selected results. No consideration or selection of any graphical entities is necessary.
Current Viewport	The derived results can be limited to only the entities (nodes or elements) associated with everything in the currently active viewport.
Nodes	Individual nodes may be selected from which to create a graph. Nodes are selected graphically from the screen and fill the databox. However, you may type in any node numbers manually. Be sure to include the word Node in front of the IDs you type in manually, (i.e., Node 1 5 55 100 etc.). Elemental based results are extrapolated to the nodes and averaged.
Elements	Individual elements may be selected for which to derive results. Elements are selected graphically from the screen and fill the databox. However, you may type in any element numbers manually or by selecting them graphically from the screen. Be sure to include the word Elem in front of the IDs you type in manually, (i.e., Elem 1 5 55 100 etc.). With elemental data, values will be extrapolated or averaged to the element centroid for reporting purposes.
Groups	Derived results can be limited to only selected groups. A selected group or groups must have nodes or elements in them otherwise the derivation will not work. A listbox allows selection of the group(s) for which the derived results will be calculated.
Materials	Derived results can be targeted at only those finite elements which have certain material properties assigned to them. A listbox appears allowing selection of the materials for whose elements will be targeted.
Properties	Derived results can be targeted at only those finite elements which have certain element properties assigned to them. A listbox appears allowing selection of the properties for whose elements will be targeted for a fringe display.
Element Types	Derived results can be limited to only certain element types also.

Important: Once a target entity has been selected, it will remain the target entity for derived results until the user physically changes it.

Result Options

Derived results have various options that are available for specifying how and what operations are to be performed.

Result Options selection button. Toggles the form to select Results options for derived results.

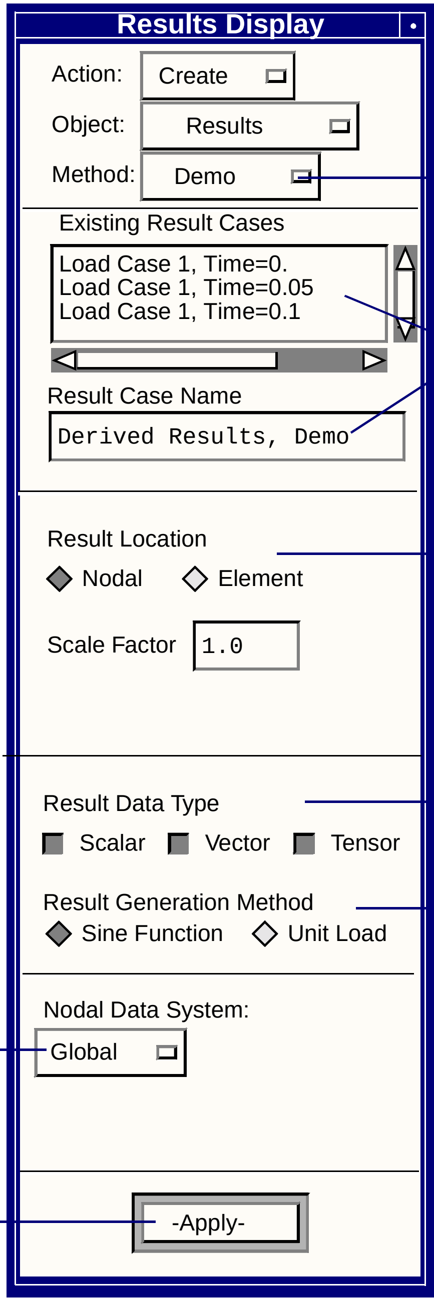
The following table describes in detail the Derived results options which can be modified:

Option	Description
Coordinate Transformations	Vector and tensor results can be transformed into any of the following coordinate systems: any user defined local system (CID), the projection of any CID, the MSC.Patran global system, a material coordinate system, element IJK coordinate system or the nodal (analysis) coordinate system depending on the type of result (vector, or tensor). See **Coordinate Systems** (p. 305) for a definition of each of these coordinate systems. The default is no transformation, which will derive data in the coordinate frame as stored in the database. Typically the solver code will calculate results at nodes in the analysis coordinate system specified by the user. These can vary from node to node. Element data can be stored from the analysis code in any coordinate system.
Scale Factor	This scale factor has the effect of simply scaling the results up or down by the specified amount before derivation.
Comparison Criteria	When doing maximum or minumum comparisons this toggle indicates the type of comparison to be made. Algebraic will consider the sign, negative numbers being less than positive numbers. Absolute will ingnore the sign and compare only the relative magnitude. Example: Comparing -6 to 5 algebraically will render 5 larger than -6, yet absolutely, -6 is larger than 5.
Save Derived Results	If this toggle is OFF, then no new Result Case and / or subcase will be created. The new result will simply be stored in a results register. This is not too terribly useful unless you are doing PCL programming and understand the use of registers.

Important: Once plot options have been selected, they will remain in effect for the derivation until the user physically changes them.

11.4 Demo Results

This form appears when the Method is set to Demo when creating results. This form allows the creation of results for demonstration and testing purposes. Demo results can be either nodal or element results. Scalar, vector and tensor result types can be created, and scale factors may be applied.



To create Demo results on models that have no results associated with them for testing or demonstration purposes set the Method to Demo.

Select an existing Results Case in which to place the Demo results or type in a new name in the databox. You must supply a Results Case and a Subcase separated by a comma and a space.

Create either nodal based or element based results and supply a scale factor if desired. If elemental based results are selected, you must also identify where to create the results (Centroid, Nodes, Gauss points, etc.)

Identify what type of data to be created (Scalar, Vector, Tensor).

Two methods are available to create demo results:

1. By product of f(x) * g(y) * h(z) where f(x), g(y) and h(z) are sine and cosine functions on the bounding box that encloses the model. The choice of functions depends on the remainder of the scale factor divided by (i.e., factor modulo 8). Results are also scaled by this factor. This method should be used if greatly varying results in the model are desired.
2. By unit load applied to the corner of the bounding box that encloses the model. The bounding box is extended to prevent singularity at the application point. The location of the unit load depends on the value of factor modulo 8. Results are also scaled by this factor. This method should be used if gradually varying results in the model are desired.

Identify the coordinate system in which to associate the results. Press the Apply button to create the results.

11.5 Examples of Usage

The following are a few typical usages of the Create/Results option in the Results application. In particular are simple examples of results combination and derivations. The examples assume the Action is set to Create and the Object set to Results.

Perform Linear Subcase Superposition

This example assumes you have two linear static analysis load cases, X and Y, and you wish to combine them to create subcase Z = X + 2Y.

1. Set the Method to Combine.

   Method: **Combine**

2. Supply a new Result Case name if desired or accept the default. If you wish the new result to be a new subcase of an existing set of Result Cases then type in the same primary name as the results that you will select for combining.

   New Result Case Name: **Combine**

3. Enter a new Subcase name (Z) or accept the default.

   New Subcase Name: **Xplus2Y**

4. Select from the listbox the Result Cases (X and Y) that you want to combine. When you select one, a subordinate form will appear allowing you to set scale factors and select the results to retain in the new combined result. Continue to select all the Result Cases you wish to combine. They will all appear in the other form. If you make a mistake simply deselect the Result Case.
5. If you wish to scale any one of the Result Cases, click on the cell which contains the Factor you wish to change on the subordinate form. In the databox at the top of the form type in a new scale factor and press the Return or Enter key to accept it. For the Y subcase this factor is changed to 2.0.

   Input Scale Factor: **2.0**

6. From the Combine Result Cases form also select the results you wish to retain in the new combined result. You may close this form down now if desired by pressing the OK button.

   OK

7. Press the Apply button.

   Apply

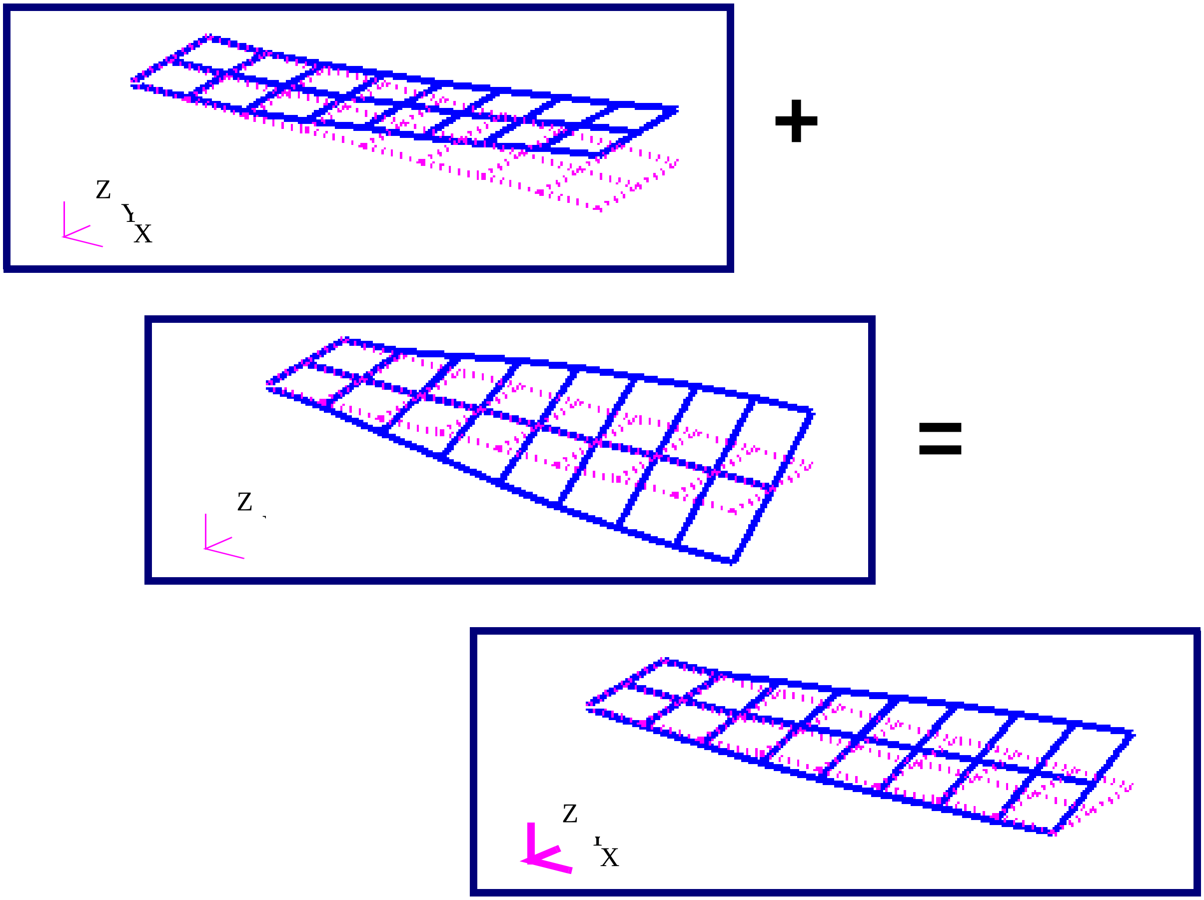


Figure 11-1 Linear Superposition of Two Load Cases to Create a Third

Derive the Maximum von Mises Stress from Multiple Layers and Subcases

This example assumes you have multiple Result Cases and that there are layers of results (such as top and bottom surface stresses) associated with each. The goal is to produce one result of the maximum von Mises stress from any given load case and layer.

1. Set the Method to Maximum.

   Method: Maximum

2. Select all the Result Cases of interest from which you want to extract the maximum values from in the top listbox.

3. Enter a new name for the derived result if desired or accept the default.

   New Result Case Name: Derived Results

4. Enter a new Subcase name or accept the default.

   New Subcase Name: MaxvonMises

5. Select either a stress tensor result or a scalar von Mises result from the second listbox.

6. If you selected a stress tensor, also change the result Quantity to von Mises.

   Quantity: von Mises

7. Select all the layers from which you want to extract the maximum values using the Positions button. The ellipses after the position name indicate that more than one layer is selected.

8. (Optional) Change the mode of Results Application to the Result Options form (press the right most button icon). Make any option changes necessary.

9. Press the Apply button.

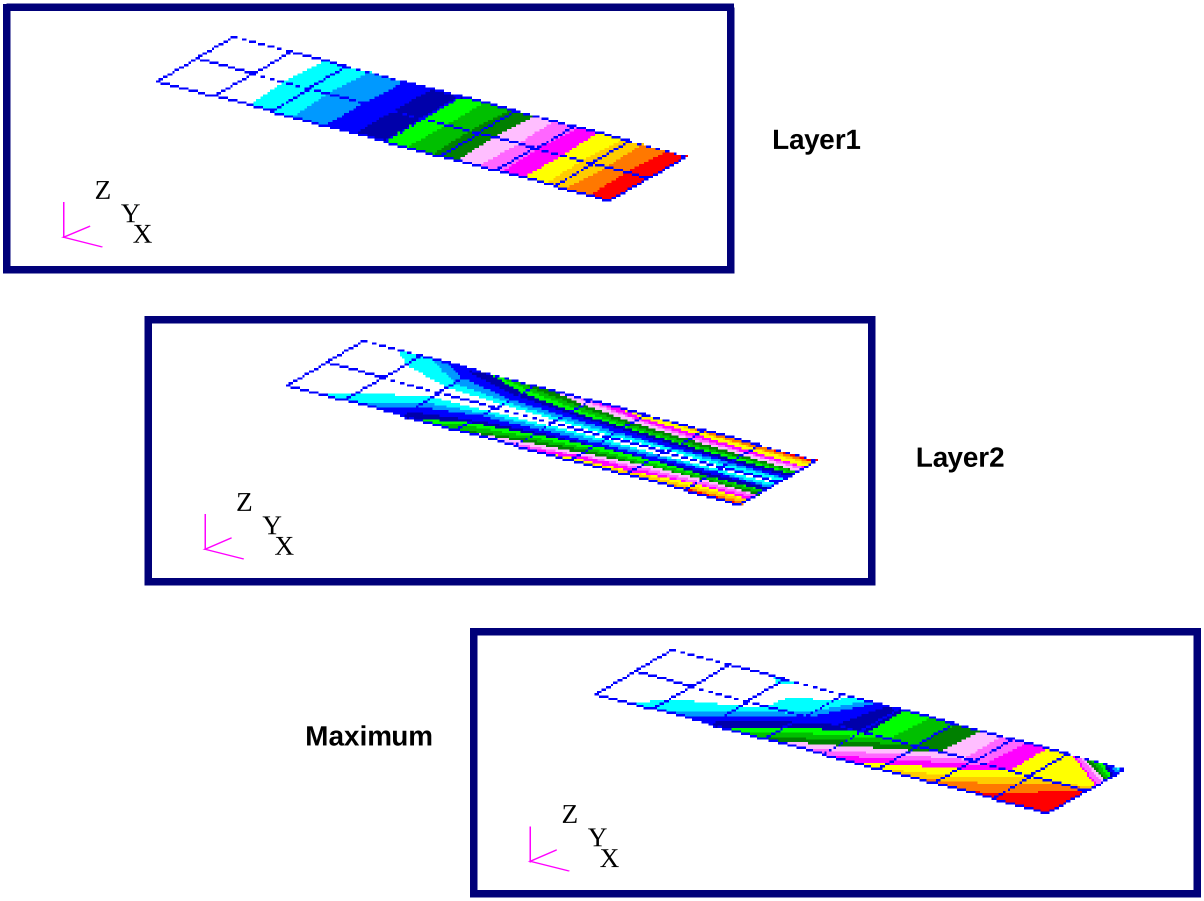


Figure 11-2 Maximum Plot of Two Different Layers

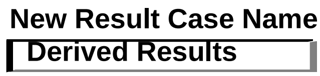
Derive von Mises Stresses in Neutral Ply Using Average Method

This example assumes you have shell elements with two layers, top and bottom. The goal is to produce one result of the average von Mises stress from the top and bottom layer, or in other word, find the von Mises stress at the neutral ply.

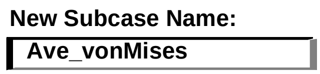
1. Set the Method to Average.

2. Select all the Result Case of interest from the top listbox.

3. Enter a new name for the derived result if desired or accept the default.



4. Enter a new Subcase name or accept the default.

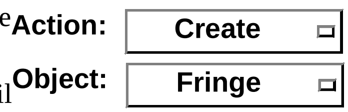


5. Select Stress Tensor result from the second listbox.
6. Select all the layers from which you want to average using the Positions button (Z1 and Z2). The ellipses after the position name indicate that more than one layer is selected.

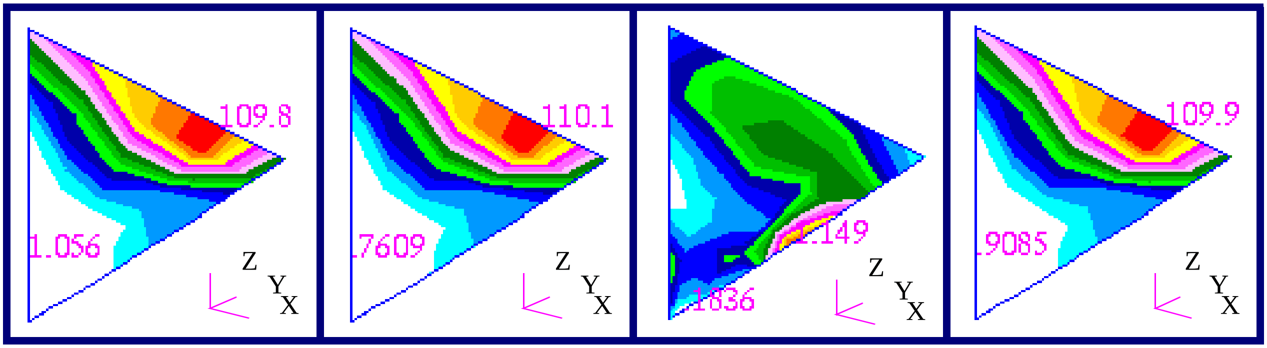
7. (Optional) Change the mode of Results Application to the Result Options form (press the right most button icon). Make any option changes necessary.

8. Press the Apply button.

9. Go to Create / Fringe and select the new Result Case and the new result and plot von Mises stress.



   von Mises stress was not actually calculated or derived until the fringe plot was made. The averaging was done over the components of the tensor only and the derived results were stored as a tensor. von Mises stress was derived from the tensor when the fringe plot was made. This mode of averaging is different than the Averaging technique available from the option pulldown menu in the Positions form where the actual von Mises stresses would have been calculated and then averaged as opposed to averaging the stress tensor components first. These two methods of averaging can result in vastly different numbers being reported. See **Figure 11-3**. The average von Mises stresses at the neutral ply in pure bending should be close to zero with the component based averaging as done in this example. The scalar based averaging will produce almost the same von Mises stress as seen in the top or bottom layers.



Top Layer- Z1 | **Bottom Layer - Z2** | **Component Based Averaging from Create/Results** | **Scalar Based Averaging from Positions Form**

Figure 11-3 Average Plots of Two Different Layers.

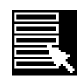
To Create a New Result from the Components of a Tensor Using PCL

This example takes the X component of a stress tensor and multiplies it by the Y component and adds the Z component to the quantity.

1. Set the Method to PCL Function.

   Method: PCL Function

2. Select all the Result Case of interest from which you want to derive new results in the top listbox.

3. Enter a new name for the derived results or accept the defaults.

   New Result Case Name: Derived Results

4. Enter a new Subcase name or accept the default.

   New Subcase Name: XtimesYplusZ

5. Select the result of interest from the second listbox, in this case a tensor quantity.

6. Change the New Result Type to Scalar.

   New Result Type: Scalar

7. (Optional) Select a layers from which you want to derive results if necessary.

   Position...(at Z1..)

8. Press the Define PCL Expression button. A subordinate form will appear. You will create an equation that looks like $XX * $YY + $ZZ. You can either type this into the textbox at the top of the form or you can create it by selecting the appropriate variables and operators in the order necessary to create the equation.

   Define PCL Expression

9. (Optional) Change the mode of Results Application to the Result Options form (press the right most button icon). Change any result option that are necessary.

10. Press the Apply button.

    Apply

To Create a New Result with User Defined PCL

This example is identical to the previous example except it does it with a User Defined PCL function.

1. First define a PCL function in an external file called `myFunction.pcl` as:

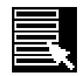
```
FUNCTION myFunction( x, y, z )
     REAL x, y, z
     RETURN x*y+z
END FUNCTION
```

2. Start MSC.Patran and issue the command:

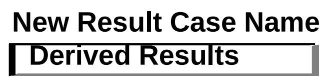
```
!!INPUT myFunction
```

3. Set the Method to PCL Function.

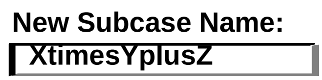
4. Select all the Result Cases of interest from which you want to derive new results in the top listbox.



5. Enter a new name for the derived results or accept the defaults.



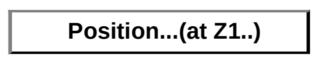
6. Enter a new Subcase name or accept the default.



7. Select the result of interest from the second listbox, in this case a tensor quantity.

8. Change the New Result Type to Scalar.

9. (Optional) Select a layers from which you want to derive results if necessary.



10. Press the Define PCL Expression button. A subordinate form will appear. This time you will enter your PCL function as:


```
myFunction( $XX , $YY , $ZZ )
```

You must type this into the textbox at the top of the form, however you can enter the variables by selecting the appropriate ones from the Independent Variables listbox.

11. (Optional) Change the mode of Results Application to the Result Options form (press the right most button icon). Change any result option that are necessary.

12. Press the Apply button.

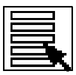
CHAPTER

12 Freebody Plots

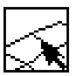
12.1 Overview

This application allows for quick and easy graphical display of freebody diagrams. Also the individual components that make up a freebody diagram can be plotted independently such as reaction forces, applied loads and internal / external loads. This application is extremely useful for graphical display of these quantities, checking for equilibrium and creating loading or boundary conditions. Once a plot is displayed, it can be converted into an equivalent load or boundary condition set for subsequent use in another analysis.

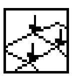
For an overview of how the Results Application works please see **Introduction to Results Postprocessing** (p. 1). To specifically make freebody plot, select Create from the Action pull-down menu on the Results application form; and select Freebody from the Object pull-down menu. The requirements and options available for a freebody plot are different than other plot types in the Results application. It is suggested that you fully read this overview and subsequent section carefully to understand how freebody plots are created in MSC.Patran.



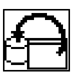
Select Results (p. 216)



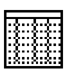
Target Entities (p. 218).



Display Attributes (p. 220).



Create Loads or Boundary Conditions (p. 223).

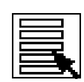


Tabular Display (p. 225).

This feature has been tailored specifically after data available from the Grid Point Force Balance Table produced from a MSC.Nastran analysis with the case control request GPFORCE = ALL. Once the results data has been loaded into the MSC.Patran database, they are treated generically. However, only MSC.Nastran results are currently supported for this feature as far as results import is concerned.

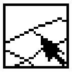
To create a freebody plot the following basic steps must be followed:

1. Set the Action to Create and the Object to Freebody.
2. Decide whether the plot will be a true freebody plot, a interface load, or forced displacement. Set the Method accordingly.
3. Select a Result Case or Cases from the Select Result Case(s) listbox. See **Select Results** (p. 216) for a detailed explanation of this process.

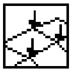


4. For freebody and interface load plots select a Result Type from the second listbox.
5. Optionally change the summation point and transformation options if necessary.

6. Select target entities. That is, define the freebody by selecting a portion of the model. More detail on this operation can be found in **Target Entities** (p. 218).

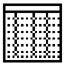


7. If necessary, change any display attributes of the freebody vector plots. See **Display Attributes** (p. 220).

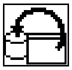


8. Press the Apply button when ready to create the freebody, interface, or displacement plot.

9. Open the tabular display for freebody load information if necessary for checking force and moment summation and equilibrium. **Tabular Display** (p. 225).

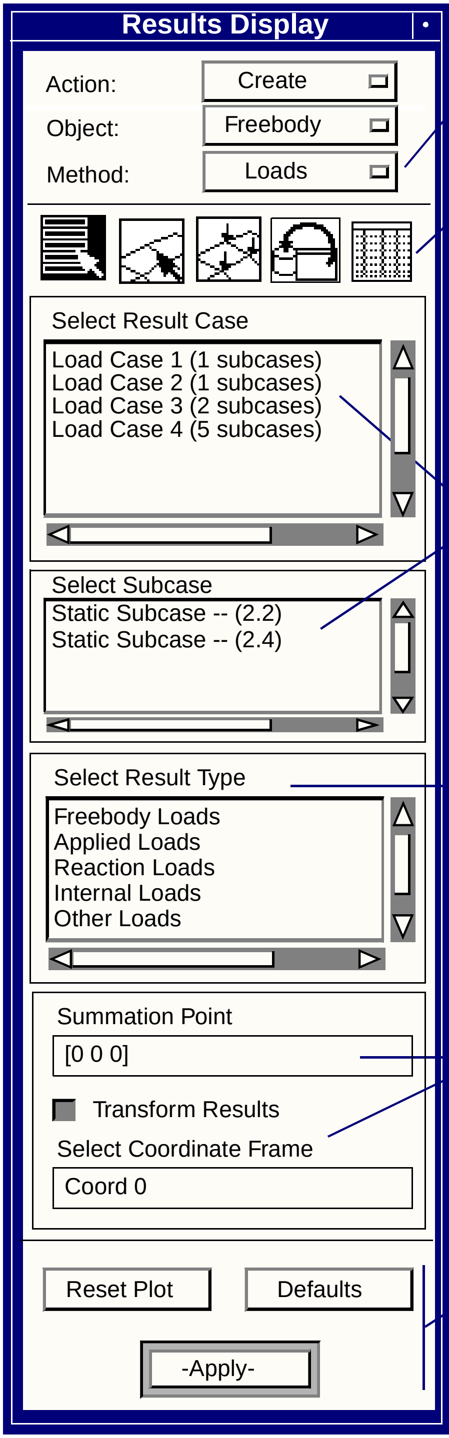


10. Finally if you wish to create a load or boundary condition from the resulting plot, do so with the Save Data option. See **Create Loads or Boundary Conditions** (p. 223).



Important:	The behavior of this plot type is somewhat different than other plots in that it cannot be saved or recalled (posted) at will. It is meant for interactive use within this Freebody application only. Once you leave this application, the freebody plots will be cleared from the screen.

Requirements. The following form is a general appearance of the Freebody plot form with a brief description. There are certain requirements that must be met in order for any results to be present in the listboxes on this form.



The Method can be Freebody Loads, Interface Loads, or Forced Displacements.

These icons change the display of the form to present different options for each Type. The first icon (default) displays the form as shown for **Select Results** (p. 216). The second is for selecting finite element **Target Entities** (p. 218). The third allows for modification of **Display Attributes** (p. 220). The forth is to **Create Loads or Boundary Conditions** (p. 223) and the fifth brings up a spread sheet for **Tabular Display** (p. 225) of results.

Displays any Result Cases that have Grid Point Force data associated with them. If listbox is empty, either no results exist in the database, or no grid point force data exists in the Result Case(s). If more than one subcase is associated with a Result Case, then another listbox will appear below this one to allow selection of the desired subcases to process.

The different result types can be Freebody, Applied, Reaction, External, Internal, or Other loads. Pressing Apply will display the loads, in vector form, on the model currently displayed. This listbox is not presented when the Method is forced Displacements.

Moments and Forces are summed about a specific point. By default this point is the origin, however it can be set to any geometric point, node or screen location using the select mechanism. Results can also be transformed to any other coordinate frame.

Pressing the Apply button will display the plot in the current viewport. The other two buttons will reset (erase) the plot and restore the default setting of the current display of the form.

1. The MSC.Nastran analysis must be run with a GPFORCE=ALL entry in the case control called out for all subcases. (Also PARAM,POST,-1 must appear in the case control or bulk data section of the input file to ensure that the Grid Point Force Balance Table (GPFB) is written to an OUTPUT2 file. This is taken care of automatically when MSC.Patran is used to create the input deck.)
2. The results in the OUTPUT2 file must be loaded into the MSC.Patran database. Results are generally in the analysis coordinate system of the nodes as specified in the MSC.Nastran input deck, however they will be converted to the basic coordinate system when used in this application.
3. Eight (8) separate vector results are loaded into the database from the GPFB table. The labels of these results can be seen in the general Results application when the Object is set to anything other than Freebody. They must not be altered or deleted or the Freebody Results application will not recognize them and therefore will not display them. In certain cases, not all these results will be present. They will be treated as zero quantities in subsequent freebody calculations if certain results do not exist such as when no applied loads actually exist. The labels are:

   Grid Point Forces, Elements

   Grid Point Forces, Applied Loads

   Grid Point Forces, Constraint Forces

   Grid Point Forces, Total

   Grid Point Moments, Elements

   Grid Point Moments, Applied Loads

   Grid Point Moments, Constraint Forces

   Grid Point Moments, Total

Description of Grid Point Force Balance (GPFB) Results. All of these base results above are vector data associated with nodes, except for the “Grid Point Forces / Moments, Elements” which are associated with elements. All of these results are accessible from other plot types in the Results application as well as for Freebody however Freebody plots will process the results differently in order to give a true freebody diagram, calculate equilibrium by summing forces and moments, and allow creation of new loading and boundary conditions.

- **Grid Point Forces/Moments, Elements -** These results are vector data associated with elements. Each element position corresponds to a node of the element where the results value at that node is that element’s contribution to the total internal load for that particular node. Processing these data with the general plot tools in the Results application is somewhat meaningless.
- **Grid Point Forces/Moments, Applied Loads -** These are the nodal equivalenced applied loads of the model. They are vector data stored at the nodes. Processing these data with this application or the general plot tools in the Results application should be very similar.
- **Grid Point Forces/Moments, Constraint Forces -** These are the nodal equivalenced constraint forces at the restrained locations of the model. They are vector data stored at the nodes. Processing these data with this application or the general plot tools in the Results application should be very similar.

- **Grid Point Forces/Moments, Total -** These are the total nodal equivalenced loads at each node of the model. They are vector data stored at the nodes. Normally they should all be zero when equilibrium exists. However, there are occasions when these values are not zero. This typically happens when multipoint constraints and rigid elements exist in the model since they are not taken into account in MSC.Nastran Grid Point Force Balance Table.

Description of Freebody Tool Plots. From the four types of GPFB results for both forces and moments, the Freebody Results application can evaluate and display six (6) different types of vector plots when the Method is set to freebody Loads or Interface loads plus displacement plots:

Freebody Loads This is by far the most interesting and important vector display. It consists of determining the total internal loads for all externally exposed nodes on the model (these are determined by selecting target finite elements or by using what is displayed on the screen) and displaying them along with any applied and reaction loads. For equilibrium to exist all of these vector loads should add up to zero in each component direction. Occasionally this is not the case since MPC and rigid element contributions are not taken into account. This can be checked visually or via the freebody load spreadsheet available by picking the right most button icon on the parent form where a force and moment summation is presented.

Applied Loads By selecting this result type, the nodal equivalenced applied loads of the model will be plotted in vector form on the displayed model. Similar operations can be done with Marker (Vector) plots with these data.

Reaction Loads By selecting this result type, the nodal equivalenced reaction loads of the model will be plotted in vector form on the displayed model. Similar operations can be done with Marker (Vector) plots with these data.

Internal Loads By selecting this result type, the nodal equivalenced total internal loads of the model will be plotted in vector form on the exposed nodes of the displayed model. The model can be cut up and different target finite elements may be selected to view the internal loads anywhere in the model.

External Loads By selecting this result type, the nodal equivalenced total of all loads associated with those elements attached to the freebody (but not part of the freebody) along the boundary will be plotted in vector form.

Other Loads The total loads (Applied + Reaction + Internal) will not sum to zero if rigid elements, MPCs, thermal loads, or other external influences not accounted for are present. By selecting this result type, these external influences are displayed.

Displacements For forced displacements, the results are plotted as vector displacement data only on the external nodes of the freebody.

All vector values from all of the above results types are also reported tabularly to a spreadsheet including the summation of forces and moments to calculate totals and check equilibrium.

Three types of graphical display are possible each available as a Method of the Freebody application.

- **Freebody Loads -** This type will allow for graphical vector display of freebody loads at all external (exposed) nodes of the target entities (elements) selected - that is, a freebody diagram. In order for this type to be graphically displayed, a freebody should be defined as a group of elements called the target entities. If no target entities are specified, the target entities will default to all displayed elements in the current viewport. In addition, each individual nodal equivalenced load type (applied, reaction, internal, external, other) that make up a freebody diagram can also be plotted in vector form on the exposed nodes of the displayed freebody. The model can be cut up and different target elements may be selected to view these anywhere in the model. Moments and Forces will be summed about the specified Summation Point and will be transformed to another coordinate system if requested. By default all graphical displays are presented in the Global coordinate system unless a transformation is requested, even if the results are stored in the database in their analysis coordinate systems. If a new load condition is created from this, it will consist of a new force and/or moment field which will be referenced by the force and/or moment load.
- **Interface Loads -** This type allows for the calculation of the total forces and moments acting across a boundary. Graphically this displays a single vector quantity at the summation point due to the loads from selected nodes in the freebody. Target entities in this case are the elements making up the freebody and the nodes in the freebody that are to be used in calculating the interface load. Both must be selected. There is no default and an error will be issued if no target entities are selected. For example, you may wish to know the total equivalent force and moment at a location across a boundary interface (such as where a wing intersects a fuselage). Therefore you would select the node or location at which to sum the forces and moments (the Summation Point) and then select the nodes along the interface and the elements to which those nodes belong as the target entities. Generally the summation point will also belong to the target entities but this is not a requirement. Results belonging to nodes not associated to the target elements will be ignored in the calculations. The reason for selecting nodes and elements is that the nodal forces and moments are used at the selected nodes and the internal elemental forces and moments from the selected elements. If a new load condition is created from this, it will consist of a new force and/or moment load at the node (summation point). If the location is something other than a node, you will be prompted if you would like to create a node at that location.
- **Forced Displacements -** This type of freebody plot simply displays displacement results at the external edges of the freebody in the form of vector quantities. The vectors are shown along the edges of the selected elements when the display attribute for displaying on free edges only is turned ON. The displacements can be saved as an enforced displacement boundary condition and then subsequently used in a local analysis of the freebody which will reproduce the exact displacement conditions.

12.2 Select Results

This is the default display of the Freebody plot form when it first appears to Select Results.

Toggles the form to select results for freebody plots. This is the default mode of the Freebody form.

The following table describes each widget entity in the freebody Select Results mode of the form:

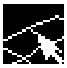
Entity	Description
Select Result Case	Select a Result Case from this listbox. If nothing appears in this listbox, be sure that the results have been imported and that Grid Point Force results exist as described in the previous section. See **Requirements** (p. 212). The number of subcases for the displayed Results Cases is displayed to the right of the title. Multiple Result Cases may be selected, but be aware that the display will cycle through them one at a time. Selecting multiple Result Cases makes the most sense when creating multiple loading conditions or automatic multiple hardcopy plots.
Select Subcase	If more than one subcase is associated with a Result Case, then another listbox will appear below this one to allow selection of the desired subcases to process. Again multiple subcases can be selected. The behavior of selecting multiple subcases is multiple plot displays.
Select Result Type	The different result types can be Freebody, Applied, Reaction, External, Internal, or Other loads. Pressing Apply will display the loads, in vector form, on the model currently displayed. This listbox is not presented when the Method is forced Displacements. An explanation of each of these plot types can be found in **Requirements** (p. 212).
Summation Point	Moments and Forces are summed about a specific point. By default this point is the origin, however it can be set to any geometric point, node or screen location using the select mechanism.
Transform Results	Results can also be transformed to any other defined coordinate frame. Turn this toggle ON if you wish to do a coordinate transformation.
Select Coordinate Frame	When the Transform Results toggle has been turned ON, this widget will appear. Simply select the coordinate system graphically from the screen or type in the name of an existing coordinate frame such as "Coord 1."
Reset Plot	This will remove any freebody plot from the graphics screen.
Defaults	This will reset all widgets to their default values and will remove any plot.

Press Apply at any time to create the desired vector plot. If no target entities have been selected, the default display is what is currently displayed on the screen except for Interface Loads which must have target entities selected. See Target Entities (p. 218). Any display attributes and target entities previously set are retained when the Apply button is pressed. Reset Plot will erase all vector displays from the screen and Defaults will set the form back to its default settings.

Important: Once a results are selected and any other settings set, they will remain until the user physically changes them, the Default or Reset buttons are pressed or the Action / Object / Method is changed. This will clear the plot and set the settings to defaults.

12.3 Target Entities

This is the Target Entity display of the Freebody plot form. This form is used to define the actual freebody of the model. For Freebody Loads the default is to plot on the entire existing display.



Toggles the form to select target display entities for freebody plots.

This form creates a list of elements and/or nodes for defining the freebody for the freebody diagram or other vector display. The following table describes the widgets on this form and how to properly select target entities:

Entity	Description
Select By	Elements and nodes may be selected individually or by association to groups, material and element properties or adjacent elements. Set this pulldown to indicate which type of entity is to be selected. For freebody plots only elements are involved. For Interface Load plots both elements and nodes need to be selected which causes this to be a two step process. First select the nodes and then select the elements on one side or the other which will contribute to the determination of the interface load. For Forced Displacement plots, only elements are necessary to define except when bar elements are involved. Then it is necessary to define the outer edge with node picks since the selecting algorithm uses free edges as its criteria. Bar elements do not have free edges. The Adjacent Elements pick is convenient for adding layers or subtracting layers of elements.
Auto Add/Remove	If this toggle is set to either Auto Add or Auto Remove then whenever you graphically select an entity or pick from a listbox, all entities are automatically added or removed from the listbox that displays the freebody entities on this form.
Select/Entities	This is the listbox that will display the freebody definition. This definition will be made up of a list of entities (nodes and/or elements). The textbox can be edited manually but this is not recommended in case the syntax is not followed properly. It is best to fill the textbox by graphically selecting entities or selecting entities from a listbox of groups, materials, or properties. If nothing is in this listbox when Apply is pressed, the Freebody Load plot will be applied to whatever is visible in the current display. The Interface Loads require that you pick both target nodes and entities (see Description of Freebody Tool Plots (p. 214) for an explanation. Forced Displacements requires only elements except for bar elements which require nodes.
Add/Remove	Press the Add or Remove buttons to add entities into the freebody entities listbox. You only need to do this if you have not turned on either Auto Add or Auto Remove or that option is not available to you. (You cannot remove adjacent bar elements).
Undo	Undo clears out the last operation done and reverts to previous selected entities. Multiple undoes are possible.

Entity	Description
Clear	Clear will clean out the Freebody Entities listbox completely and two Clears will clear the Undo memory dimming the Undo button.
Show Selected Elements	When switched ON, the viewport displays only what is in the Freebody Entities listbox. The freebody or other vector plot can be displayed on the freebody with only the freebody displayed or with the entire posted model displayed.
Show All Posted FEM	If ON, the entire posted model will be displayed. The freebody or other vector plot can be displayed on the freebody with only the freebody displayed or with the entire posted model displayed.
Create New Group	Turn this toggle ON if you wish to create a group from the selected target entities that define the freebody.
Include Nodes	If this toggle is ON the nodes of the target entity elements will be included in the new group.
Overwrite	Specifies whether you wish to overwrite the group if it already exists, otherwise you will be prompted for overwrite permission is the group already exists.
Group Name	The default name for the new group to be created will be `Fbdy_Group`. Change it if desired.
Create Group of Entities	Press this button to actually make the group.
Reset Plot	This will remove any freebody plot from the graphics screen.
Defaults	This will reset all widgets to their default values and will remove any plot.

When Apply is pressed, the display will update using the new entities and will retain whatever results and display attributes have been selected previously.

Important:	Once a target entity has been selected, it will remain the target entity for the freebody plot until the user physically changes it or he changes the Action / Object / Method or presses the Default or Reset buttons. This will reset the target entities selection to its default and clear the plot.

12.4 Display Attributes

Freebody plots can be displayed with various attributes. Display attributes for freebody plots area accessible by pressing the Display Attributes selection button on the Results application form with the Object set to Freebody. It is not required to use this option. Appropriate defaults will be used.

Toggles the form to change display attributes for freebody plots.

The following table describes in detail the freebody display attributes which can be modified:

Attribute	Description
Show:	For freebody plots you can display Forces, Moments, or Forces and Moments simultaneously. This is also true for interface loads. For forced displacement plots these choices are Translational, Rotational or both simultaneously.
Display As:	For all three types of freebody plots, the resulting vectors can be plotted as either components or resultants.
Dimensions:	This option is an easy way to display only the components that you want. By default, vectors in three dimensions will be displayed. You can constrain this to particular planes with respect to the global coordinate system. Only those components will be displayed or, if a resultant display is requested, the resultant will be calculated with respect to that plane only.
Color/Components	You can turn ON or OFF the display of any component or resultant and change their colors with these toggles and color widgets.
Display Free Edges Only	Only vectors of freebody or forced displacement plots on free edges of the model will be displayed if this toggle is ON. This essentially will eliminate any internally applied loads or internal reaction loads from the display. This type of display will not work for bar element models.
Display Nodal Contributions	For Interface Loads only the contributions from each individual node can be plotted simultaneously as well as the total interface load if this toggle is turned ON.
Scale Arrows / Constant	If the Scale Arrows toggle is turned ON then vector lengths will be scaled relative to one another based on their magnitudes. Constant will keep all vector arrows the same length. Both are scaled relative to the a percentage of the screen size.
Length	This is a scale factor to be applied for sizing the vector arrow lengths. By default it is set to 10% of the screen size.
Hide Results Near Zero	If you wish to see all results whether they are zero or not, turn this toggle OFF.

Attribute	Description
Zero Tolerance	Sometimes it may appear that some vectors of the freebody plot are not being displayed. This could be because the Hide Results Near Zero toggle is ON and the Zero Tolerance is set too high. Change this tolerance if necessary to view or not to view vectors below this gate value.
Display Values	Turn this toggle ON or OFF depending on whether you wish to see the values associated with the vector. You also have control of the display of these values.
Exponential /Fixed	Values on vectors can be displayed as either fixed (real) values or in exponential form.
Significant Digits	You may also set the number of significant digits of the vector label values.
Display Title	Turn this title on if you wish to display a title with the plot. For more versatile title display, use the Titles utility from the main **Titles** (p. 284) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.
Color / Title	Titles can be colored and specified in the textbox. If no title is supplied but the Display Title toggle is ON, a default title will appear.
Font / Location	You may create a title to display in the upper or lower left corners of the screen and specify the font size.
Automatic Print	When toggled ON, a hardcopy print command will be generated and sent to the currently selected printer definition under the Print application. If multiple results are selected, then multiple plots will be sent. This does not work when MSC.Patran is run in batch mode with no graphics.
Text Report	If this toggle is turned on the data associated with this plot will be dumped to a file called <db_name>_freebody_data.dat.
Append	If this toggle is ON then the text report will append to any already existing file.
Display via Session File	This option will put the session file call to create the freebody plot in the MSC.Patran command line. The plot will not be created until you issue a RETURN from the keyboard with the cursor control in the command line. This is useful if you wish to look at multiple plots sequentially. Select all the Result Cases and / or subcases and / or result types, then turn this toggle on. Each time you want to see the subsequent plot press the RETURN key when the focus is in the command line. This will also create a session file called <db_name>_play_freebody.ses.
Reset Plot	This will remove any freebody plot from the graphics screen.
Defaults	This will reset all widgets to their default values and will remove any plot.

When Apply is pressed, the display will update using the target entities and results which have been selected previously and the new display attributes. Defaults will set the form back to its default settings and Reset Plot will erase the vectors from the screen.

Important:	Once display attributes have been selected, they will remain in effect for the freebody plot until the user physically changes them or the Action / Object / Method is modified.

12.5 Create Loads or Boundary Conditions

This is the portion of the Freebody plot tool that saves the displayed data as load or boundary condition sets. It is not necessary to use this option unless you wish to create them from the Freebody, Interface, or Displacement plots.

Toggles the form to create loads and boundary conditions from freebody plots.

Creating loads and boundary conditions from freebody plot displays is a two step process. First a field is created for the appropriate type such as force, moment, or displacement. Then the fields are assigned to a Load Case. Only what is graphically displayed on the screen is actually created in the fields. The following table explains each option.

Attribute	Description
Create Force/Moment Displ/Rotational Field	Create both a force and a moment field or one or the other (translational and rotational fields for forced displacements). What is displayed graphically is what is created in the new field. You must create at least one field in order to create a load set. Once fields are created you may use them as you would any other field. Then access them and modify or delete them from the Fields application. The fields are created in the coordinate system specified as the transformation coordinate on the main (Select Results) form, otherwise they will be written in the global system. For Interface loads, the display attribute “Display Nodal Contributions” must be on in order to create fields for this type of plot.
Assign Fields to LBC	Toggle ON to assign the fields to a new LBC set (load set). Then enter a name and assign it to a load case. Once a load set has been created you may access, modify, or delete it from the Loads/BCs application. The same naming convention is used here as is used when creating fields. If the Type is Interface Loads, then only this option is presented since only at a single node will force and moments be created. No fields will be created. Therefore, no fields are necessary.If you wish you may assign a new load case name and a new load case will be created. For Interface loads only the “Total Load” is assigned to the LBC, not the fields.
Overwrite / Increment	When multiple results are selected for display, multiple fields and LBC set names will be created. The names of the fields and LBC set names will be augmented by appending a version on the end of them if Increment is selected (e.g., Fbdy_Force.001, Fbdy_Force.002, etc.). You can overwrite or increment the names.
Field Name	The default name for a field is `Fbdy_<type>` where the type can be Force, Moment, Translation, or Rotation. Change it if desired.
LBC Name	The default LBC name is `Fbdy_LBC` or `Fbdy_Disp_LBC`. Change it if desired.
Load Case Assignment	When creating a new LBC you can select to which Load Case it is assigned from this listbox. The default is the Default load case.

Attribute	Description
Insert / Increment	This works similar to Overwrite/Increment with field names. For Load Case names however the option is to either insert the new LBC into the currently selected Load Case or to create a new Load Case with the name incremented with a version number (e.g., LBC.001,LBC.002, etc.).
LC Name	You can type in a new Load Case name here is you wish or select an existing one from the Load Case Assignment listbox.
Reset Plot	This will remove any freebody plot from the graphics screen.
Defaults	This will reset all widgets to their default values and will remove any plot.

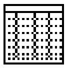
Pressing Apply here will create the specified fields and load set for the currently displayed plot for access and assignment in a subsequent finite element analysis. Defaults will set the forms back to their default settings and Reset Plot will erase the vectors from the screen.

Some consideration to be aware of when creating fields and LBC sets.

- If a planar display is plotted of three dimensional data, then the appropriate column is zeroed in the field.
- The display attribute “Display Free Edges Only” can limit the amount of information that is placed in a field for Freebody and Displacement plots.
- The display attribute “Display Nodal Contributions” for Interface loads must be on in order to create fields. Only the “Total Load” will be written to the new LBC set, not the fields.
- The display attribute “Hide Results Near Zero” affects the amount of data written to the fields for Freebody Loads and Interface Loads but not Displacement. All displacement data is written no matter how small since all the data is necessary for a re-analysis. Zero displacement is a necessary boundary condition.
- Once a new LBC is created or an existing one is modified, you must make sure that the LBC is the current LBC in order to visualize the LBC markers on your model. This is done in the Loads/BCs application.

12.6 Tabular Display

The Show Spreadsheet icon on the Freebody plot tool accesses a spreadsheet.



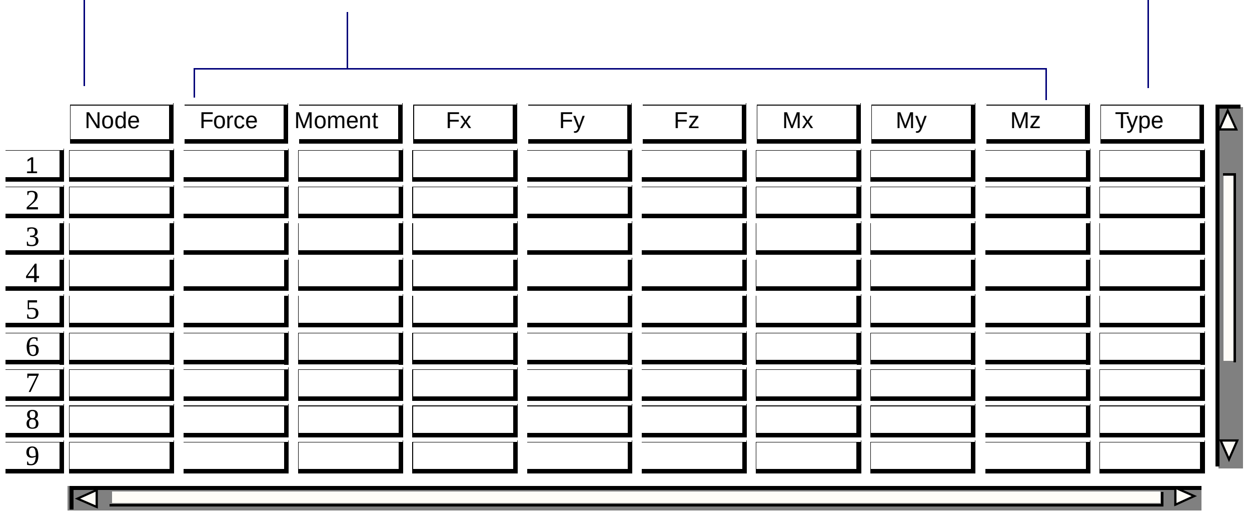
Pressing this button icon makes the freebody spreadsheet appear.

This spreadsheet will fill itself with result values for each node that has a vector plot summarizing the loads or displacements whether they be internal, reaction, applied or other. It also summarizes the totals for equilibrium checks and other purposes. Only the nodes of the currently displayed freebody plot will be reported in the spreadsheet. Also the spreadsheet will be cleared and redisplayed each time a new plot is created. You may bring up and close down the spreadsheet and the data will remain intact until a new plot is created.

The Node number is reported in this column. The same node may appear more than once if more than one load type is associated with it.

The two Force and Moment resultants and each component are reported in the spreadsheet independent of what is displayed on the screen.

Type refers to the load type for the particular node that is reported. Type can be Applied Load, Reaction Load, Internal Load, or Other load. Other load refers to the fact that the Total load is other than zero, therefore some unaccounted load contribution exists for that node.



The labels on the spreadsheet above will change to displacements when the Forced Displacement method is displayed.

The totals for each component are reported at the bottom of the spreadsheet. When Freebody Loads are plotted on the graphics screen, these totals should be zero (equilibrium should exist). If not, then other unaccounted load contributions must exist, such as those from rigid element, MPCs and externally coupled stiffness matrices. You may also click the mouse on each cell to get more detailed information in a text box below the spreadsheet. Also if the Method is set to Interface Loads, contributions from all nodes specified as target entities will be displayed even though graphically only the summation point displays the vector plot.

A Report button is also available from this spreadsheet that will write its contents out the a file called <db_name>_freebody_data.dat where <db_name> is the name of the database.

A special utility PCL function can be used in conjunction with the freebody spreadsheet to interact in almost any way imaginable. If you define the function below and compile it into MSC.Patran with a !!input <function_name> before the spreadsheet is opened, then anytime you click on a cell in the spreadsheet this call back function is called. Please see **The PATRAN Command Language (PCL) Introduction** (p. 5) in the *PCL and Customization* for details on how to program in PCL.

freebody_spreadsheet_cell_cb (spreadsheet_id, textbox_id, segment_id, from_column, from_row, to_column, to_row)

Description:

This is a call back function that is called anytime a cell or group of cells is selected in the freebody spreadsheet. The arguments to the function are all inputs supplied by the callback function of the spreadsheet. The function must be defined by the user and compiled into MSC.Patran before the freebody spreadsheet is opened. The contents of this function are left up to the user's imagination.

Arguments:

WIDGET	spreadsheet_id	This is the widget ID of the spreadsheet. Knowing the widget ID you have full control over it and can make modification to the spreadsheet such as adding or removing rows or columns.
WIDGET	textbox_id	This is the widget ID of the textbox on the bottom of the form under the spreadsheet. Knowing the widget ID of the textbox you have full control over it such as adding and removing text from the textbox.
INTEGER	segment_id	This is the graphics segment ID. Knowing the segment ID into which the freebody plot has been graphically written (the vectors and labels) you have full control over them such as deleting, modifying or adding to the graphics already in that segment.
INTEGER	from_column	This is the first column in the range of selected cells from the spreadsheet.
INTEGER	from_row	This is the first row in the range of selected cells from the spreadsheet.
INTEGER	to_column	This is the last column in the range of selected cells from the spreadsheet.
INTEGER	to_row	This is the last row in the range of selected cells from the spreadsheet.

Example.

A good example of usage of this PCL utility is the highlighting of nodes when a cell or range of cells is selected. For example, say that you wish to know graphically which node a particular value in the spreadsheet referred to. Although the node ID is indicated in the spreadsheet, with a little programming, a user could create a function that when a cell or cells are selected, the corresponding node(s) is/are highlighted graphically on the screen.

12.7 Examples of Usage

The following example illustrates the basic usage of the Freebody plot application. Figure 12-1 represents a continuous truss structure subject to a vertical end force and gravity force at the center of gravity. The structure is fixed to a wall. The goal is to run a static analysis and to determine and graphically display the reaction loads experienced at the wall, and to cut the structure at any point and display a freebody diagram as well as display the internal loads at various locations. Also a subsequent detailed analysis is required at one of the joints, therefore the internal loads need to be saved as a load case set.

An MSC.Nastran analysis is performed with a Case Control statement requesting a Grid Point Force Balance table: GPFORCE = ALL. The applied loads are shown as displayed by the Results Freebody application. The resulting OUTPUT2 file is imported into the MSC.Patran database.

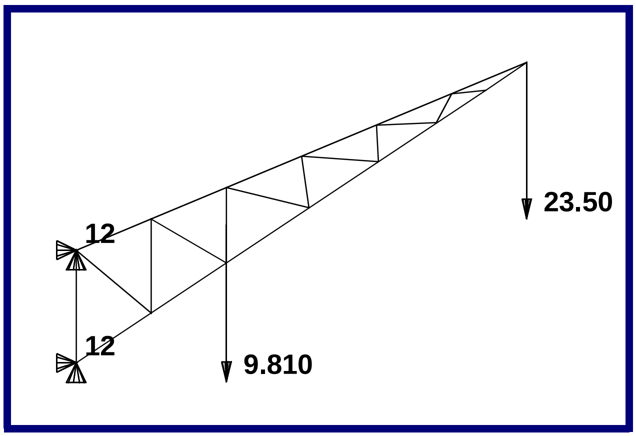
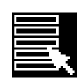


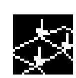
Figure 12-1 Example Plot of Applied Loads for the Continuous Truss Example

Display the Applied Loads from the Model

1. Set the Action to Create, the Object to Freebody and the Method to Loads.
2. Make sure the Select Results button icon is selected (Method = Load).



3. Select a Result Case (and a subcase if one exists).
4. Select the result type Applied Loads from the Result Type listbox.
5. Press the Apply button to display the nodal equivalenced applied loads.
6. To change any display attributes of the vector plot, press the Display Attributes button icon. Press the Apply button again to affect any additional changes.

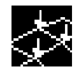


   In Figure 12-1, the two nodal forces appear. The gravity load in this case is concentrated at a single node. Had the gravity load been applied as a MSC.Nastran GRAVity force, equivalent nodal forces would have been distributed at all node points. In this case it was not necessary to select Target Entities. The whole model was used by default.

Display the Reaction Forces from the Model

1. Set the Action to Create, the Object to Freebody and the Method to Loads.
2. Make sure the Select Results button icon is selected (Method = Load).

3. Select a Result Case (and a subcase if one exists).
4. Select the results type Reaction Loads from the Result Type listbox.
5. Press the Apply button to display the nodal equivalenced reaction loads.
6. To change any display attributes of the vector plot, press the Display Attributes icon. Press the Apply button to affect any additional changes.



   Again, the entire model was used as the default display. No target entities were selected in **Figure 12-2**.

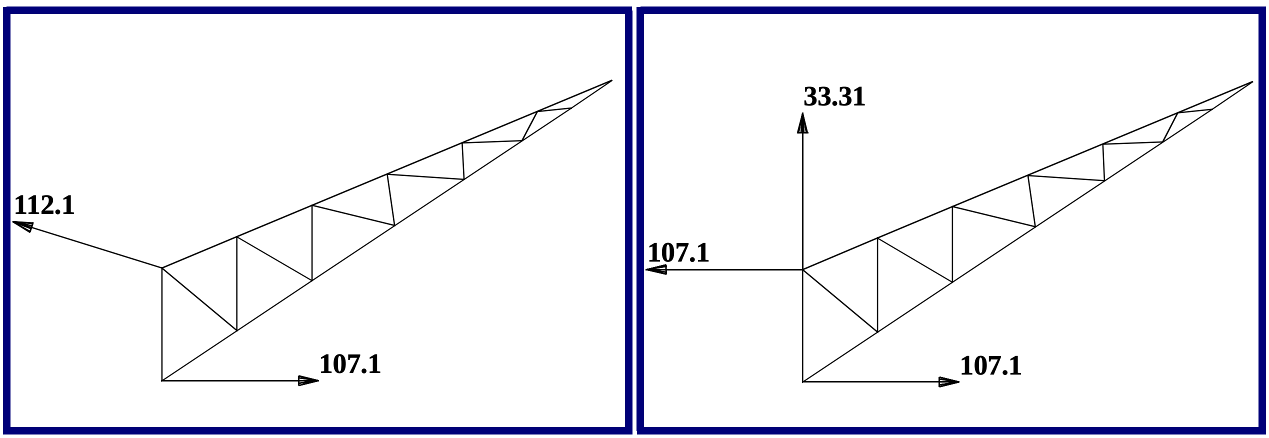
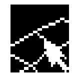


Figure 12-2 Example Plot of Reaction Loads at the Wall for the Continuous Truss Example. Both Resultant and Component Displays are Shown.

Display a Freebody Diagram

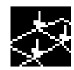
1. Set the Action to Create, the Object to Freebody and the Method to Loads.
2. Make sure the Select Results button icon is selected (Method = Load).

3. Select a Result Case (and a subcase if one exists).
4. Select the results type Freebody Loads from the Result Type listbox.
5. Press the Target Entities button icon, and select target entities as required. If this step is skipped the entire model, or whatever is currently posted will be used as the target entities. If the entire model is used, only applied and reaction loads will be displayed.



6. Press the Apply button to display the freebody diagram plot.

7. To change any display attributes of the vector plot, press the Display Attributes button icon. Press the Apply button to affect any changes.



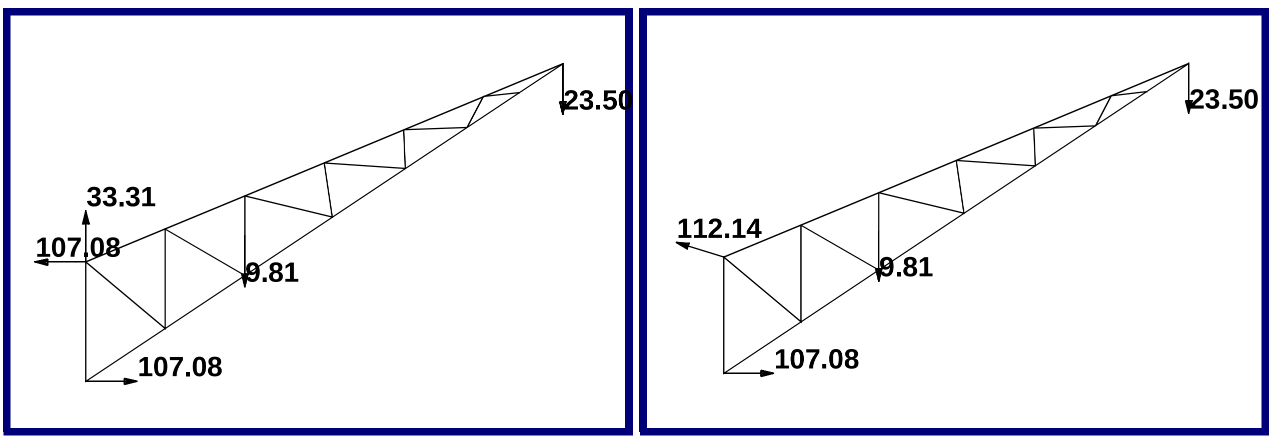
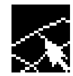


Figure 12-3 Example Plot of the Total Freebody Diagram for the Continuous Truss Example. Both Resultant and Component Displays are Shown.

View the Internal Loads at a Node or Nodes

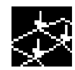
1. Set the Action to Create, the Object to Freebody and the Method to Loads.
2. Make sure the Select Results button icon is selected (Method = Load).

3. Select a Result Case (and a subcase if one exists).
4. Select the results type Internal Loads from the Result Type listbox.
5. Select the Target Entities button icon and select target entities as required. This is required for internal load display unless only a portion of the model is posted via groups.



6. Press the Apply button to display the internal loads plot.

7. To change any display attributes of the vector plot, press the Display Attributes button icon. Press the Apply button to affect any changes.



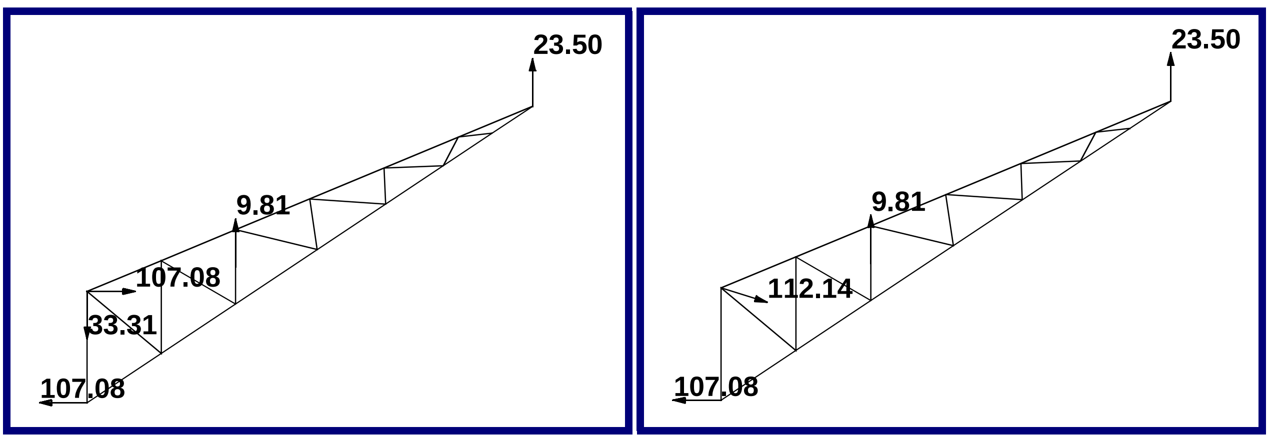


Figure 12-4 Example Plot of Internal Loads for the Continuous Truss Example. Both Resultant and Component Displays are Shown.

Create a Load Case Set for Use in a Subsequent Analysis

1. Set the Action to Create, the Object to Freebody and the Method to Loads.
2. Make sure the Select Results button icon is selected (Method = Load).

3. Select a Result Case (and a subcase if one exists).
4. Select the appropriate result from the Results Type listbox of which you would like to create a load set.
5. Press the Apply button to display the internal loads plot.

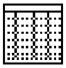
6. Press the Save Data button icon.

7. Enter a field name for the forces, a field name for the moments, a load set name or assign them to an existing load case. Or you can simply accept all the defaults.
8. Press the Apply button. Loads and fields will be created for whatever results type is currently displayed on the screen.

   The newly created fields and load set can be modified, viewed, deleted or otherwise manipulated from the Field and Loads/BC applications respectively. They can also be re-assigned to other load cases via the Loads/BC and Load Cases applications.

View the Vector Values on the Nodes Tabularly

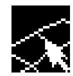
1. Display the desired plot graphically (freebody, applied, reaction, internal, other) as explained in previous examples.
2. Bring up the spreadsheet by pressing the Show Spreadsheet icon. The results (resultants and components) for the current plot will be displayed in the spreadsheet for the target entities.



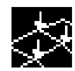
Display the Total Interface Load Across a Boundary

1. Set the Action to Create, the Object to Freebody and the Method to Interface.
2. Make sure the Select Results button icon is selected (Method = Interface).

3. Select the Result Case and subcase if necessary and Freebody Loads as the Result Type.
4. Select a node or location to act as the Summation Point or accept the origin as the default.
5. Select target entities. Press the Target Entities icon. The target entities must be all the nodes along an interface boundary for which you are interested in calculating the total load. In addition you must select the element on one side or the other of the nodes that define the interface line. This is a two step process. You must first select the nodes by changing the Select By pull-down menu to Node, select the nodes, then change the pull-down to Element, and select the elements.



6. Set the display attributes to show what you want (Forces/Moments, Resultants/Components) if desired. Press the Display Attributes button icon to do this.



7. Press the Apply button. A single component vector will be displayed on the screen.

In **Figure 12-5**, we have selected nodes 3 and 1 to be the interface line. There are element contributions from elements 3, 20, and 7, therefore these elements must be also selected as target entities. The summation point is node 2. A single vertical force of 23.5 and z-moment of 94.0 are the total interface loads experienced internally by cutting the model at this interface boundary.

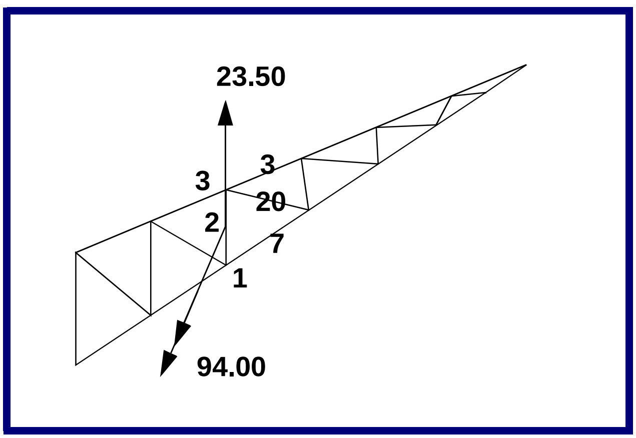
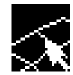


Figure 12-5 Example Plot of Applied Loads for the Continuous Truss Example

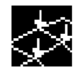
Create a Forced Displacement Boundary Condition

1. Set the Action to Create, the Object to Freebody and the Method to Displacement.
2. Make sure the Select Results button icon is selected (Method=Displacement).

3. Select the Result Case and subcase if necessary.
4. Select target entities if necessary. Press the Target Entities button icon. Only elements or groups containing elements need be selected. The default is use all elements posted to the current viewport.

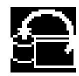


5. Set the display attributes to show what you want (Translational/Rotational, Resultants/Components) if desired. Press the Display Attributes button icon to do this.



6. Press the Apply button. Component or resultant vectors will be displayed on all exposed nodes of the freebody (along the free edges of the model) by default.

7. Press the Save Data button icon.



8. Enter a field name for the translational displacements, a field name for the rotational displacements, and a load set name or assign them to an existing load case. Or you may simply accept all the defaults.
9. Press the Apply button. Enforced displacement boundary conditions and fields will be created for the currently displayed plot on the screen.

   The newly created fields and load set can be modified, viewed, deleted or otherwise manipulated from the Field and Loads/BC applications respectively. They can also be re-assigned to other load cases via the Loads/BC and Load Cases applications.

CHAPTER 13

Insight

13.1 Overview of the Insight Application

The Insight application provides capabilities for interactive results postprocessing. Insight is object oriented, providing postprocessing tools which are created, displayed, and manipulated to obtain rapid *insight* into the nature of results data. The imaging is intended to provide graphics performance sufficient for real time manipulation. Performance will vary depending on hardware but consistency of functionality is maintained as much as possible across all supported display devices.

The Insight visualization tools remain in the database and provide the means for results manipulation and review in a consistent, easy to use manner. The available tools include Isosurfaces, Streamlines, Streamsurfaces, Contours, Tensors and others described in detail in this chapter.

Definitions

Tool Definitions. Insight provides thirteen different tools for results visualization. These tools allow you to graphically examine your analysis results using a variety of imaging techniques. Insight allows for the simultaneous display of multiple tools to aid in the understanding of interactions between results. This section will provide a basic understanding of the different tool types which you will create and manipulate using the Insight application.

The following table summarizes the tools available in Insight followed by a detailed description of each.

Isosurface	Surfaces of constant result or coordinate value.
Streamline	Tracing of a particle path (or paths) through a vector field.
Streamsurface	Surfaces generated along streamlines displaying a vector field's vorticity as a ribbon along a streamline.
Threshold	3-dimensional contour surfaces through solids.
Fringe	Contoured bands of color representing ranges of results value.
Contour	Lines of constant result value.
Element	Element-based polygon-fill or wireframe plot with color representing results values.
Tensor	Tensors are depicted at the result locations as three dimensional cubes or squares with vectors showing direct and shear components.
Vector	Vector plot of result components or resultants.
Marker	Colored scaled symbols representing scalar results.
Value	Annotation representing the actual result value displayed at the result location.
Deformation	Displays the model and posted tools in a deformed state.
Cursor	Displays result values at nodes/elements selected with the left mouse button and optionally loads these values into a spreadsheet.

Isosurfaces are surfaces of constant value. The constant value may be based upon either a scalar nodal-averaged result or a constant coordinate value. Coordinate-based isosurfaces may be defined in either the global cartesian (cid=0) system or in any alternate coordinate system (cartesian, cylindrical or spherical). An isosurface tool may be defined as a single surface at a constant value (isovalue) or a series of up to five surfaces uniformly distributed between a high and a low value. Other tools may be targeted at isosurface tools and they will update accordingly as the isosurfaces are manipulated in the **Isosurface Control** form. Isosurfaces may also be used to clip the displayed model. The portions of the model which are greater than and / or less than the isovalue(s) may each be displayed as None, Transparent, Free Edge, Wireframe, Shaded, or Hidden Line. These clipping attributes allow for displays such as a shaded model only where a result value exceeds a specified amount (e.g., display all the volume which exceeds the allowable stress).

Streamlines are a tracing or set of tracings of particle paths through a vector field. They are associated with nodal vector data. They emanate from nodes defined by a database entity called a rake. A rake is referred to by its name; a user-definable string identifier. Each rake defines a list of nodes or node pairs for stream entities. A streamline's color may be set to a fixed value or mapped to display the magnitude of the vector field along the stream path. Other tools may be targeted at streamlines. These tools will be visualized along the streamline(s) and will update as the streamlines are manipulated.

Streamsurfaces are surfaces generated as a ribbon along a streamline. They are associated with nodal vector data. Other tools may be targeted at a streamsurface or set of streamsurfaces defined by a streamsurface tool. Streamsurfaces permit you to visualize vortices in fluid flow problems.

Threshold tools are 3-dimensional contour surfaces. They are associated with nodal scalar data. As contour lines depict the lines of constant value on surfaces, thresholds depict the surfaces of constant value through solids. Each individual threshold within a threshold tool is an isosurface. Unlike the isosurface tool, thresholds are not a targettable tool nor can they be used to define a clipping surface. Thresholds are controlled by the current range levels and spectrum definition for color, number of displayed surfaces, and surface values.

Fringe tools map color to surfaces or edges based on the result data defined for the tool. Fringes are developed from nodal-averaged scalar values. Fringes may be targeted at the model's element faces or free edges or to previously created isosurfaces, streamlines, and streamsurfaces. The fringe tool will supersede all existing or default color definition for the entity at which the fringe is targeted. The fringe tool also supports transparency.

Contour tools display lines of constant results value on surfaces. As with thresholds, the current range and spectrum definition control the number of contours, contour location, and contour color. Contours are based on nodal-averaged scalar values. The contour tool displays lines of constant results values whereas the fringe tool displays ranges or bands of results value. Contour line displays are useful to create a hardcopy image of the Insight viewport on a black and white hardcopy device.

Element tools display elements as polygon fills or wireframe with color mapped to each element by a single (assigned) results value. Element tools are based on element centroidal scalar data. Element tools may be targeted at model elements or isosurfaces. Isosurface-targeted element tools will display the elements which traverse the targeted isosurface(s).

Tensor tools display an iconic representation of a symmetric stress tensor. Tensors may be oriented in the axes of principal stress or the tensor's defined coordinate system. Tensors may be defined by element- or nodal-based tensor data. Nodal tensors are mapped from element tensors and are used when a tensor tool is targeted at other tools. Tensors may be targeted at nodal- and element-based model features as well isosurfaces, and streamline tools.

Vector tools display nodal or element based vector data as component or resultant vectors. Vectors may be colored and scaled based on magnitude and may be targeted at model features, isosurfaces and streamlines.

Marker tools display nodal or element based scalar results as icons at the result locations. The icons may be scaled and colored based on the assigned data. Icons shapes may be a triangle, square, diamond, hourglass, cross, circle, dot, point, sphere, or cube. Markers may be targeted at model features, isosurfaces and streamlines.

Value tools display the data values of nodal- or element-based scalar results as text at the results locations. The value text color may be fixed to a value or mapped based on the results data, the current range and spectrum definitions. Values may be targeted at model features, isosurfaces, and streamlines.

Deformation tools are used to display the current model and posted tools in a deformed state. Only one deformation tool may be posted at a time and the tool will modify the display of the posted model and all posted tools. An optional display of an undeformed model is controlled as an attribute of the deformation tool. The targeting of deformation tools is not controllable since they will have global effects. Deformations may be used to display any nodal vector data as a deformation.

Cursor tools are used to display result values and load result values into a spreadsheet using the cursor. Pressing the left mouse button will select the nearest node or element and display the value of the assigned result on the screen. Holding the mouse button down and dragging it will interactively move the displayed value or id to represent the current entity closest to the cursor. The selected entity ids and results will also be added to the cursor tool spreadsheet for review, sorting and output to a text file.

The Insight application provides the means of Creating, Modifying, Deleting, Posting and Unposting these tools as well as means for dynamically manipulating these tools for interactive results imaging. Each tool created has assigned attributes which determine its characteristics. All tools have the following attributes.

Name	A unique user-definable string descriptor to identify the tool.
Type	One of the thirteen tool types described above.
Result(s)	A list of load cases and a corresponding result which the tool is to display. A coordinate based isosurface would refer to a coordinate frame and axis versus a result.
Target	Where the tool is to be displayed. This is either on a model feature or on a targettable tool (isosurfaces, streamlines, streamsurfaces).
Display Attributes	Each tool type has specific settings to control how the tool type is to be displayed.
Animation Attributes	Attributes to describe whether the tool is to be animated and how the results are to be mapped to animation frames.
Posting Status	Each tool is either Posted (displayed) or Unposted (not displayed).

Tool Targets. Insight tools may be targeted at selected model entities or selected tools. The model based targets may be defined by a list of posted groups or by all posted entities. The model entities and tools which may act as targets for Insight tools are described:

Elements will generally be displayed through the model. For example, an isosurface targeted at the model's elements will be displayed as a plane passing through the model orthogonal to some coordinate system axis.

Free faces describe those element faces common to only one element. This includes faces lining the outside surface of a model or those inside surfaces exposed to internal voids. Free faces are appropriate targets for displays such as fringe plots which are normally displayed on the surface of the model or on a cutting plane through the model.

All Faces display results on each face of each element.

Free Edges display results on edges common to only one element. Use this target type when you wish to display results on the same edges which are used to draw the model when Free Edge is selected as the display method in the Preferences-Insight... form.

All Edges display results on all element edges. Using this target selection allows you to map results onto a wireframe representation of the model.

Nodes display the selected tool's result at each nodal location of the model. Tensor, vector, marker and value tools may all be displayed at nodal locations.

Corners display the selected tool's result at nodes which are common to only one element. Tensor, vector, marker and value tools may all be displayed at corner locations.

Isosurfaces are selectable tools. The isosurface can be used as a target for fringes, contours, tensors, elements, vectors, markers, and values, as well as being a visualization tool by itself. A tool posted on an isosurface will be displayed on its calculated surface(s) of constant value (coordinate or result).

Streamlines are selectable tools. Fringes, tensors, vectors, markers and values may be targeted to streamline tools.

Streamsurfaces are selectable tools for fringes and contours.

The following table summarizes the valid targets for all tools.

TOOL	Element	Free Faces	All Faces	Free Edges	All Edges	Nodes	Corners	Isosurface	Streamline	Streamsurface
Isosurface	•									
Streamline	•									
Streamsurface	•									
Threshold	•									
Fringe		•	•	•	•			•	•	•
Contour		•	•					•		•

Element	•							•		
Tensor	•	•		•		•	•	•	•	
Vector	•	•		•		•	•	•	•	
Marker	•	•		•		•	•	•	•	
Value	•	•		•		•	•	•	•	
Deformation						•				
Cursor	•					•				

Other Definitions

Named Posting	A user defined grouping of tools to be displayed simultaneously.
Post	To display the entity.
Unpost	To remove the entity from the display.
Range	A MSC.Patran database entity defined by a number of levels and (1-12 for Insight) and a series of threshold values for each level within a range. Ranges are used to map spectrum colors to results values. A spreadsheet form is available to control range levels.
Viewport Range	The range entity currently assigned to the Insight viewport.
Auto Range	A range which is not a database entity but is automatically calculated for a tool based on the results values. This type of range may be manipulated dynamically to change the range extremes and the number of intermediate levels.
Rake	A database entity which describes a list of nodes. A rake entity is needed for Streamline and Streamsurface tools.
Reduced Rendering	A graphical rendering style available to eliminate surface rendering while performing various tasks with Insight. Reduced Rendering will display only vector outlines for all filled faces. This is useful for manipulating large models or for increasing speed on machines without hardware shading. This capability is controlled from the Insight Preferences form.

Capabilities and Limitations

Insight provides the capabilities of the following actions, and the capability to manipulate results visualization tools as well as view the finite element model.

- Create
- Modify
- Delete
- Post/Unpost Tools

Dynamic control is provided for moving isosurfaces through the model, manipulating the color/range assignment for tools, and for controlling and creating animations of static and transient results. The forms controlling this functionality are described in **Insight Control** (p. 265).

Results are selected from the database and assigned to tools using simple forms. Results transformations are provided to derive scalars from vectors and tensors as well as to derive vectors from tensors. This allows for a wide variety of visualization tools to be used with all of the available results. Result transformations include component extractions as well as calculations such as magnitude, vonMises stress, or maximum principal vector (see **Result Options** (p. 250)).

The Insight imaging routines are optimized for graphical speed. You should be aware of the following limitations:

- With the exception of element targeted fringe tools, results will be displayed as nodal averaged or element centroidal data. Other result types will be averaged prior to display.
- The combination of results and the manipulation of loadcases for display is not available in Insight. Insight *will only access results* currently residing in the database. Combined and derived results cases can be created in the Results application within MSC.Patran. Refer to **Create Results** (Ch. 11), for more information.

13.2 Using Insight

The Insight application is based on the creation and manipulation of results visualization tools. The first action to be performed using Insight is to create a tool. This is done using the **Create** (p. 245) action on the **Insight Imaging** (p. 244) form.

Creating a tool involves five basic steps:

1. Select a tool type from the**Insight Imaging** (p. 244) form.
2. Select a result from the **Results Selection** (p. 248) form, or (for an isosurface) select a coordinate from the **Isosurface Coordinate Selection** (p. 253) form. For an isosurface, this step requires both selecting result data or a coordinate axis and specifying an initial value for the data. An isosurface tool can either be tied to a single value or span a range of values using two, three, four, or five equally surfaces.
3. Set the tool attributes using the applicable **Tool Attributes** (p. 254) form.
4. Select the target for the tool from the **Tool Targets** (p. 239) pull-down menu in the Insight Imaging form.
5. Press the Apply button on the Insight Imaging form. The tool will be displayed.

Once a tool has been created, it may be modified using the **Modify** (p. 246) action on the Insight Imaging form. To modify a tool, first select the tool from the Existing Tool listbox, modify as needed, and click the Apply button to update the display.

Multiple tools may be created and posted simultaneously. One of the most common uses of multiple tools is to first create an isosurface, then create a fringe tool targeted to the isosurface. This tool combination will display a surface of constant coordinate or scalar result which is colored with a fringe contour of some other result. The isosurface could, for example, display a surface where vonMises Stress = 10,000 psi, while the surface is displaying a fringe of maximum principal strain. Or, an isosurface of constant global X could be created and a fringe of temperature created and posted to it.The combinations of superimposed tools are virtually endless. All displayed tools are updated as their targeted tools are moved or modified.

You can display a deformed shape with a tool as well. Creating and posting a deformation tool will display all posted entities in a deformed state.

Once a tool or set of tools has been created and posted, capabilities for moving isosurfaces, creating animations, manipulating the range and color assignment, and posting and unposting existing tools are accessible through the **Insight Control** (p. 265) menu on the MSC.Patran main form. Tools already created may be posted or unposted using the **Post/Unpost Tools** (p. 265) form, which is accessible by selecting the Post / Unpost Tools option in the Insight Control menu.

Posted isosurfaces may be moved by manipulating the value(s) defining the tools. This capability is provided through the **Isosurface Control** (p. 267) form, which is accessible by selecting the Isosurface Control option in the Insight Control menu. A posted isosurface is selected from the listbox as the active surface to control. The surface can be moved using the slidebar or by entering a value in the databox. The Start value applies to all isosurfaces. The End value is used if the isosurface tool was created with multiple surfaces (a maximum of five is allowed). The speed at which an isosurface will respond to the slidebar is dependent on the size of the model, the speed of the display device, and the current slidebar resolution which is set in the **Insight Preferences Form** (p. 278) form. The Insight Preferences form is accessible by selecting the Insight option in the Preferences menu on the MSC.Patran main form.

Selecting and manipulating the active range tool is done in the **Range Control** (p. 268) form, which is accessible by selecting the Range Control option in the Insight Control menu. Although many tools may be displayed simultaneously, only one spectrum and range is available for display. Any one of the posted tools may be selected as being the active range tool; i.e., the tool which is associated with the displayed range and spectrum. The other tools posted may be color coded to show a gradation of results data but may not necessarily be tied to the active range.

The ranges and color assignment for the active range tool can be defined using two methods; Auto and Viewport. An option on the **Range Control** (p. 268) form is used to select the desired method. The Auto method is used if the range is to be automatically calculated based on the result assigned to the active range tool, or if you wish to dynamically manipulate the range values or number of levels using the slidebars on the Range Control form. The Viewport range method uses the MSC.Patran range assigned to the Insight viewport. MSC.Patran ranges are created and manipulated using **Display>Ranges** (p. 307) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*, which is accessible by selecting the Ranges option in the Display menu on the MSC.Patran main form. The Viewport range provides greater user control for range levels but cannot be used for dynamic range manipulation.

The other posted tools can have color assigned using three methods: Auto, Viewport, and Active. Auto will automatically show color spanning the current spectrum for each tool regardless of the current range levels. This method is good for understanding qualitative result levels for all tools, but colors assigned will not correspond to the displayed spectrum levels. Viewport will assign colors based on the Viewport range. The Active tool range method will guarantee that all tools displayed map to color range levels as displayed. Adjusting the slidebars on the Range Control form will dynamically update the display as the values are modified.

Animations are setup and controlled using either the **Animation Control** (p. 269) form, or the Modal Animation form which are accessible from the Insight Control menu.

The Modal Animation form provides for rapid access to a modal animation display with minimal user interaction. You need only to select a vector result and press the animate button. A wireframe display of the selected result’s deformed shape will be animated using a sinusoidal variation of scale.

The Animation Control form provides access to all of the animation functionality provided by Insight. Its **Animation Setup** (p. 270) form allows for the modification of the posted tools as needed to set the animation attributes. All posted tools will be displayed during an animation but only those tools with the animation enabled will change from frame to frame. Animation frames are created by mapping the tool results onto a requested number of animation frames. Each posted tool enabled for animation may be animated using a global variable for transient results (more than one load case assigned to the tool) or using a ramp or sinusoid scaling for static results (only one loadcase used). Linear interpolation is used for mapping transient results to animation frames. Once an animation has been created and is being displayed, the Animation Control form provides capabilities for cycling and stepping through the animation sequence.

13.3 Insight Forms

The Insight forms used to create and modify imaging tools are described in detail in the sections titled **Insight Imaging** (p. 244) and **Tool Attributes** (p. 254). Capabilities for control and manipulation of the Insight viewport and tools are accessed through the forms described in the sections **Insight Control** (p. 265) and **Global Settings** (p. 276). These forms are accessible through the Insight pull-down menu on the MSC.Patran main form.

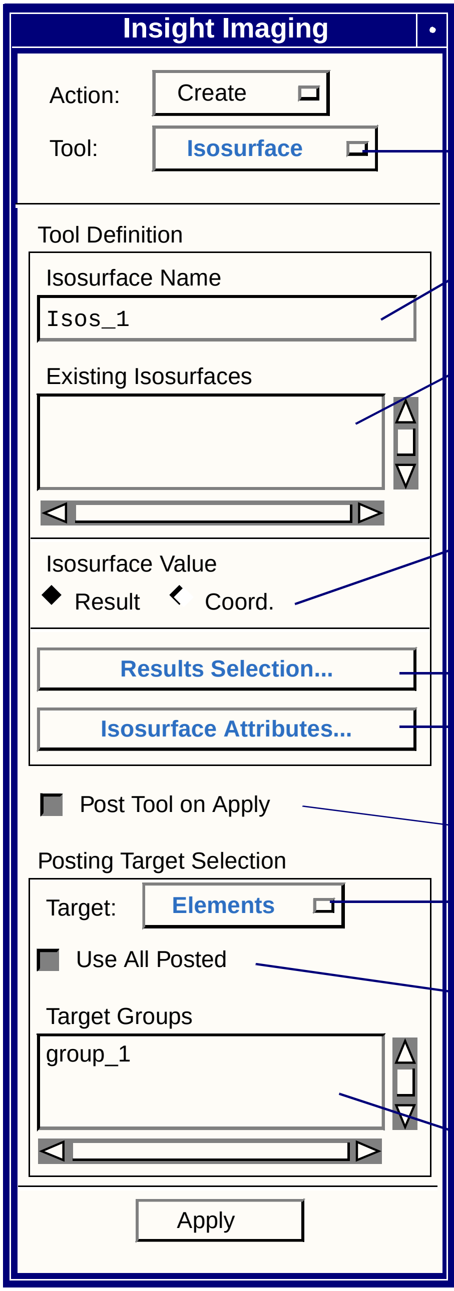
Insight Imaging

The Insight Imaging application form provides the capabilities to Create, Modify, and Delete tools. Additional forms for selecting results for tools (or coordinate axes for isosurfaces) and for controlling display attributes are accessed from the Insight Imaging form during the Create and Modify actions.

❑ Create	This action is used to create Insight visualization tools.
❑ Modify	This action is used to modify existing Insight visualization tools.
❑ Delete	This action is used to delete existing Insight visualization tools.
❑ Results Attributes	This form is accessed from the Results Selection button on the Insight Imaging form. This form is used to create a result based tool.
❑ Animation Attributes	This form is accessed either from the Select Results form or the Animation Setup form. Animation attributes are assigned to a tool as part of its definition. Each tool may have its own unique animation attributes to control how each tool is handled during animations.
❑ Result Options	This form is accessed from the Results Selection form after a Load case(s) and result is selected. This form is used to define more details regarding the selected result such as selecting a transformation method if the result must be transformed, selecting a layer if multi-layered results data exists, and selecting a default loadcase if multiple load cases are selected.
❑ Isovalue Setup	This form is accessed from the Results Selection form for result based isosurfaces. The initial result value(s) for the isosurface and the number of surfaces for the isosurface tool are defined through this form.
❑ Isosurface Attributes	This form is used instead of the Results Selection form for coordinate based isosurfaces. An isosurface can be tied to any axis of a user defined coordinate frame or the global system (frame 0). The initial coordinate value(s) and number of surfaces for the isosurface tool are also specified in this form.
❑ Tool Attributes	There is a unique Attributes form for each tool type. Each tool may have unique display attributes assigned. They are controlled from the Attributes form associated with the tool type.

Create

Select Create from the Action pull-down menu to create new visualization tools.



Select a tool type. See **Tool Attributes** (p. 254).

A default name is provided. If desired, enter a new name for the tool.

Existing tools of the same type will be listed here. Select a tool from the list to copy its attributes to the new tool, if desired.

Select the Result or Coordinate option for an isosurface value. Select Scalar, Vector or Tensor for Cursor Tool Result type. These options will not be displayed for other tool types.

Displays the **Results Selection** (p. 248) form.

Displays the applicable tool attributes form. See **Isosurface Attributes Form** (p. 255).

Select this option to post the tool, if desired.

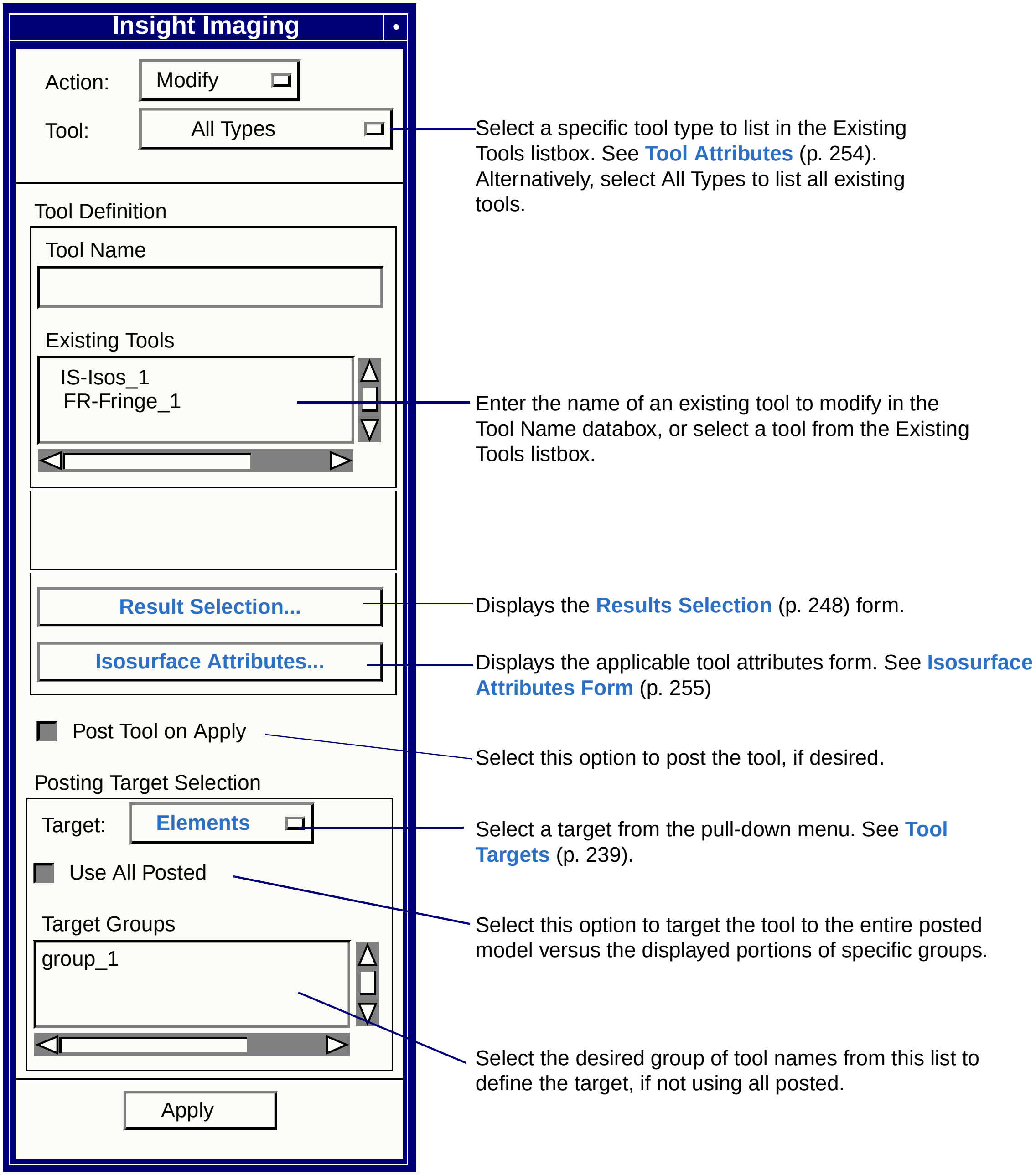
Select a target from the pull-down menu. See **Tool Targets** (p. 239).

Select this option to target the tool to the entire posted model versus the displayed portions of specific groups.

Select the desired group of tool names from this list to define the target, if not using all posted.

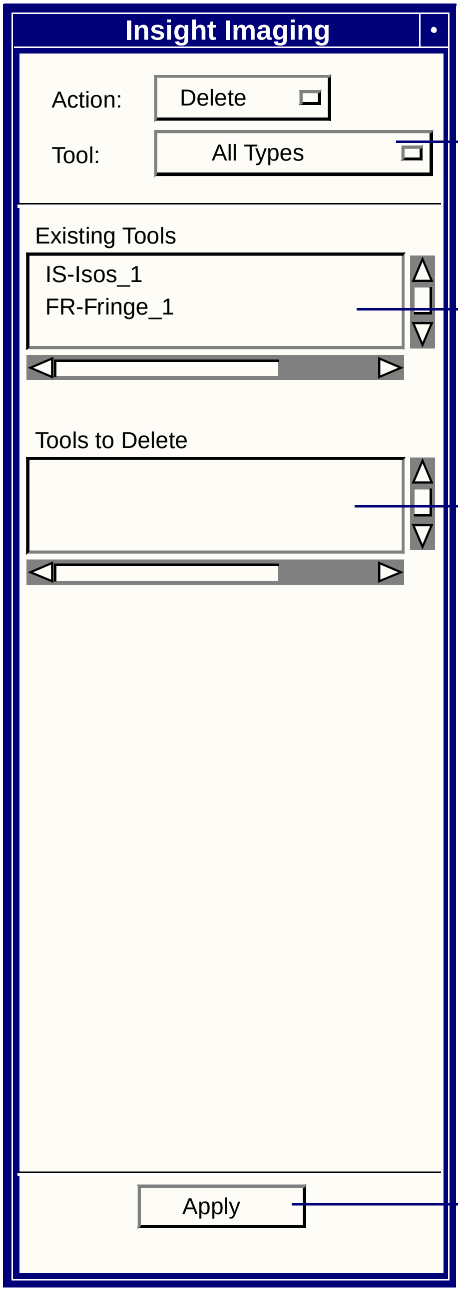
Modify

Select Modify from the Action pull-down menu to modify existing visualization tools.



Delete

Select Delete from the Action pull-down menu to delete existing tools.



Select a specific tool type to list in the Existing Tools listbox. See **Tool Attributes** (p. 254). Alternatively, select All Types to list all existing tools.

Select tools from the Existing Tools listbox. They will be added to the Tools to Delete listbox.

Selecting tools displayed in the Tools to Delete listbox will return them to the existing Tools listbox.

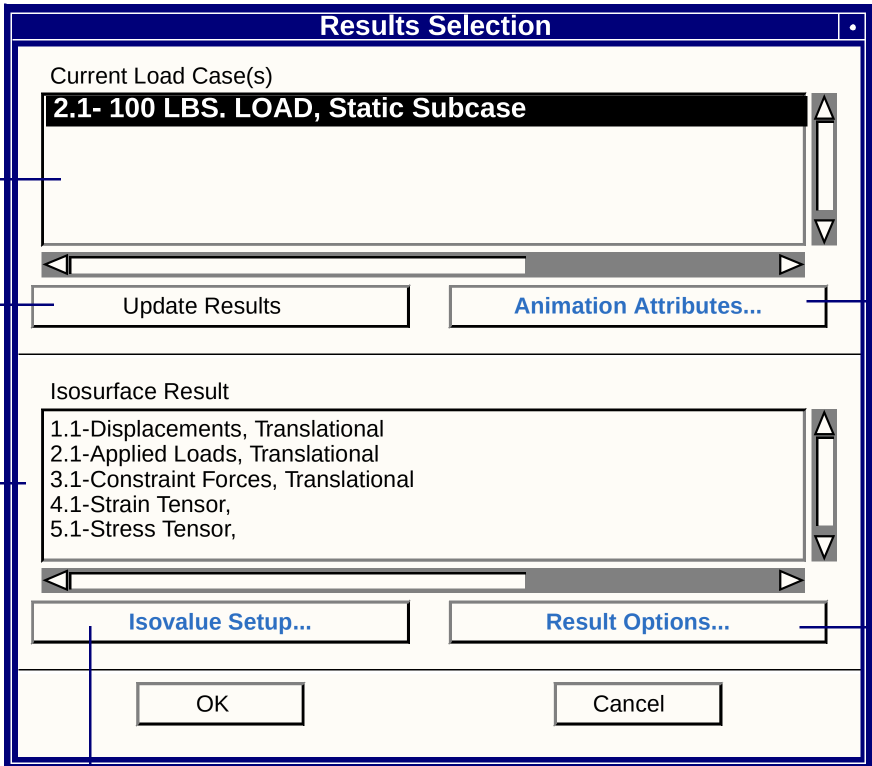
Deletes the tools listed in the Tools to Delete listbox.

Results Selection

The following form appears when the Results Selection button is selected on the Insight Imaging form when Create or Modify is the selected Action.

Updates the results list based on the selected loadcase(s).

Select the Load Case(s) from the list to be associated with the tool. If multiple load cases are selected, a default load case is assigned using the Results Options Form. Only the default load case results will be displayed, except during an animation.



Displays **Animation Attributes** (p. 249).

Displays **Result Options Form** (p. 251).

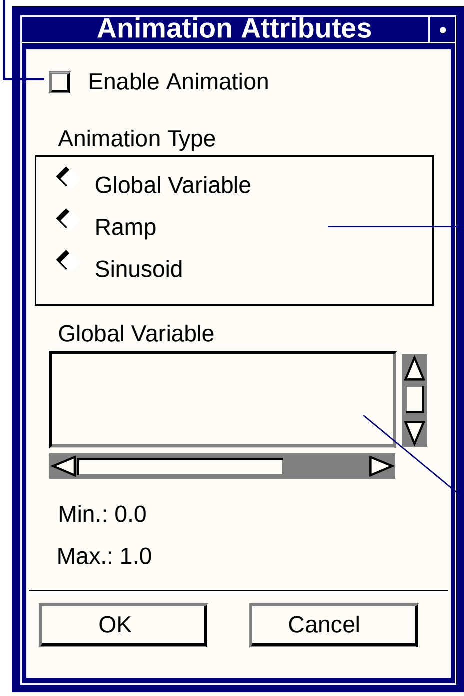
Displays **Isovalue Setup** (p. 252) to specify an initial isovalue and number of surfaces for an isosurface tool type.

Select a desired result from this list. Only those results which are of the same type as the selected tool, or those that can be transformed to the tool's result type, are listed. For example, a streamline tool displays vector results, therefore only vector and tensor result types will be listed. The transformation method for different type results is specified using the Result Options form. If the result has multiple layers defined, the layer to display is also selected from the Result Options form.

Animation Attributes

The following subordinate form appears when the Animation Attributes button is selected on the Results Selection form.

Select this option to enable animation for the tool. All posted tools will be displayed during an animation, but only those that have been enabled will animate (i.e., change from frame-to-frame).



Select an animation type.

Global Variable is used to animate transient results through the load cases selected for the tool. This selection is only available for tools with more than one load case.

Ramp is used to ramp the default load case result or coordinate isovalue multiplier from 0.0 to 1.0.

Sinusoid is used to vary the default load case result or coordinate isovalue multiplier sinusoidally from -1.0 to 1.0.

If the animation type selected is Global Variable, a variable must be selected. All global variables defined for the selected load cases will be listed as well as LOAD CASE INDEX which may be selected to animate only the tool, versus the selected load cases.

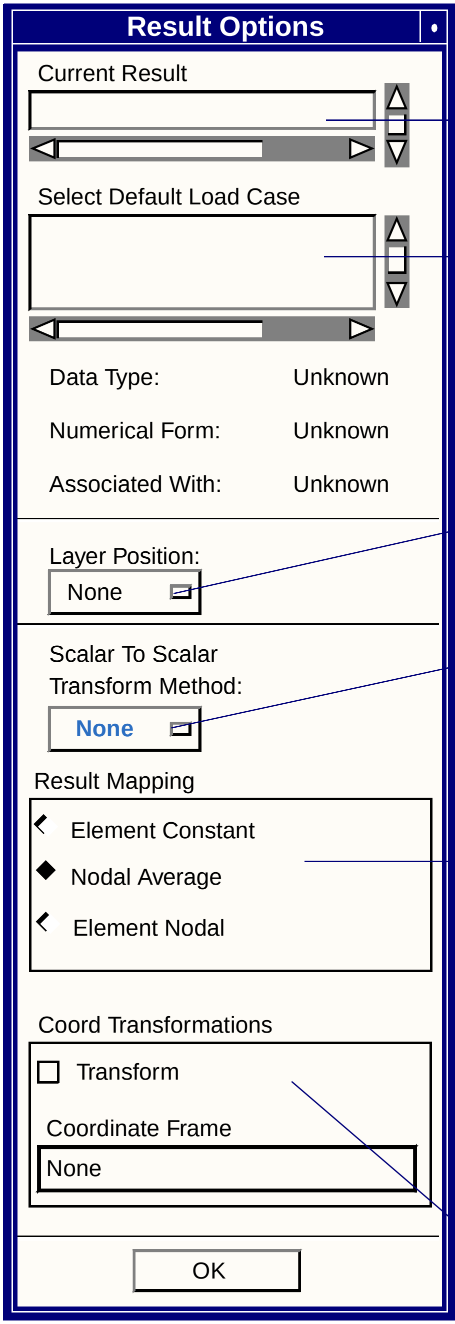
Result Options

The following table provides additional definitions for the selected result transformation method.

Transform Type	Transform Method	Description
Scalar to Scalar **Vector to Vector** **Tensor to Tensor**	**None**	No transformation is used if the result data type matches the tool's data type.
Vector to Scalar	**Magnitude**	Vector magnitude.
	X Component	1st vector component.
	Y Component	2nd vector component.
	Z Component	3rd vector component.
Tensor to Scalar	**XX Component**	XX tensor component.
	YY Component	YY tensor component.
	ZZ Component	ZZ tensor component.
	XY Component	XY tensor component.
	YZ Component	YZ tensor component.
	ZX Component	ZX tensor component.
	Min Principal	Calculated minimum principal magnitude.
	Mid Principal	Calculated mid principal magnitude.
	Max Principal	Calculated maximum principal magnitude.
	Min Deviatoric	Minimum principal magnitude of the deviatoric.
	Mid Deviatoric	Mid principal magnitude of the deviatoric.
	Max Deviatoric	Maximum principal magnitude of the deviatoric.
	Mean	Calculated mean of the tensor.
	von Mises	Calculated effective stress using von Mises criterion.
	Determinant	Determinant of the symmetric tensor matrix.
	Max Shear	Calculated maximum shear magnitude.
	Octahedral	Calculated Octahedral shear stress.
Tensor to Vector	**Min Principal**	Calculated minimum principal vector.
	Mid Principal	Calculated mid principal vector.
	Max Principal	Calculated maximum principal vector.

Result Options Form

This subordinate form appears when the Result Options button is selected on the Results Selection form.



Displays the current selected result.

Lists all of the selected load cases. A default load case *must* be selected.

If multiple layers are defined, select a desired layer position from the pull-down menu.

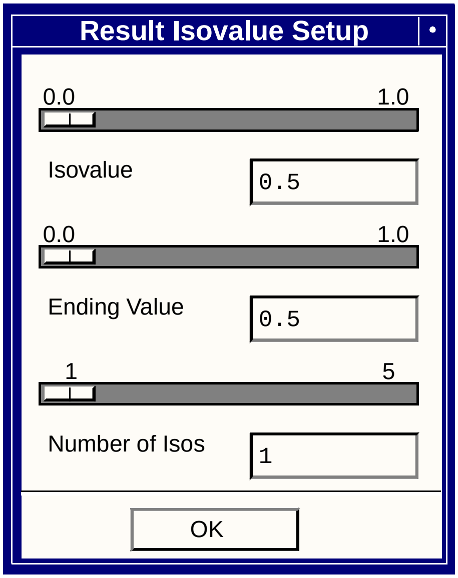
Select a desired transform method if the result data type does not match the tool data type.

Results associated with elements can be handled in one of three ways. Element Constant will average all the results of the specified type associated with each element to obtain a single value. This value will then be used for any subsequent operations such as obtaining nodal averaged values for fringe or isosurface display. Nodal Average will obtain a value at the nodes by averaging the extrapolated nodal values for all of the elements associated the each node. Element Nodal will preserve multiple results at element nodes until the final display. This method will show discontinuity for element targeted fringe tools and will not average results for entities not displayed or targeted by a tool.

To specify an alternate coordinate system for display of vector or tensor results, depress the Transform toggle and enter a coordinate frame in the databox using the list format. For example, Coord 1.

Isovalue Setup

The following subordinate form appears when the Isovalue Setup button is selected on the Results Selection form.



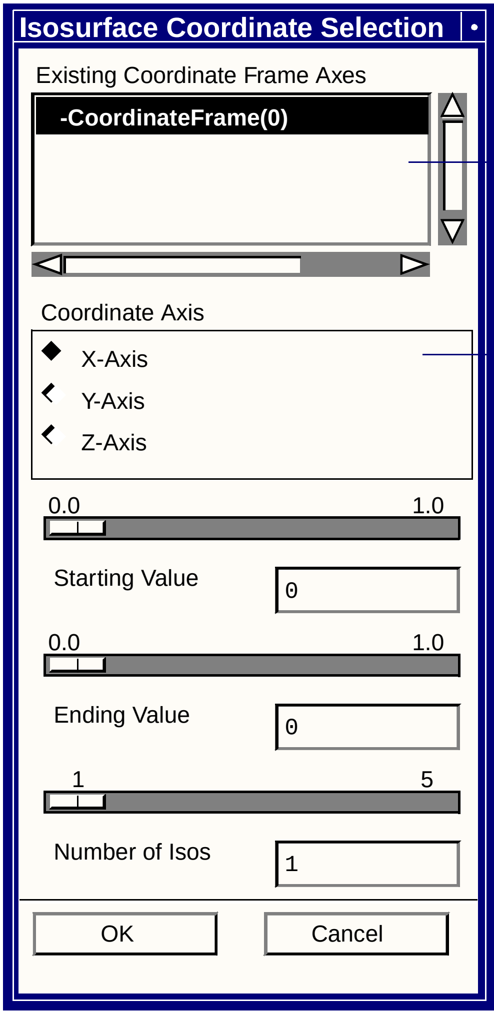
Adjust the slidebar to set an initial value for the isosurface, or enter a value in the databox. The valid range will be automatically calculated based on the range of the current result.

If the number of isosurfaces is greater than one, adjust the slidebar to set an end value for the isosurface, or enter a value in the databox.

Adjust the slidebar to set a value for the number of isosurfaces to be displayed, or enter a value in the databox. The surfaces will be evenly spaced from the starting to ending value. If only one surface is used, the starting and ending value will be equal.

Isosurface Coordinate Selection

The following subordinate form appears when the Isosurface Coordinate Selection button is selected on the Insight Imaging form.



Select a coordinate frame from the list of existing frames. Any rectangular, cylindrical, or spherical coordinate frame is valid. Frame 0 is the global system.

Select an axis for the selected coordinate frame.

Adjust the slidebar to set an initial value for the isosurface, or enter a value in the databox. The valid range will be automatically calculated based on the range of the selected coordinate frame/axis for the posted groups.

If the number of isosurfaces is greater than one, adjust the slidebar to set an end value for the isosurface, or enter a value in the databox.

Adjust the slidebar to set a value for the number of isosurfaces to be displayed, or enter a value in the databox. The surfaces will be evenly spaced from the starting to ending value. If only one surface is used, the starting and ending value will be equal.

Tool Attributes

The following forms are accessible from the Insight Imaging form when the applicable Attributes button is selected. Each tool type has a unique Attributes form to set the graphical display controls specific to the tool.

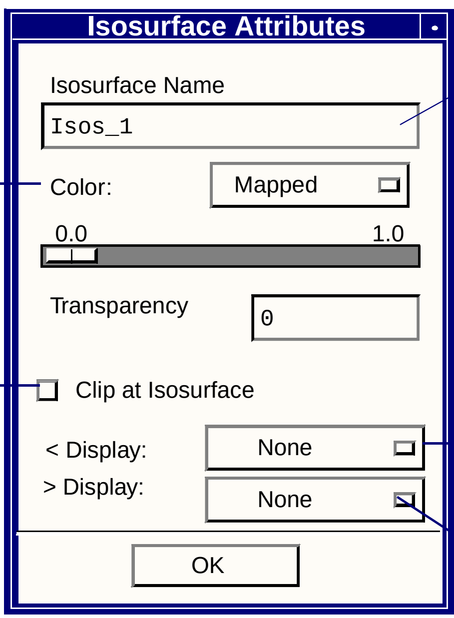
❑ Isosurface Attributes	Surfaces of constant result or coordinate value.
❑ Streamline Attributes	Tracing of a particle path (or paths) through a vector field.
❑ Streamsurface Attributes	Surfaces generated along streamlines displaying a vector fields vorticity as a ribbon along streamlines.
❑ Threshold Attributes	Three dimensional contour surfaces through solids.
❑ Fringe Attributes	Contoured bands of color representing ranges of results value.
❑ Contour Attributes	Lines of constant result value.
❑ Element Attributes	Element-based polygon-fill or wireframe plot with color representing results values.
❑ Tensor Attributes	Tensors are depicted at the result locations as three dimensional cubes or squares with vectors showing direct and shear components.
❑ Vector Attributes	Vector plot of results components or resultants.
❑ Marker Attributes	Colored scaled symbols representing scalar results.
❑ Value Attributes	Annotation representing the actual result value displayed at the result locations.
❑ Deformation Attributes	Displays the model and posted tools in a deformed state.
❑ Cursor Attributes	Assigns a result to the mouse for screen and spreadsheet display.

Isosurface Attributes

None	Display nothing on this portion of the clipped model.
Free Edge	Display the clipped model as a wireframe image with inter-element boundaries visually suppressed. Shows only those edges belonging to only one element.
Wireframe	Display the clipped model as a wireframe image including any inter-element boundaries.
Shaded	Display the clipped model as a solid-shaded image.
Transparent	Display the clipped model as a transparent, solid-shaded image whose degree of transparency is specified by the Model Transparency slidebar on the Insight Preferences form.
Hidden Line	Display the clipped model as a hidden line plot by suppressing the rendering of those edges which would not normally be visible on a solid object.

Isosurface Attributes Form

Select a color for the isosurface. Mapped will map the current result or coordinate range to the spectrum for color assignment.



A default name for the isosurface is provided. Enter a new name, if desired.

Adjust the slidebar to set a value for the transparency for the isosurface, or enter a value in the databox. A value of 0.0 is opaque, 1.0 is completely transparent. A completely transparent isosurface is often useful as a target for other tools.

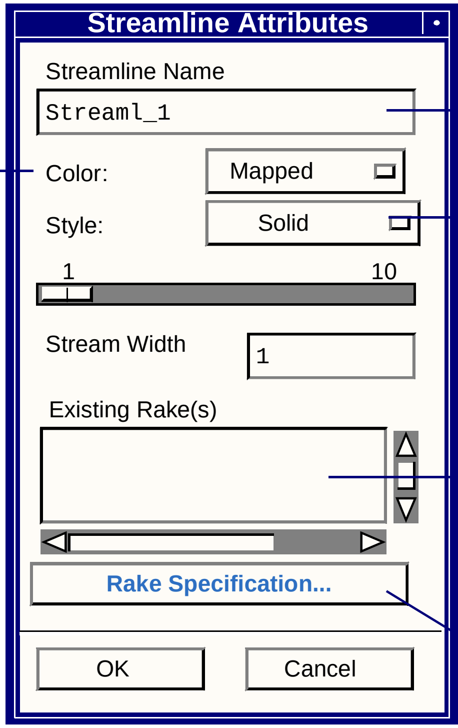
Select a display method for all of the model which is less than the start isovalue.

Select a display method for all of the model which is greater than the end isovalue.

Select this option to define the isosurface as clipping. A clipping isosurface will display the model in the specified methods on either side of the isosurface. Setting different display methods on either side of a clipping isosurface will provide greater visualization of where specified thresholds are exceeded.

Streamline Attributes

Select a color for the streamline. Mapped will map the current result to the spectrum for color assignment.



A default name for the streamline is provided. Enter a new name, if desired.

Select a streamline style from the pull-down menu.

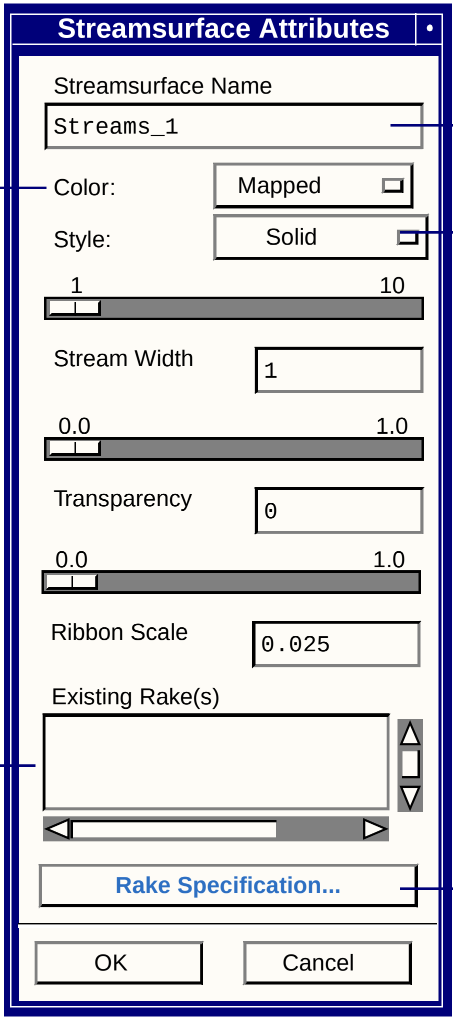
Adjust the slidebar to set a value for the stream width multiplier, or enter a value in the databox.

Existing rakes will appear in the listbox. A rake MUST be selected for a streamline tool. If no rakes exists, create one using the Rake Specification form.

Displays **Rake Specification** (p. 272).

Streamsurface Attributes

Select a color for the streamsurface. Mapped will map the current result to the spectrum for color assignment.



A default name for the streamsurface is provided. Enter a new name, if desired.

Select a streamsurface style from the pull-down menu.

Adjust the slidebar to set a value for the stream width multiplier, or enter a value in the databox.

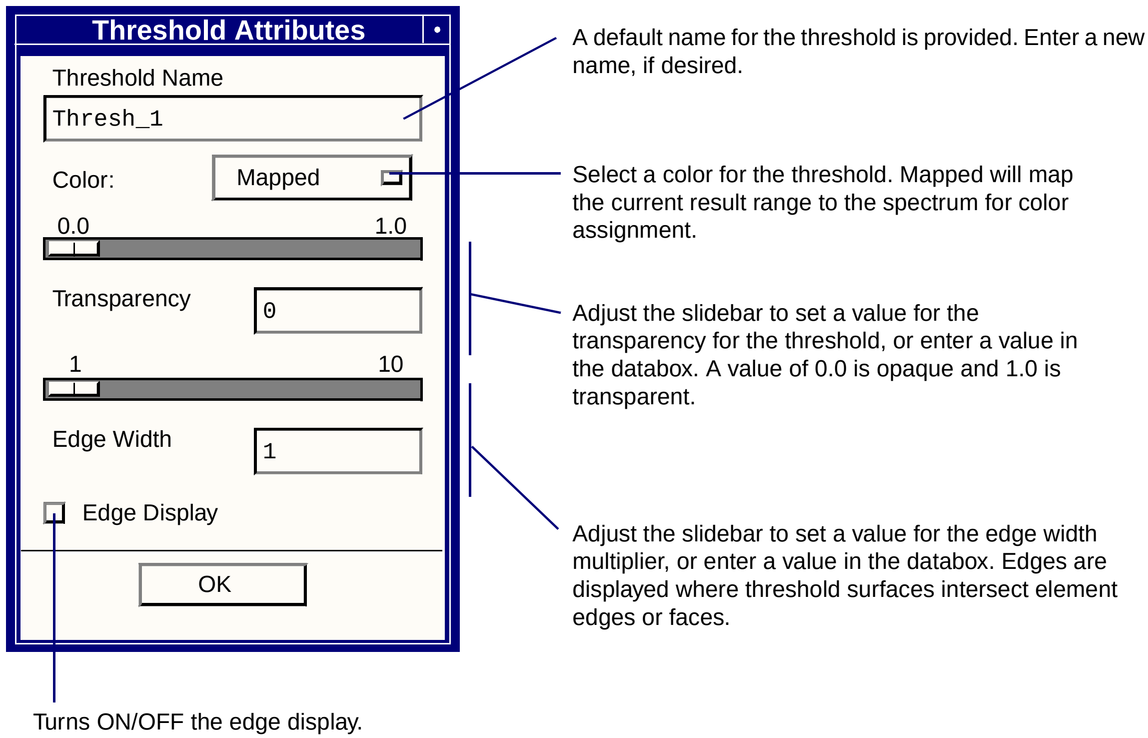
Adjust the slidebar to set a value for the transparency for the streamsurface, or enter a value in the databox. A value of 0.0 is opaque, 1.0 is transparent.

Adjust the slidebar to set a value for the ribbon width as a screen fraction, or enter a value in the databox.

Displays **Rake Specification** (p. 272).

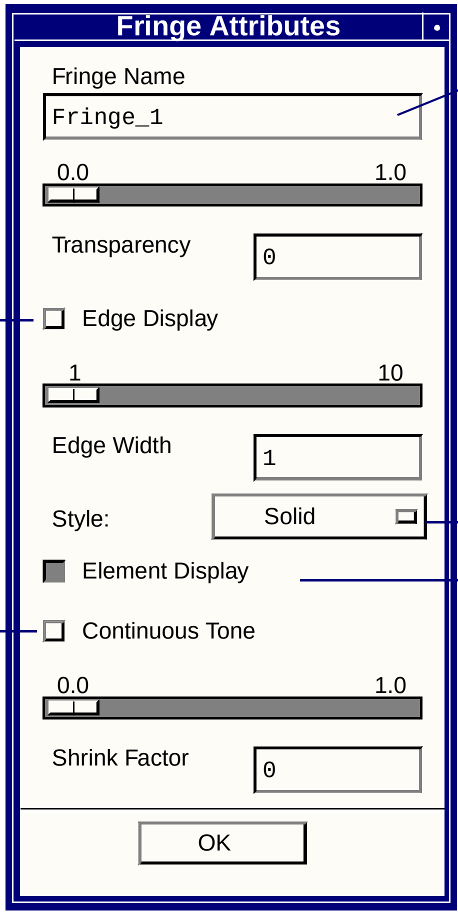
Existing rakes will appear in the listbox. A rake MUST be selected for a streamsurface tool. If no rakes exists, create one using the Rake Specification form.

Threshold Attributes



Fringe Attributes

Turns ON/OFF the contour edge display. Contour edges are displayed at the current range thresholds.



A default name for the fringe is provided. Enter a new name, if desired.

Adjust the slidebar to set a value for the transparency for the fringe, or enter a value in the databox. A value of 0.0 is opaque and 1.0 is transparent.

Adjust the slidebar to set a value for the contour width multiplier, or enter a value in the databox.

Select a contour line style from the pull-down menu.

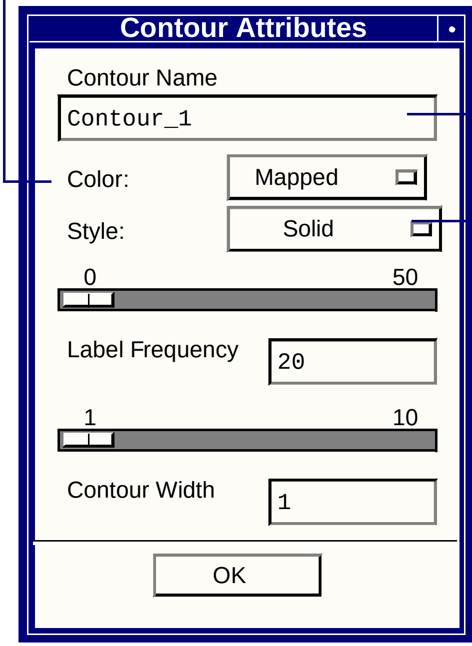
Turns ON/OFF the element edge display.

Adjust the slidebar to set a value for the shrink factor for the targeted faces for the fringe tool. A value of 0.0 is full size, 1.0 is completely collapsed to the face centroid.

Turns ON/OFF continuous tone fringe plots. This feature will be disabled on display devices not supporting true color.

Contour Attributes

Select a color for the contour lines. Mapped will map the current result range to the spectrum for color assignment.



A default name for the contour is provided. Enter a new name, if desired.

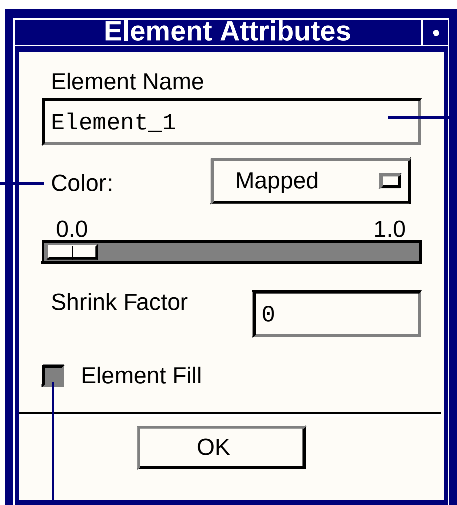
Select a contour line style from the pull-down menu.

Adjust the slidebar to set a value for the label frequency, or enter a value in the databox. A value of zero will display no spectrum level labels and a non-zero value will display labels along the contour lines.

Adjust the slidebar to set a value for the contour width multiplier or enter a value in the databox.

Element Attributes

Select a color for the elements. Mapped will map the current result range to the spectrum for color assignment.



A default name for the element is provided. Enter a new name, if desired.

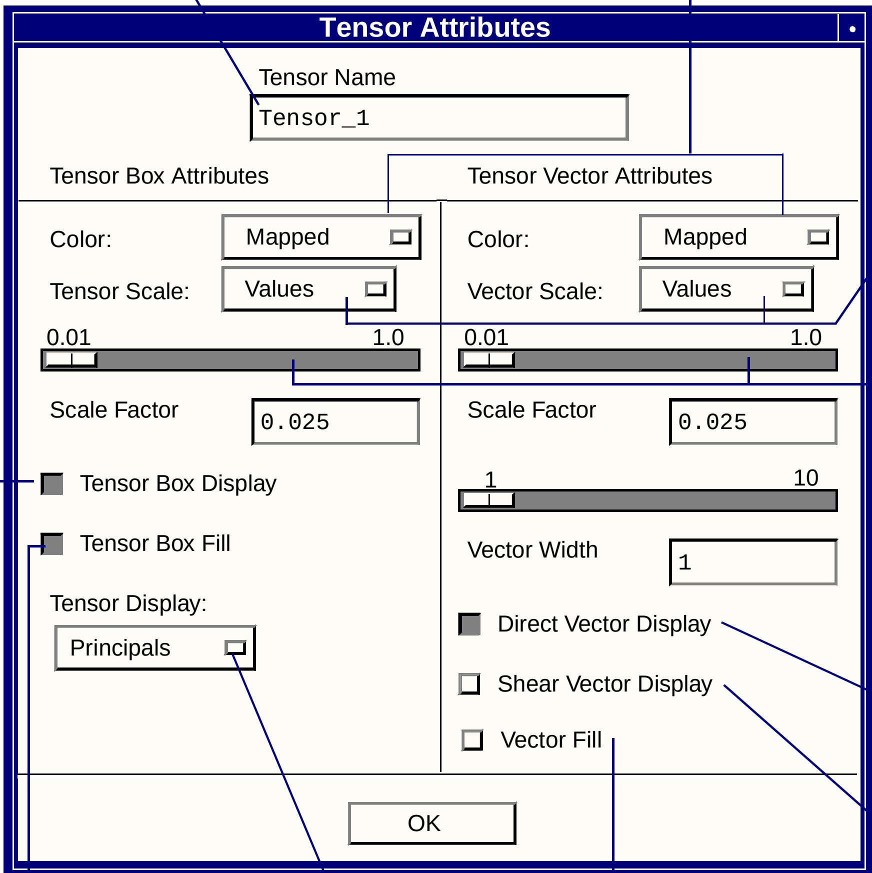
Adjust the slidebar to set a value for the shrink factor for the element attributes for the tool, or enter a value in the databox. A value of 0.0 is full size and 1.0 is completely collapsed to the element centroid.

Turns ON/OFF the element face fill. Color coded elements will either be displayed as shaded faces or wireframe edges.

Tensor Attributes

A default name for the tensor is provided. Enter a new name, if desired.

Select a color for the tensor box and the tensor vectors. Mapped will map the current result range to the spectrum for color assignment.



Select a scaling method for the tensor box and vectors. *Values* will vary the size based on the result magnitude. *Screen* will use a fixed scale based on the screen size.

Adjust the slidebar to set a scale factor for the tensor box and vectors, or enter a value in the databox. This will be a screen size multiplier.

Adjust the slidebar to set a vector width multiplier, or enter a value in the databox.

Turns ON/OFF the display of the direct vectors.

Turns ON/OFF the display of the shear vectors.

Turns ON/OFF the polygon fill of the tensor box faces.

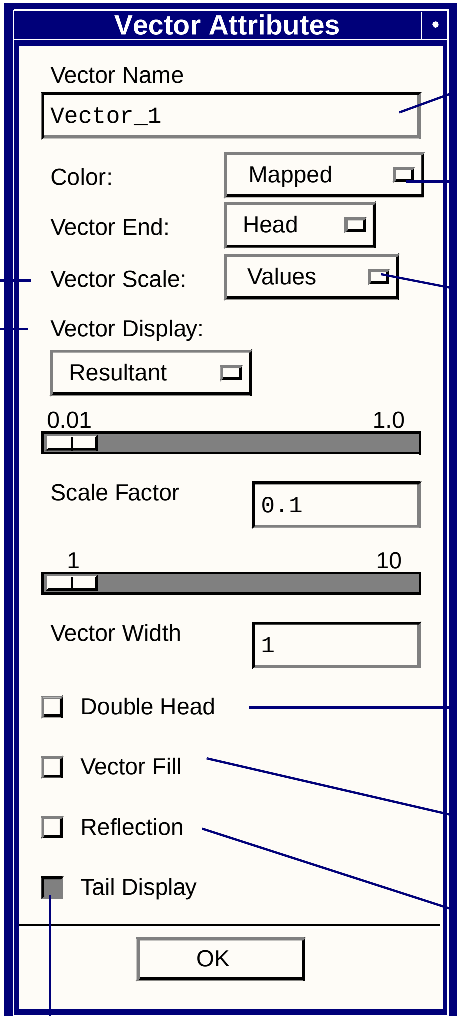
Turns ON/OFF the display of polygon filled vector arrowheads.

Select a display method for the tensor orientation from the pull-down menu. *Principals* will orient the tensor in the direction of principal stress. *Components* will orient the tensor in the coordinate system defined for the result.

Turns ON/OFF the display of the tensor box.

Vector Attributes

Select a scaling method for the vectors. *Values* will vary the size based on the result magnitude. *Screen* will use a fixed scale based on the screen size.



A default name for the vector is provided. Enter a new name, if desired.

Select a color for the vectors. Mapped will map the current result range to the spectrum for color assignment.

Select a vector end registration method. *Head* will connect the vector head to the result location. *Tail* will register to the tail.

Adjust the slidebar to set a scale factor value for the vectors, or enter a value in the databox. This will be a screen size multiplier.

Adjust the slidebar to set a vector width multiplier, or enter a value in the databox.

Turns ON/OFF the display of double headed vectors.

Turns ON/OFF the display of polygon filled vector arrowheads.

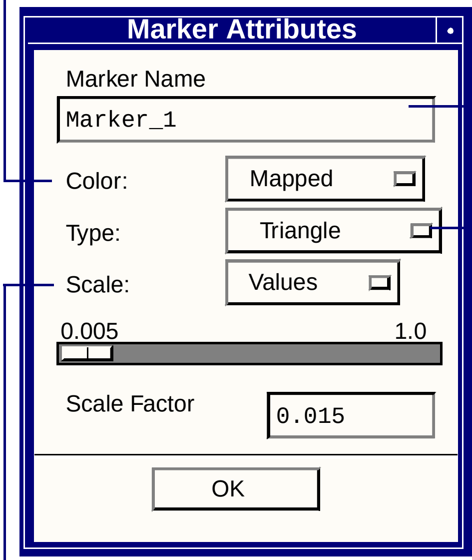
Turns ON/OFF the display of reflected (two way) vectors.

Turns ON/OFF the display of vector tails.

Select a display method for the vectors. *Resultant* will display the resultant vector. *Components* will display the individual vector components oriented in the coordinate system defined for the result.

Marker Attributes

Select a color for the markers. Mapped will map the current result range to the spectrum for color assignment.



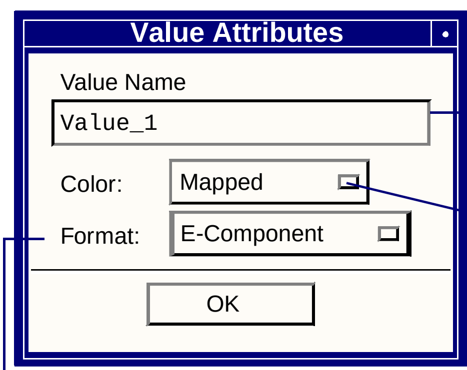
A default name for the marker is provided. Enter a new name, if desired.

Select a marker icon type style. These icons will be positioned at the element centroids or nodal locations.

Adjust the slidebar to set a scale factor value for the markers, or enter a value in the databox. This will be a screen size multiplier.

Select a scaling method for the marker icons. *Values* will vary the size based on the result magnitude. *Screen* will use a fixed scale based on the screen size.

Value Attributes



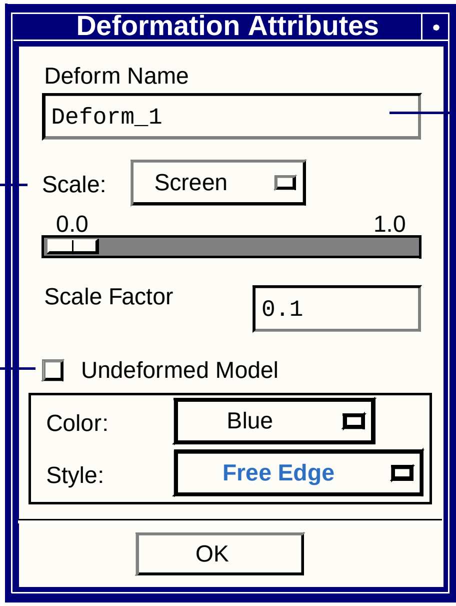
A default name for the value is provided. Enter a new name, if desired.

Select a color for the values. Mapped will map the current result range to the spectrum for color assignment.

Select a format for displaying the values. “*E*” will use exponential format (i.e., 1.0E+04). “*F*” will use fixed format (i.e., 1000.0). “*G*” will use either “E” or “F” depending on the value magnitude. “*I*” will use an integer format (i.e., 1000).

Deformation Attributes

Select a scaling method for the deformation. *Screen* will scale the deformation based on the current screen size. *True* will show the actual deformation multiplied by the scale factor.

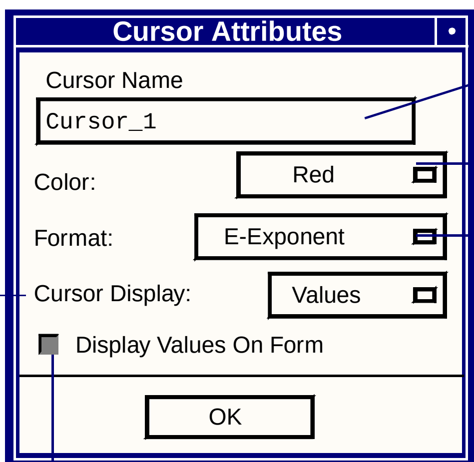


A default name for the deform is provided. Enter a new name, if desired.

Adjust the slidebar to set a scale factor for the deformation, or enter a value in the databox. If the method is *Screen*, the scale factor will be multiplied by the screen size. If the method is *True*, the scale factor will be used as a true scale multiplier of the actual deformation. True scale should be used to display transient animations. The slidebar extremes will expand if a value exceeding the current range is entered in the databox.

Turns ON/OFF the display of the undeformed model. The color and style of the undeformed model can be controlled with the Color and Style menus.

Cursor Attributes



A default name for the value is provided. Enter a new name, if desired.

Select a color for the values.

Select a format for displaying the values. “*E*” will use exponential format (i.e., 1.0E+04). “*F*” will use fixed format (i.e., 1000.0). “*G*” will use either “E” or “F” depending on the value magnitude. “*I*” will use an integer format (i.e., 1000).

Toggle this button to control if the cursor tool information is to be added to the spreadsheet as displayed on the screen.

Select the cursor tool to display either the result values or the node/element IDs. Both the IDs and result values will be displayed on the spreadsheet.

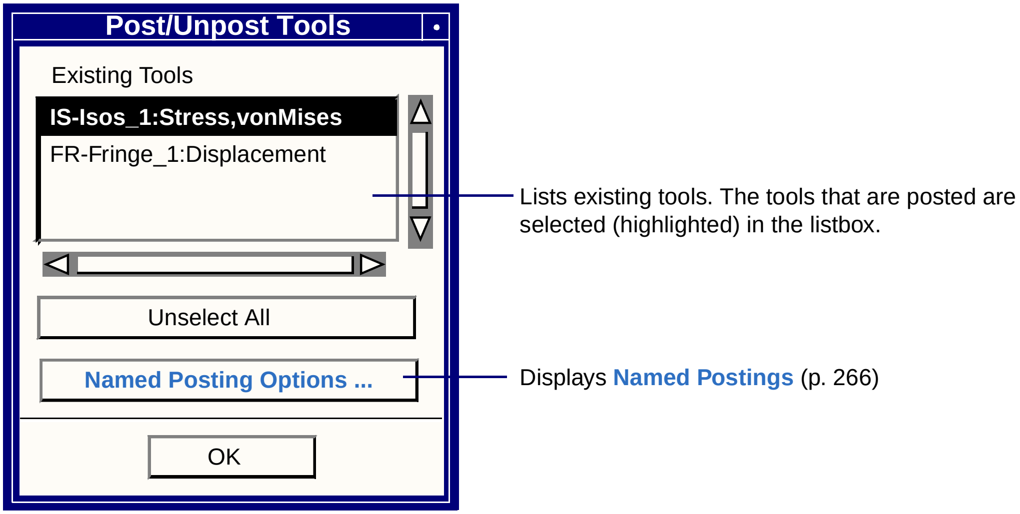
Insight Control

The Insight Control capabilities are found in the Insight Control pull-down menu on the MSC.Patran main form. There are five control forms which are listed:

- Post/Unpost Tools
- Isosurface Control
- Range Control
- Animation Control
- Modal Animation
- Rake Specification
- Cursor Results

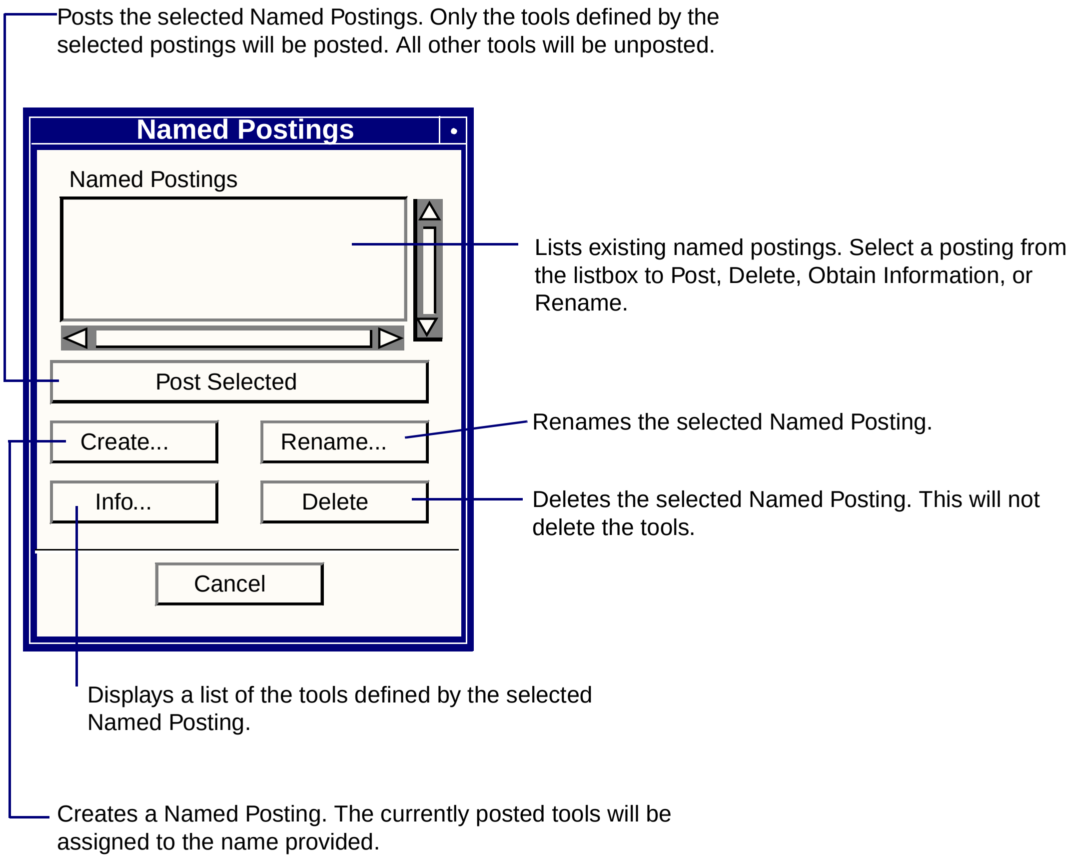
Post/Unpost Tools

The following form appears when the Post/Unpost Tools option is selected in the Insight Control pull-down menu.



Named Postings

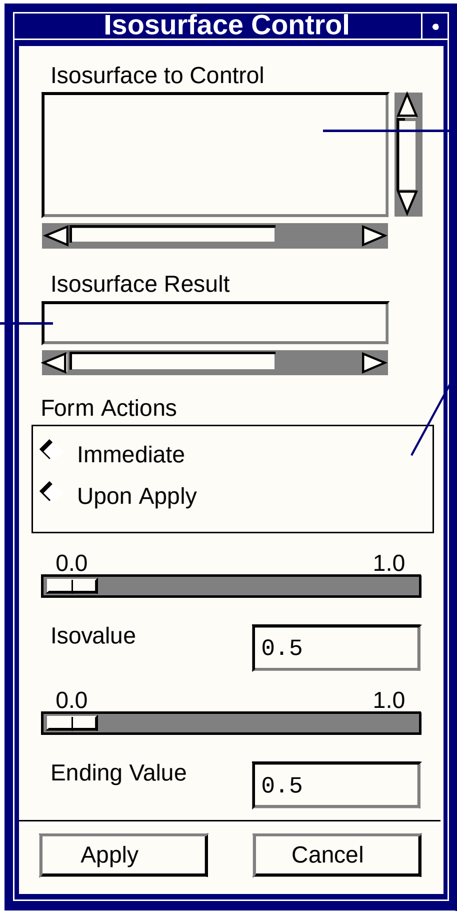
The following subordinate form appears when the Named Postings Options button is selected on the Post/Unpost Tools form.



Isosurface Control

The following form appears when the Isosurface Control option is selected in the Insight Control pull-down menu.

The result/coordinate associated with the selected isosurface will be displayed here as information.



Lists all posted isosurfaces. Select an isosurface from the list to activate it for dynamic control using this form.

The form actions may be switched to be either interactive or user controlled. Immediate mode will allow for display updates as the values are being changed. Upon Apply will update the display only when you press the apply button.

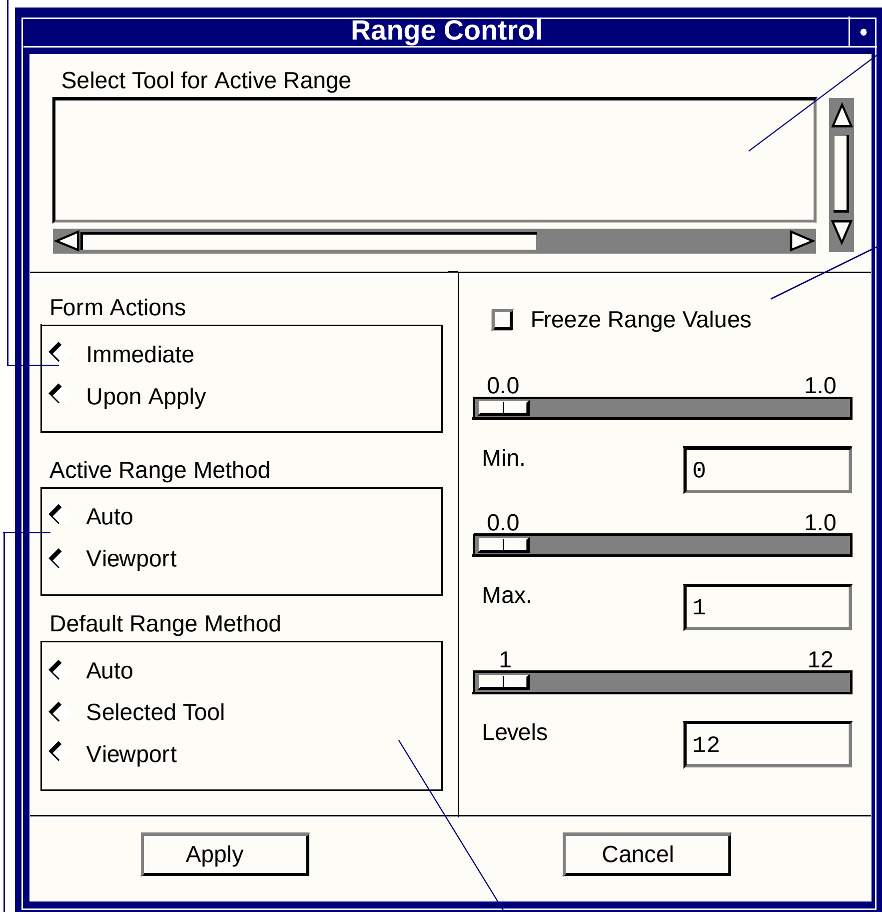
Adjust the slidebar to set a value, or enter a value in the databox, for single surface isosurfaces, or a starting value for multiple surface isosurfaces. The number of displayed surfaces for an isosurface tool is modified using **Isovalue Setup** (p. 252) (accessed through the Results Selection form,) or by accessing **Isosurface Coordinate Selection** (p. 253) for coordinate driven isosurfaces. The valid range for the values entered is determined by the current result or coordinate range. Adjusting the slidebar or entering a value in the databox will update the display in immediate mode. The frequency of the immediate mode display update is controlled from the**Insight Preferences Form** (p. 278).

Use the slidebar to set a value for the ending value for multiple surface isosurface tools or enter an ending value in the databox. This value will be tied to the isovalue for single surface tools.

Range Control

The following form appears when the Range Control option is selected in the Insight Control pull-down menu.

May be switched to be either interactive or user controlled. Immediate mode will allow for display updates as the values are being changed. Upon Apply will update the display only when the apply button is pressed.



Select a tool from this list to display or modify its spectrum and range values. This tool will be associated with the displayed spectrum.

Used to hold the current minimum and maximum values for the active Auto range. If the values are not frozen, they will be set automatically based on the active range tool result.

If the Active Range Method is *Auto*, adjust the slidebars to set a value, or enter a value in the appropriate databox, for the minimum and maximum range, and the number of levels for the active tool range.

Select a method for assigning color to the active range tool. *Auto* will assign color and display the spectrum ranges based on values automatically determined by the tools result. *Viewport* will assign color and display the spectrum ranges as those currently assigned to the Insight viewport.

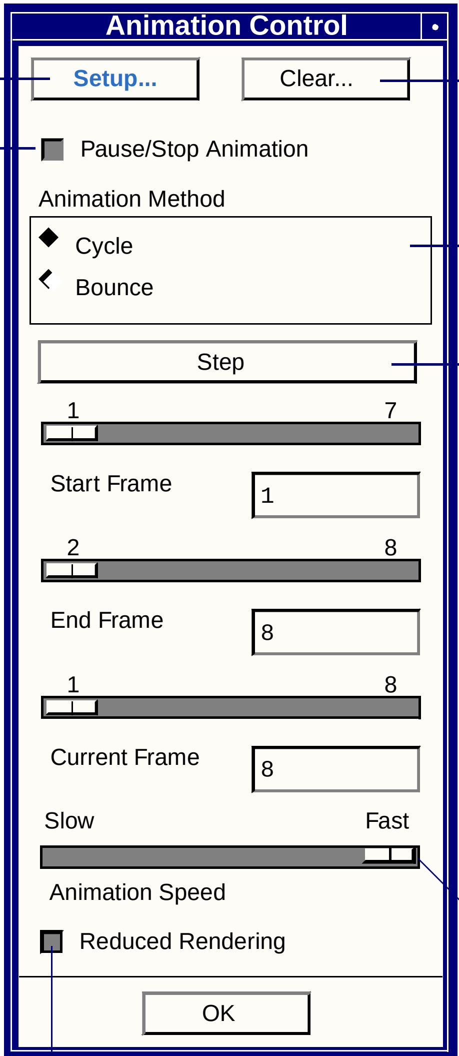
Select a method for assigning color to all other tools posted. *Auto* will assign color based on each tools result/coordinate range. The assigned color will most likely NOT correspond to the displayed spectrum and range values, but will give a good qualitative indication of the tools' result variations. *Selected Tool* will guarantee a one-to-one correspondence of assigned color to the displayed spectrum and ranges. However, it may not provide any color variation for tools other than the active range tool. *Viewport* will assign color levels based on the current range assigned to the Insight viewport, regardless of the Active Range method.

Animation Control

The following form appears when the Animation Control option is selected in the Insight Control pull-down menu.

Pause/Stop or Restarts the animation currently running. The animation must be paused to enable any capability other than the start and end frame values.

If an animation is not currently defined, click this button to display **Animation Setup** (p. 270) to first define an animation.



Clears the current animation. If an animation is currently running, it may be necessary to first *Pause* the animation.

Select a method for displaying the current animation. *Cycle* will display the animation frames 1:n, 1:n,... *Bounce* will display the frames 1:n:1:n...

If an animation is currently paused, click this button to step through the animation frame-by-frame using the current method.

Adjust the slidebar to set a value, or enter a value in the appropriate databox, for the Start Frame and End Frame of the animation currently running. Changing these values will skip the display of all the frames above or below the entered values, respectively.

Adjust the slidebar or enter a value for the current frame to display a paused animation. Adjusting the slidebar or entering a value in the databox will update the display accordingly.

Adjust the slidebar to control the speed of the animation. Having the slidebar set fully to Fast will animate as quickly as the graphics device is able.

Toggle the reduced rendering mode for animations. The reduced rendering mode will display only the wireframe to speed up animation on non-3D devices.

Animation Setup

The following subordinate form appears when the Setup button is selected on the Animation Control form. Use this form to create an animation sequence.

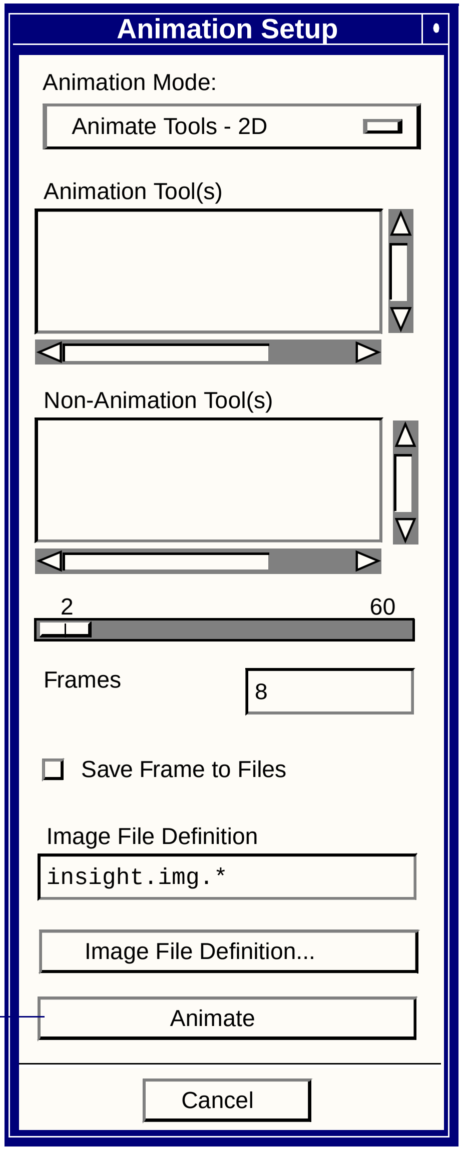
There are two basic methods of animation, 2D and 3D. 2D animations will be faster on display devices without hardware accelerators and with large models. View manipulations are not supported during 2D animations. 3D animations might be slower but do support rotations, panning and zooming during an animation. The animation of tools will automatically generate frames using the animation attributes defined for each tool. All posted tools will be displayed, but only tools with animation enabled will change from frame to frame. Each tool with animation enabled will have its results interpolated (linear interpolation only) to each animation frame based on the attributes specified for the tool on the Animation Attributes form.

If a tool's animation method is Global Variable, each animation frame result will be interpolated based on the available results associated with global variables bracketing the current animation frame. This method is only available for tools associated with multiple loadcases. For example, if a tool is to be animated using six result load cases with time ranging from zero to two seconds, and the number of frames is set to eleven, results will be linearly interpolated from the existing data at Time=0.0, 0.2, 0.4,..., 2.0. If a desirable global variable does not exist for the selected results for a tool, the default LOAD CASE INDEX variable may be used. LOAD CASE INDEX assigns an incremental integer count to each selected load case and uses it to interpolate animation frames as with any other global variable.

The Ramp and Sinusoid animation methods will scale the static default result for each enabled tool using a ramp or sinusoid scale factor which is determined for each frame. These methods will scale the current starting isovalue for animation of coordinate driven isosurfaces.

The 2D animations provide additional support for the generation and retrieval of image files. These files can either be generated automatically during an animation of tools or created individually to be played back later. The creation of individual animation frames enables the user to form arbitrary animations. The image files are handled using their version numbers to determine which frame they define.

Caution should be used in animating multiple tools with multiple methods pointing to multiple results simultaneously. The resultant display will be difficult to interpret.



The animation modes are Animate Tools - 3D, Animate Tools - 2D, Animate Images - 2D, and Create Images-2D. The 3D animations may be slower on some devices but allow view manipulations during animations. The 2D animation capabilities also provide for the optional storage and retrieval of animation frame images. The Create Images mode will capture the current display and create a file for future redisplay.

All the posted tools will be listed in these two listboxes. Tools with animation enabled will be listed in the upper listbox and those with animation disabled will be listed in the lower listbox. Only the tools listed in the upper listbox will animate although *all tools will be displayed* during an animation. Select a tool in either listbox to display the applicable **Animation Attributes** (p. 249) form. The listboxes will be updated depending on the Enable Animation toggle value on the Animation Attributes form. These listboxes are not used when animating images.

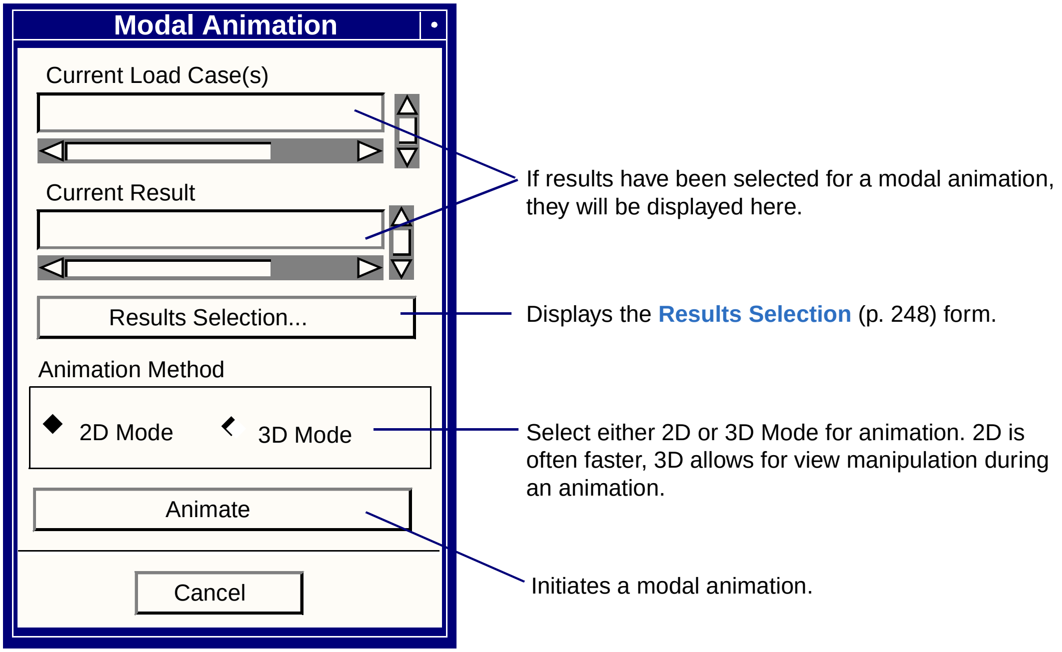
2D animations can optionally be stored or retrieved from image files. The image files must all have the same filename with the version numbers consecutive from .01.

The Animate/Create Image File button will either initiate the create of an animation or create an image file if the mode is Create Images.

Adjust the slidebar to set a value, or enter a value in the databox, for the number of animation frames to be created. This value is used for the redisplay frame number when creating an image file.

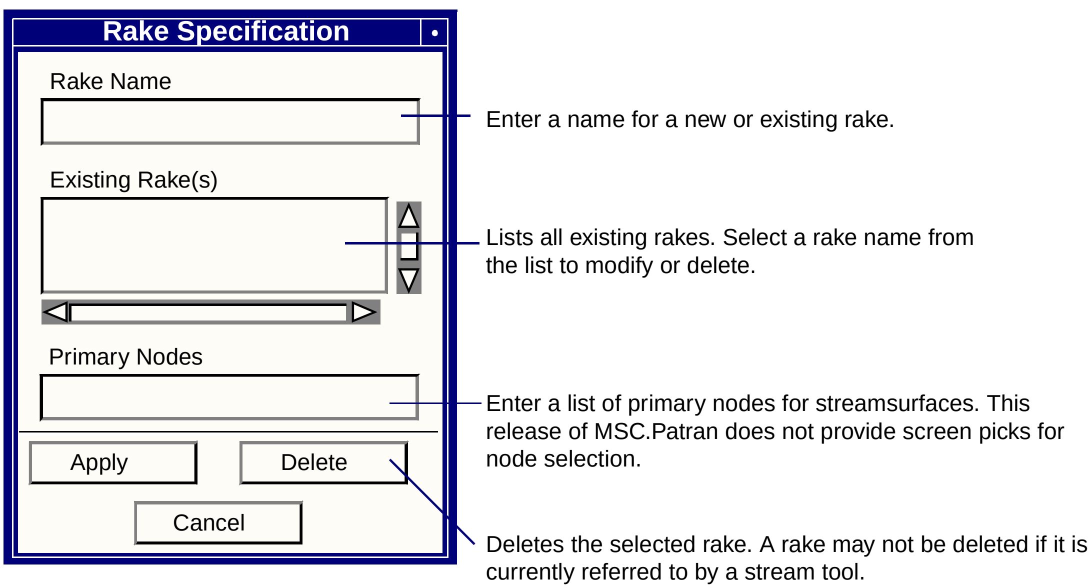
Modal Animation

Provides rapid access to a modal animation display. The animation is initiated when the Animate button is pressed. The Cancel button will terminate the animation. The Animation Control form can be displayed to provide further control of the running animation if desired.



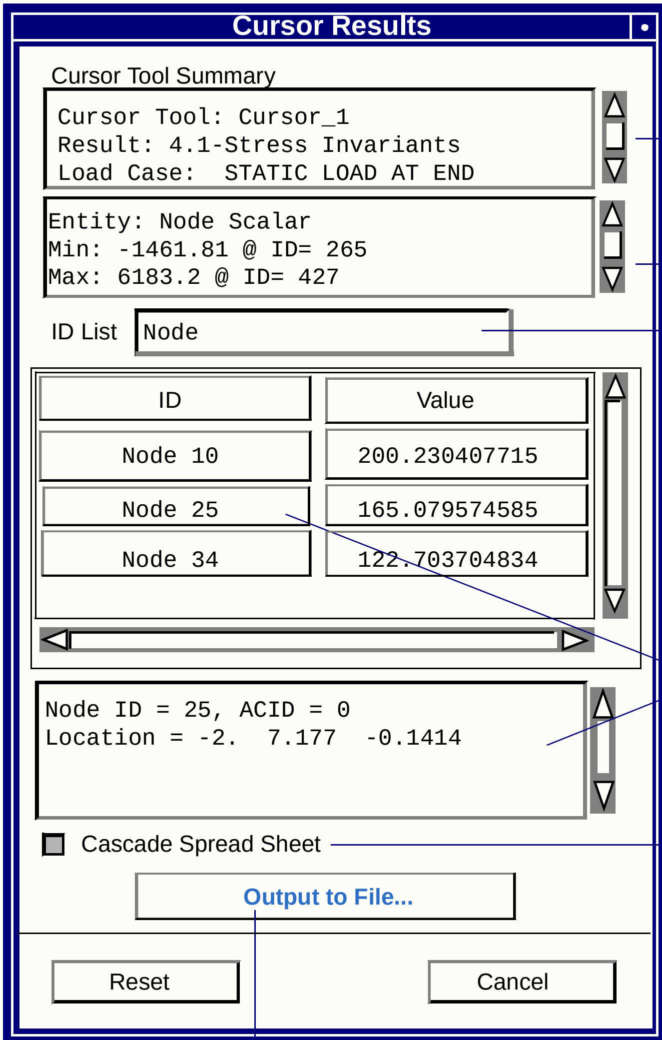
Rake Specification

The following form appears when the Rake Specification option is selected in the Insight Control pull-down menu.



Cursor Results

The following form is displayed automatically when a cursor tool is posted or by selecting the Cursor Results option from the Insight Control menu.



Information regarding the current posted cursor tool will be displayed here.

Enter a node or element list depending on where the cursor tool is targeted. The IDs specified will be added to the spreadsheet and the values will be displayed on the screen.

Note: Be aware that adding large numbers of entries to a spreadsheet is very time consuming and not recommended.

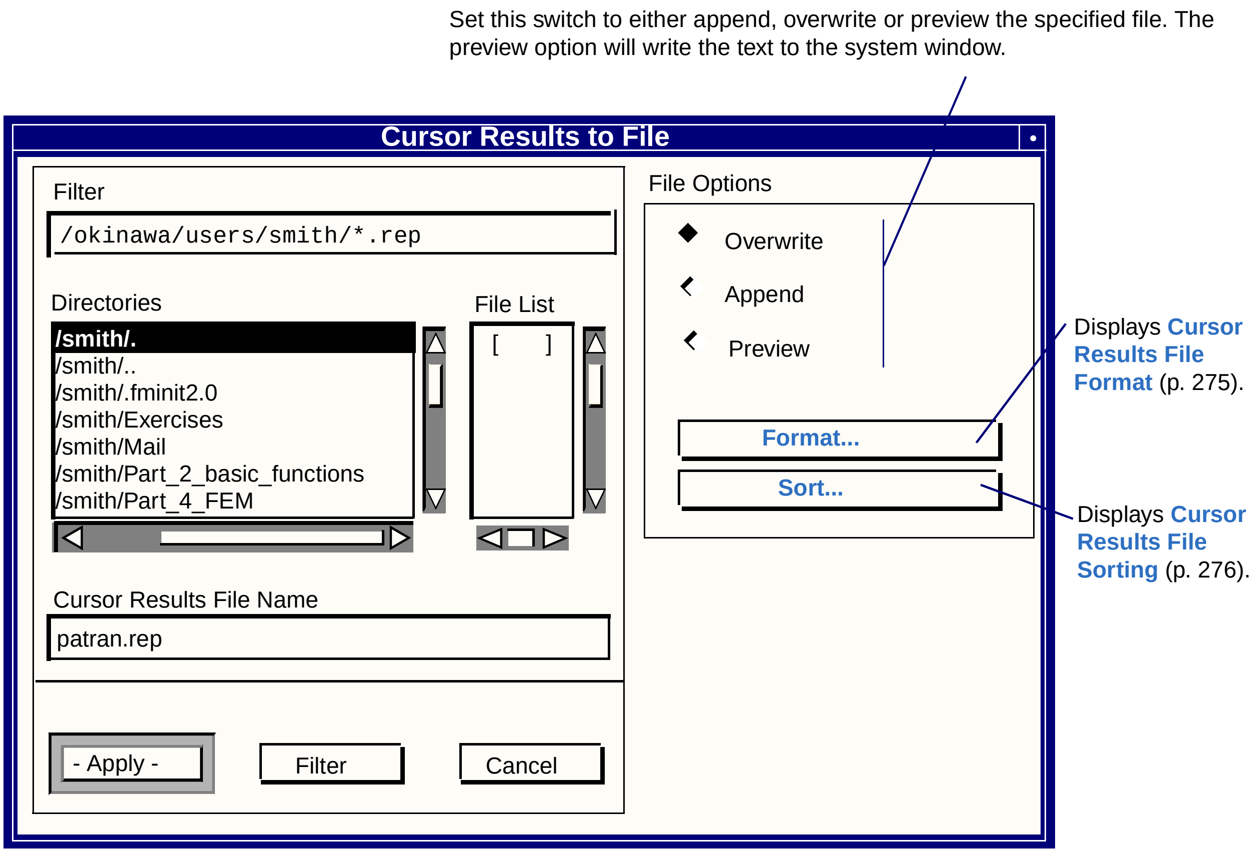
Selecting a cell in the spreadsheet will provide additional information in the textbox.

Allows screen selected entities to either overwrite the last spreadsheet row or add a new one. Entering a list of multiple entities in the databox will automatically toggle the spreadsheet to cascade.

Displays **Cursor Results to File** (p. 274).

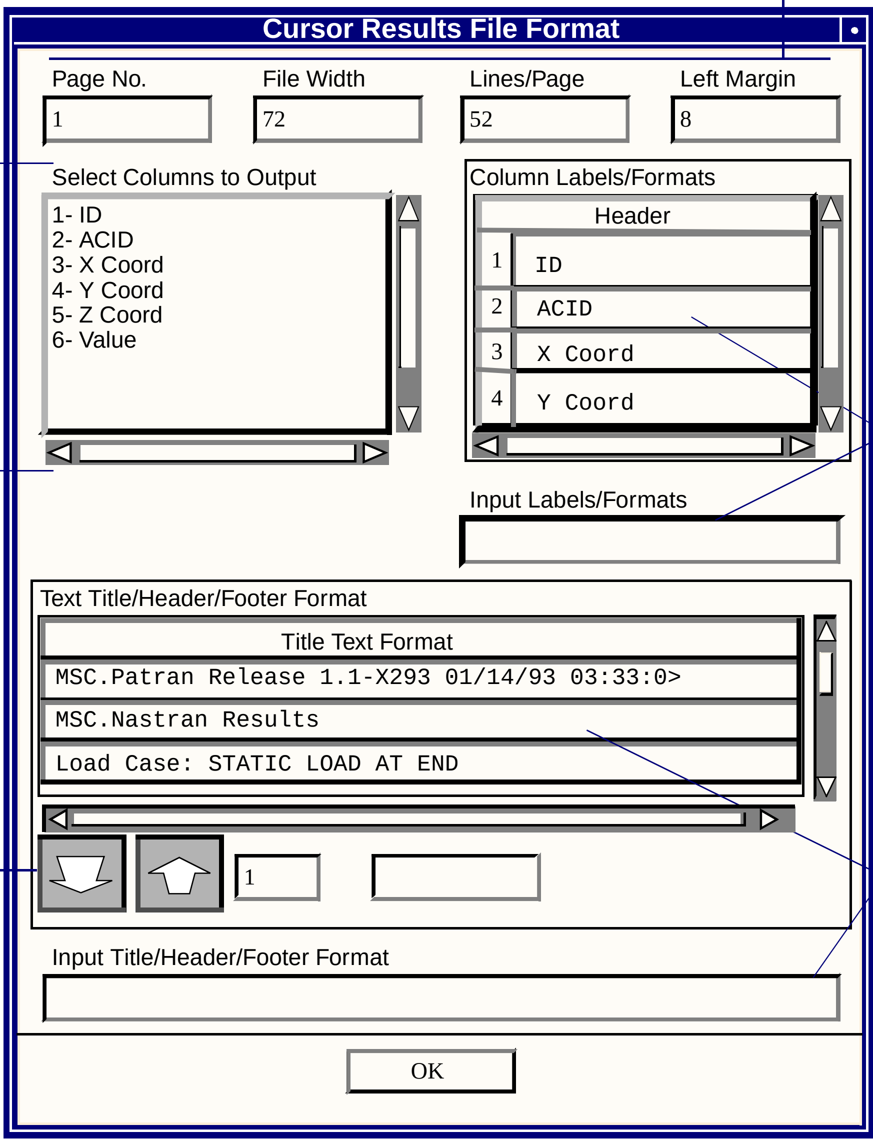
Cursor Results to File

The following form is displayed from the Cursor Results spreadsheet form.



Cursor Results File Format

The following form is displayed from the Insight Cursor Results File Form. This form provides control of the file formatting for output of the information currently in the cursor tool spreadsheet.



The page, width, lines and margin databoxes are used to control the page numbering and page sizing.

The text for the column headings and the formats for the columns are defined by the cells in the format spreadsheet. The column formats are viewed by sliding the horizontal scroll bar. Entries are edited by selecting a cell and modifying the text as it appears in the databox. The format strings are described in the **text_write_string** (p. 112) in the *PCL and Customization*.

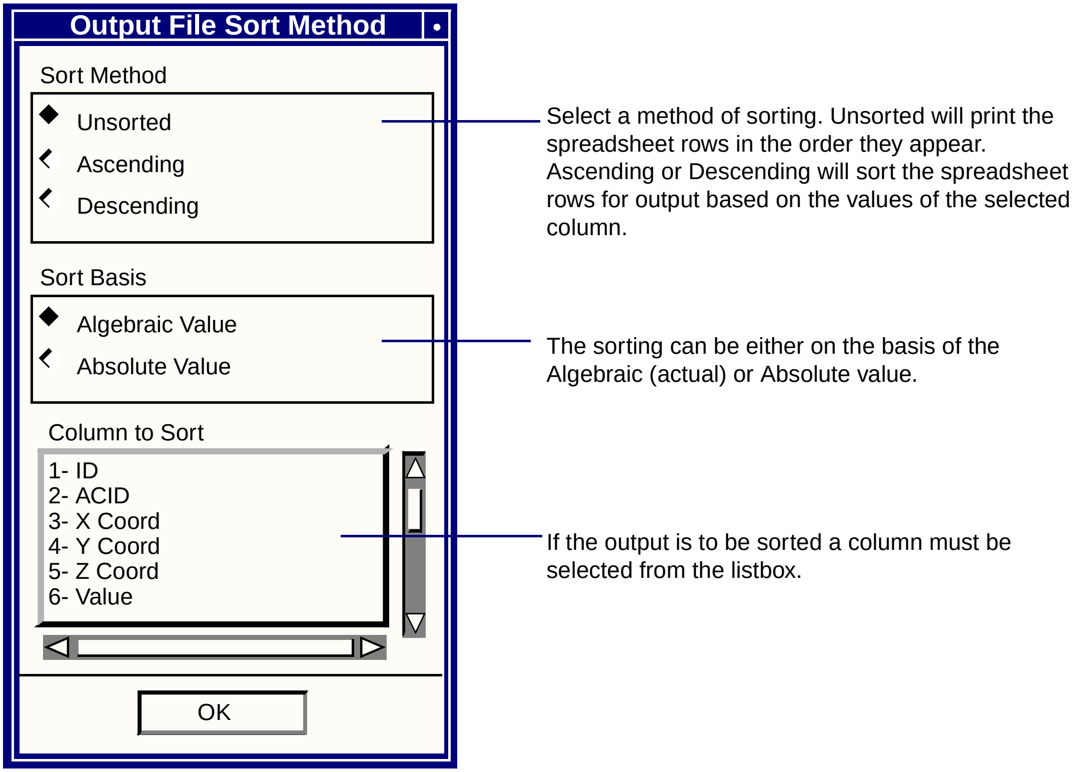
Up to 5 lines of text are available to be defined for the file title and page headers and footers. Entries may be edited by selecting the spreadsheet cell and modifying the text as it appears in the databox.

Access the lines of text to be used for the title (appears once at the top of the file), the header (appears at the top of each page) and the footer (appears at the bottom of each page). If the format string %I% appears in the header or footer, the current page number will be inserted in its place.

Select any number of items from the available list to be included as columns in the output file.

Cursor Results File Sorting

The following form is displayed from the Cursor Results File Form. It controls the output columns and methods for sorting of the spreadsheet data for file output.

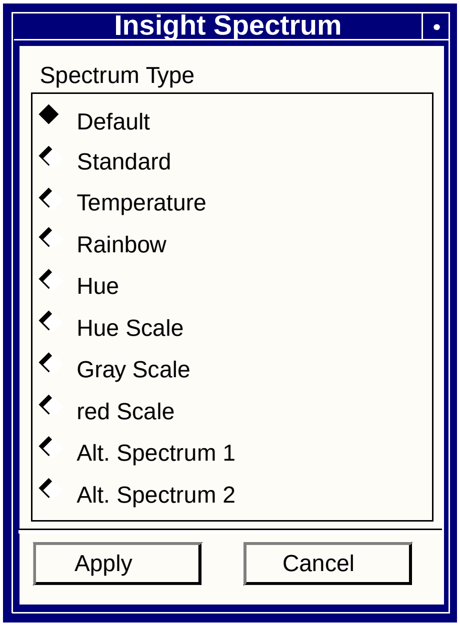


Global Settings

The global settings are controlled through two forms. The **Insight Spectrum** (p. 277) form controls the colors used in the current spectrum. The **Insight Preferences** (p. 277) form controls the default graphics such as model rendering style, color, entity labeling, and others.

Insight Spectrum

The following form is displayed when the Spectrum option is selected in the Display menu.



Select a *spectrum type* to assign to the Insight viewport. Each spectrum has a maximum of twelve color levels. Any spectrum setting other than the default will effect the colors of shaded entities (faces).

Insight Preferences

The Insight Preferences form provides capabilities for control settings which are specific to the Insight application but have global effects on its function. The Display Method pull-down menu controls the method of displaying the finite element model in the Insight viewport. Posting a Fringe tool to the model faces or a clipping isosurface will take precedent over the default methods.

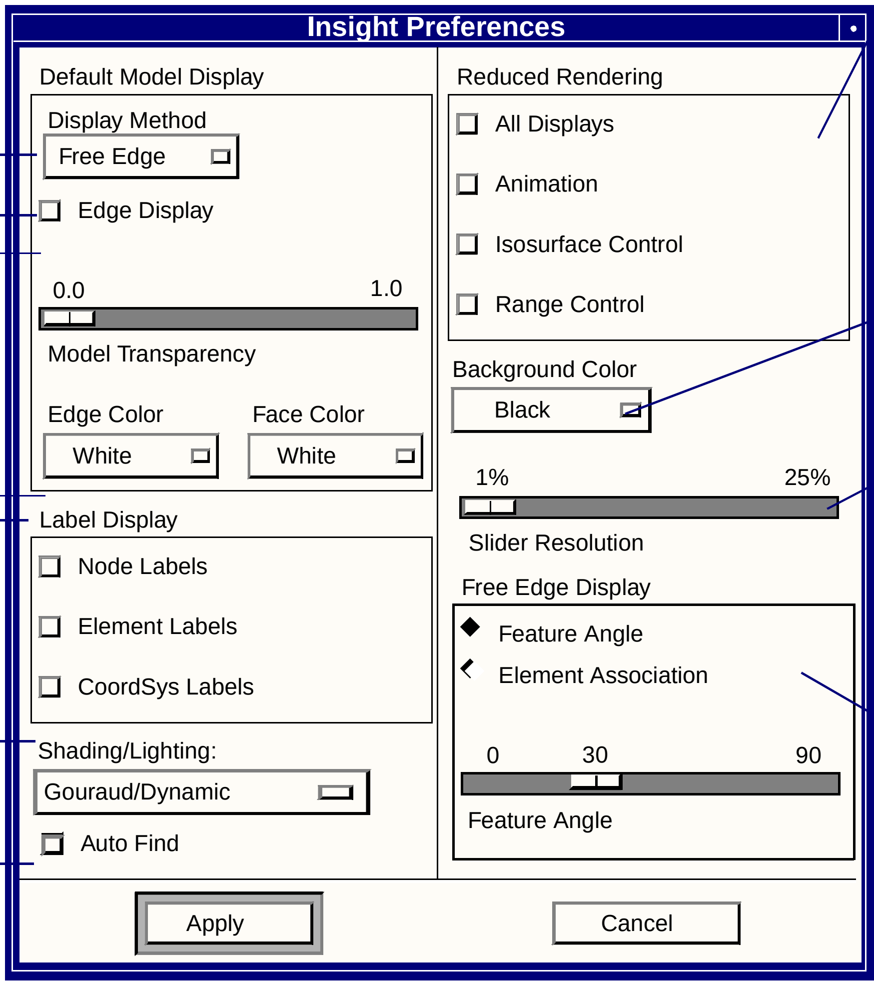
Free Edge	Display the model as a wireframe image with inter-element boundaries visually suppressed. Shows only those edges belonging to only one element.
Wireframe	Display the model as a wireframe image including any inter-element boundaries present in the posted group.
Shaded	Display the model as a solid-shaded image.
Transparent	Display the model as a transparent, solid-shaded image whose degree of transparency is specified by the Model Transparency slidebar.
Hidden Line	Display the model as a hidden line plot by suppressing the rendering of edges not normally visible on a solid object.

Insight Preferences Form

Set a value for the default model face transparency. A value of 0.0 is opaque and 1.0 is transparent. Control face and edge colors.

Displays the element edges on faces.

Select a display method from this pull-down menu for the default model rendering style.



Select any of these options on to have all filled polygons drawn as *wireframe outlines* during the specified operation(s). This will increase the graphics speed when hardware shading is not available.

Select a color for the display background. *Fancy* is only available for true color devices.

Adjust the slidebar to define the percentage change between updates for all non-integer dynamic slidebars used in the Insight application forms.

The free edges of a model can be determined either by the angles between adjacent faces (Feature Angle) or by displaying only edges belonging to one element (Element Association). The slidebar is used to set the angle used for the feature angle method. All edges with an angle greater than the feature angle between normal will be displayed.

Controls the display of faces using either Flat, Gouraud, or No shading and use either static or dynamic lighting of shaded faces. With dynamic lighting, the shading is modified as the model is rotated. Auto Find controls the automatic fit performed after view rotations.

Control the display of Node, Element, and Coordinate labels. Element labels will only be displayed when elements are displayed by setting the model display method to Wireframe, or by posting an element tool.

CHAPTER

14 Numerical Methods

- Introduction
- Result Case(s) and Definitions
- Derivations
- Averaging
- Extrapolation
- Coordinate Systems

14.1 Introduction

A result in MSC.Patran is an array of one, three, or six numbers that represent a physical results quantity associated with finite element entities. These results are computed by the analysis program and loaded into or referenced by the MSC.Patran database by an application interface which translates the results.

Results can be retrieved only after the following questions are resolved:

- What Result Case does it belong to? See **Result Case(s) and Definitions** (p. 281).
- Are the results a scalar, vector, or tensor quantity? See **Data Types** (p. 281).
- In what coordinate system do the results belong? See **Coordinate Systems** (p. 305).
- Are the results associated with nodes or elements? See **Target Nodes and Elements** (p. 284).
- Are the results complex or single valued? See **Numerical Form** (p. 283).
- What layer or position do the results belong to? See **Layer-Position** (p. 284).
- For element results, where in the element are they computed? See **Element Position** (p. 285).

Each question involves an attribute which characterizes the result and differentiates it from other results. Every result must have all of these attributes defined before it can be retrieved from the database. This chapter dedicates itself to explaining these attributes and the internal workings of the program when these attributes are modified and manipulated.

14.2 Result Case(s) and Definitions

Results for each model are partitioned into identifiable sets called Result Cases. A Result Case may correspond to a static load case, a load step in a nonlinear analysis, a mode shape in a normal mode analysis, or a myriad of other analysis types. Result Cases generally correspond to Load Cases from an analysis run as defined and set up in MSC.Patran. The terminology between Load Case and Result Case is interchangeable in many cases. However it is possible in MSC.Patran to import results that are not associated to a Load Case. For this reason result sets are referred to as Result Cases as opposed to Load Cases.

Each item in the listbox filled with result sets corresponds to only one Result Case. Each Result Case is associated with one or more global variables. Global variable are results that are global to a particular set of result and not each individual finite element entity. All results have at least one global variable, that being the LOAD CASE INDEX. This is basically an internal ID of the Result Case. Other global variables can include mode number, frequency, time, and design cycles / variables. The global variables can be used:

- to select Result Cases for filter display in the listbox
- as variables in graph (XY) plots
- as animation start and end values (transient)
- in text reports

The MSC.Patran result postprocessor treats all Result Cases on an equal basis. There is no distinction between a static, nonlinear analysis, transient, or modal analysis except as indicated in its title. A result type can exist in one or more Result Cases. Once a group of Result Cases are selected, the associated result types are retrieved from the database. Duplicate and conjugate (for complex results) result types are removed. The result type labels are listed in the Result listboxes. An item in a result listbox may represent one or more result types in the database.

The MSC.Patran results postprocessor has no pre-defined result types. The tool attaches no internal significance to the labels of the result types. Result processing is completely based on the result attributes. The external translators that import results determine which labels to assign to which results from the analysis program output. Once translated and stored in the MSC.Patran database, the source of these results are transparent to the results postprocessor.

The MSC.Patran results postprocessor allows for selection of multiple Result Cases for processing. This functionality is important for animation, xy plots, load case combination, and finding the maximum / minimum of results across load cases. More information about result data types is presented below,

Data Types. Results may be scalar, vector, or tensor quantities. The data type of each result is determined and set by the result translator. Scalar results have no coordinate system attributes, but vector and tensor results are always associated with some coordinate system. A tensor is defined as a symmetric matrix of rank 2 which is stored as six associated values (xx, yy, zz, xy, yz, zx). Unless otherwise noted, vectors and tensors always denote the components that represent them in a certain coordinate system. See **Coordinate Systems** (p. 305).

The plot types are associated with the data type and the associativity of the results as follows:

Data Type	Associativity	Available Plot Types
Scalar	Node Element	Fringe, Graph, Report, Animation, Combine & Derive Results (Scalar Marker, Value, Isosurface, Threshold, Contour, Element, Value & Cursor in the Insight application)
Vector	Node Element	Deformation, Vector, Report, Animation, Combine & Derive Results (Streamline & Streamsurface in the Insight application)
Tensor	Node Element	Report, Tensor, Animation, Combine & Derive Results

The data type of the results can be changed to make its associated plot type available for plots. This change may involve a derivation and / or coordinate system definition. See **Derivations** (p. 286).

From	To	Coordinate System	Remark
Scalar	Vector	Yes	Scalar value inserted into the specified component. Other components = 0.
	Tensor	Yes	Same as Scalar to Vector above.
Vector	Scalar	Yes / No	The data system is the system as defined in the database. You can specify the output system. Only vector components need coordinate systems.
	Tensor	Yes	The data system is the system as defined in the database. You can specify the output system for the tensor. A vector component can be inserted into a tensor component.
Tensor	Scalar	Yes / No	Coordinate system is required if the scalar is one component of the tensor. Principal values are invariant with respect to coordinate systems. Only tensor components need coordinate system.
	Vector	Yes	The data system is the system as defined in the database. You can specify the output system for the vector. A tensor can be reduced to its principals in vector form.

Associativity. Results are associated with either nodes or elements, but not both. This associativity characterizes the result type. If a particular result exists both as a nodal result and an element result (e.g., energy), the result translator must create two result types with distinct result labels.

You cannot change this associativity and, once defined, the attribute along with the data type determines the available plot types. Some plot types only deal with a particular associativity (e.g., deformed plot only for nodal vector results, tensor plot only for element tensor results) but other plot types deal with both (e.g., fringe plot, xy plot, report). Some processing methods are only applicable to either nodes or elements, whereas other methods are applicable to both.

If a processing method involves elements, but results are associated with nodes, the results at these nodes will be assigned to the elements sharing the common nodes. Interpolation functions are then used to compute results at any point within the element from the results at element nodes. See **Averaging** (p. 292).

The converse is also true. If a processing method involves nodes, but results are associated with elements, the results within the elements will be extrapolated out to the nodes. See **Extrapolation** (p. 299). To report results at the nodes from elemental data, the contribution at a node from each surrounding element is averaged to a single scalar value (or vector or tensor depending on the data type). When derived results are requested from vector or tensor data, the order in which averaging and derivation are done is important. Control of this order is given to the user which can give different answers.

Numerical Form. The MSC.Patran Results Processor can process complex results. Each result type has an attribute to indicate its numerical form:

Primary Numerical Form	Associated Numerical Form	Considered As
Real	Imaginary	Complex
	Other	Single
Imaginary	Real	Complex
	Other	Single
Magnitude	Phase (radians)	Complex
	Other	Single
Phase (radians)	Magnitude	Complex
	Other	Single

If results are complex, the option to display the values as real, imaginary, magnitude, or phase are computed (in temporal space) at a particular phase angle.

The result at any angle θ is:

$$\text{Real} * \cos(\theta) + \text{Imaginary} * \sin(\theta)$$

or

$$\text{Magnitude} * \cos(\text{Phase}) * \cos(\theta) + \text{Magnitude} * \sin(\text{Phase}) * \sin(\theta)$$

where $\theta = 0$ corresponds to purely real results and $\theta = 90°$ corresponds to purely imaginary results.

Magnitude and phase of complex results are computed from real and imaginary result pairs as

$$\textbf{Phase} = \tan^{-1}\frac{\text{real}}{\text{imaginary}}$$

$$\textbf{Magnitude} = \sqrt{\text{real*real + imaginary*imaginary}}$$

which is then converted to range $[0, 2\pi]$.

Except for the numerical form, conjugate results must have the same attributes as those of their paired results. They must belong to the same Result Case, same data type, same associativity, same layer-position, same nodes/elements, and for element results, computed at the same output location within elements.

Once converted to single form (i.e., real, imaginary, magnitude, phase angle or at a particular phase angle), results are treated the same way as non-complex results.

Layer-Position. For plate or shell elements, results can be computed at a particular location through the thickness. For composite elements, results can be computed for a particular layer, and/or at a particular location in the thickness of a layer. These two through-thickness positions are collapsed into an attribute called layer position. Each layer position specifies a unique layer ID (0 for homogeneous elements) and a unique location within the layer (labelled NON-LAYERED for solids). The labels for layer positions created by the results translators indicate the actual location of the output points.

For homogeneous beams or bars, each layer position corresponds to an output location in the beam section. The layer position attributes contain the actual physical coordinates of the output points. All other layer position coordinates are dimensionless parametric coordinates. For layered beams/bars, the results are treated like plates and shells. The labels for layer positions in beams indicate the actual locations of the output points.

For homogeneous beams or bars, a dummy layer-position is created so that it has a layer position ID for access. Composite solids are treated as composite plates/shells.

The MSC.Patran Results Postprocessor only uses the ID of the layer position to retrieve its associated results. The labels for layer position are transparent to the processor.

Target Nodes and Elements. Each result in MSC.Patran has to be associated with a node or an element ID. A result that is associated with the whole result case is called a global variable, such as time or frequency. Results can be displayed only if their associated nodes/elements exist in the targeted entities for any given plot type. A variety of options exist to specify at which entities to target result plots:

- ID list (i.e., list of nodes and element IDs).
- Range for result values. This filter is based on result values from the database.
- Lists of material properties (element results).
- Lists of element properties.
- Lists of element types.
- Lists that are based on material properties.
- Lists that are based on element properties.
- Lists that are based on element types.
- Lists of points that lie along an arbitrary path

Results in the database can belong to a superset or subset of these ID lists, but only the results that belong to the elements/nodes effected by the intersection of these lists are able to be displayed.

Element Position. Element results can be computed at any point in the element. The location of the output point is part of an attribute called element position. The element position contains, among other things, the parametric coordinates of the output. The result translators create these attributes and assigns them to the element results. It makes no difference to the MSC.Patran Results Processor if this point is one of the element nodes, the element faces, the element edges, the Gaussian quadrature points, or the element centroid. The coordinate system type depends on the element topology.

Topology	Coordinate System Type	Coordinates
Bar/Beams Tria	Parametric Area	s_1 s_1, s_2, s_3
Quad Tet	Parametric Volume	s_1, s_2 s_1, s_2, s_3, s_4
Wedge Hex	Area/Parametric Parametric	s_1,s_2,s_3,s_4 s_1,s_2,s_3

It is important to note that:

1. All coordinates are in range [0..1].
2. For wedges, s_1, s_2, s_3 = area coordinates, s_4 = parametric coordinates.

14.3 Derivations

The following table provides additional definitions for the selected result derivations. These include tensor to vector, tensor to scalar, and vector to scalar resolutions.

Transform Type	Derivation Method	Description
Scalar to Scalar **Vector to Vector** **Tensor to Tensor**	**None**	No transformation is used if the result data type matches the plot tool's data type.
Vector to Scalar	**Magnitude**	Vector magnitude.
	X Component	1st vector component.
	Y Component	2nd vector component.
	Z Component	3rd vector component.
Tensor to Scalar	**XX Component**	XX tensor component.
	YY Component	YY tensor component.
	ZZ Component	ZZ tensor component.
	XY Component	XY tensor component.
	YZ Component	YZ tensor component.
	ZX Component	ZX tensor component.
	Min Principal	Calculated minimum principal magnitude.
	Mid Principal	Calculated middle principal magnitude.
	Max Principal	Calculated maximum principal magnitude.
	1st Invariant	Calculated 1st invariant
	2nd Invariant	Calculated 2nd invariant
	3rd Invariant	Calculated 3rd invariant
	Hydrostatic	Calculated mean of the three normal tensor components.
	von Mises	Calculated effective stress using von Mises criterion.
	Tresca	Calculated Tresca shear stress.
	Max Shear	Calculated maximum shear magnitude.
	Octahedral	Calculated Octahedral shear stress.
Tensor to Vector	**Min Principal**	Calculated minimum principal vector.
	Mid Principal	Calculated middle principal vector.
	Max Principal	Calculated maximum principal vector.

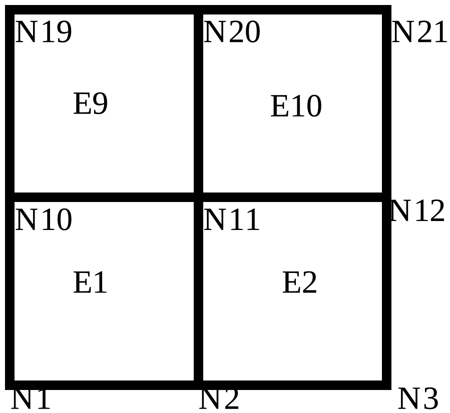
Below are the equations and examples of the derivation methods:

Important: These equations for calculating invariants are not recommended for complex results since phase is not taken into account.

von Mises Stress. von Mises stress is calculated from the following equation:

$$\sigma' = \sqrt{\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2)}{2} + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

Example: The elements shown below have the following stress contributions:



Elem. ID	**Node ID**	σ_x	σ_y	σ_z	τ_{xy}	τ_{yz}	τ_{zx}
1	1	46.2	13.01	0.00	5.13	0.00	0.00
	2	93.39	25.33	0.00	17.45	0.00	0.00
	11	68.37	12.16	0.00	-19.73	0.00	0.00
	10	44.32	10.40	0.00	-1.01	0.00	0.00
2	2	93.39	25.33	0.00	17.45	0.00	0.00
	3	88.67	24.41	0.00	23.95	0.00	0.00
	12	57.42	5.44	0.00	-34.02	0.00	0.00
	11	59.37	10.16	0.00	-20.73	0.00	0.00
9	10	44.32	10.40	0.00	-1.01	0.00	0.00
	11	67.37	11.16	0.00	-18.73	0.00	0.00
	20	4.72	8.15	0.00	-15.28	0.00	0.00
	19	17.99	7.68	0.00	-4.61	0.00	0.00

Elem. ID	Node ID	σ_x	σ_y	σ_z	τ_{xy}	τ_{yz}	τ_{zx}
10	11	100.37	14.16	0.00	-30.73	0.00	0.00
	12	57.42	5.44	0.00	-34.02	0.00	0.00
	21	-5.63	5.72	0.00	-22.03	0.00	0.00
	20	4.72	8.15	0.00	-15.28	0.00	0.00

The von Mises stress calculated at node 11 when nodal averaging is done first due to the contribution from each element is 78.96. When the von Mises derivation is done first and then averaging at the nodes takes place, the calculated von Mises stress is 79.02. Thus a difference can arise depending on whether the averaging is done first or the derivation. This can be true for all derived results.

Node 11	σ_x	σ_y	σ_z	τ_{xy}	τ_{yz}	τ_{zx}	von Mises Stress
E1	68.37	12.16	0.00	-19.73	0.0	0.0	71.82
E2	59.37	10.16	0.00	-20.73	0.00	0.00	65.68
E9	67.37	11.16	0.00	-18.73	0.00	0.00	70.45
E10	100.37	14.16	0.00	-30.73	0.00	0.00	108.10
Average	73.87	11.91	0.00	-22.48	0.00	0.00	79.02
Average then Derive							78.96
Derive then Average							79.02

Important: It must be noted also that for von Mises and other derived results, the calculations are generally valid only for stresses. Although these operations can be performed for any valid tensor or vector data stored in the database, quantities such as tensor strains are not appropriate for von Mises calculations. To calculate a true von Mises strain the strain tensor must be converted to engineering strains by multiplying the shear components by a factor of two.

Octahedral Shear Stress. Octahedral shear stress is calculated from the following equation:

$$\tau_{oct} = \frac{\sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2) + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}}{3}$$

From the von Mises example above the octahedral shear stress is:

Octahedral Shear Stress	Node 11
Average / Derive	37.22
Derive / Average	37.25

Hydrostatic Stress. Hydrostatic stress is calculated from the following equation:

$$\sigma = \frac{\sigma_x + \sigma_y + \sigma_z}{3}$$

From the von Mises example above the hydrostatic stress is:

Hydrostatic Stress	Node 11
Average / Derive	28.59
Derive / Average	28.59

Invariant Stresses. 1st, 2nd, and 3rd invariant stresses are calculated from the following equations:

$$\sigma_{1st} = (\sigma_x + \sigma_y + \sigma_z)$$

$$\sigma_{2nd} = \sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_z\sigma_x - (\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)$$

$$\sigma_{3rd} = \sigma_x(\sigma_y\sigma_z - \tau_{yz}^2) + \tau_{xy}(\tau_{xy}\sigma_z - \tau_{yz}\tau_{zx}) + \tau_{zx}(\tau_{xy}\tau_{yz} - \sigma_x\tau_{zx})$$

From the von Mises example above the invariant stresses are:

Invariant Stresses (Node 11)	1st Invariant	2nd Invariant	3rd Invariant
Average / Derive	85.78	374.44	0.00
Derive / Average	85.78	373.38	0.00

Principal Stresses. Principal stresses are calculated from either a Mohr's circle method for 2D tensors ($\sigma_z = \tau_{yz} = \tau_{zx} = 0$) or from a 3x3 Jacobian Rotation Eigenvector extraction method for a 3D tensors. The User Interface allows for either a tensor-dependent derivation, or a 2D calculation. The tensor-dependent calculation will choose either a 2D or 3D calculation depending on values of each tensor encountered. A 2D calculation will be used when the ZZ, YZ and ZX are exactly zero (which is the case when the analysis code does not calculate these values), otherwise the full 3D tensor will be considered. Both the magnitudes of the principals and their direction cosines are calculated from these routines.

The magnitudes of the two principal stresses from the 2D Mohr's circle method are calculated according the following equations:

$$\sigma_{major} = \sigma_{ave} + \sqrt{(\sigma_x - \sigma_{ave})^2 + \tau_{xy}^2}$$

$$\sigma_{minor} = \left(\sigma_{ave} - \sqrt{(\sigma_x - \sigma_{ave})^2 + \tau_{xy}^2}\right)$$

where:

$$\sigma_{ave} = \frac{(\sigma_x + \sigma_y)}{2}$$

The direction cosines for the 2D Mohr's circle method are calculated by assembling the following 3x3 transformation matrix:

$$\begin{bmatrix} \cos\theta & \sin\theta & 0 \\ -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad \text{where } \theta = \frac{\operatorname{atan}\left(\dfrac{\tau_{xy}}{\sigma_x - \sigma_{ave}}\right)}{2}$$

From the von Mises example above the principal stresses are:

Principal Stresses (Node 11)	Maximum	Minimum
Average / Derive	81.17	4.61
Derive / Average	81.20	4.58

Also the principal stress determinant is the product of the three principals and the major, minor, and intermediate principal deviatoric stresses are calculated from:

$$\sigma_{(maj,\ dev)} = \sigma_{major} - \frac{(\sigma_{major} + \sigma_{inter} + \sigma_{minor})}{3}$$

$$\sigma_{(min,\ dev)} = \sigma_{minor} - \frac{(\sigma_{major} + \sigma_{inter} + \sigma_{minor})}{3}$$

$$\sigma_{(int,\ dev)} = \sigma_{inter} - \frac{(\sigma_{major} + \sigma_{inter} + \sigma_{minor})}{3}$$

Tresca Shear Stress. Tresca shear stress is calculated from the following equation:

$$\tau = (\sigma_{major} - \sigma_{minor})$$

where σ_{major} and σ_{minor} are calculated as mentioned under Principal stress derivations above. From the von Mises example above the Tresca shear stress is:

Tresca Shear Stress	Node 11
Average / Derive	76.55
Derive / Average	76.61

Maximum Shear Stress. Maximum shear stress is calculated from the following equation

$$\tau = \frac{(\sigma_{major} - \sigma_{minor})}{2}$$

where σ_{major} and σ_{minor} are calculated as mentioned under Principal stress derivations above. From the von Mises example above the Tresca shear stress is:

Tresca Shear Stress	Node 11
Average / Derive	76.55
Derive / Average	76.61

Magnitude. Magnitude (vector length) is calculated from the components with the standard formula:

$$magnitude = \sqrt{x^2 + y^2 + z^2}$$

14.4 Averaging

For Fringe and other plots and reports that must display or report values at nodes from elemental data regardless of where the element results are computed, must be converted to results at element nodes. The interpolation functions are then used (e.g., by the graphics module for fringe plot and other operations) to compute the results at any point within the element. The interpolation functions may or may not be the shape functions that were used by the analysis program to compute the element results.

As a rule, each element sharing a common node has its own result values. To compute results for continuous fringe plots, these values need to be averaged and distributed to the sharing elements. The options for the averaging process are described below:

No Averaging	Each element retains its value at the element nodes. Or in other words, each element is its own averaging domain. This selection from the Averaging Domain pull down is called None. The fringe plot will have jumps (not continuous regions) at element boundaries.
Averaging Based on All Entities	All elements will contribute to the sum and will receive the averaged result regardless of whether only certain entities have been selected for the display of the fringe plot. All surrounding elements will contribute to the averaging process.
Averaging Based on Target Entities	Only the elements defined as the target entities will contribute to the sum and will receive the averaged result. Surrounding elements that are not part of the target entities will not contribute to the averaging process.
Averaging Based on Materials	Elements with the same material IDs will contribute to the sum and will receive the averaged result. The fringe plot will have jumps at material boundaries.
Averaging Based on Properties	Elements with the same property IDs will contribute to the sum and will receive the averaged result. The fringe plot will have jumps at property boundaries.
Averaging Based on Element Types	Elements of the same type will contribute to the sum and will receive the averaged result. The fringe plot will have jumps at element type boundaries.
Difference	The minimum and maximum results from the elements sharing a common node are computed. The difference is determined as the delta between the maximum and minimum contributor to each node. The fringe plot of this max difference indicates the quality of the mesh and the location where this mesh needs to be refined by comparing its values with the actual values of the results. Nodal results will have zero max-difference.
Sum	The sum of all contributing nodes will be displayed. This step skips the averaging.

Below are some examples of the averaging techniques. The model in Figure 14-1 is used for illustration purposes. It consists of 8 QUAD4 elements and 4 TRI3 elements with a total of 17 nodes.

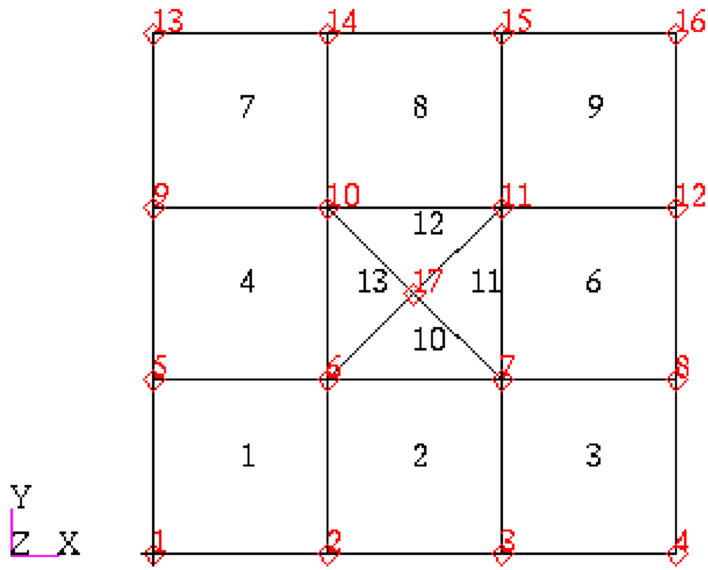


Figure 14-1 Square Plate Model to Illustrate Averaging Techniques.

The above model is also broken up into various material and property sets as such:

Prop1	**Mat1**	Elem 1:3
Prop2	**Mat2**	Elem 6 8:9
Prop3	**Mat3**	Elem 4 7
Prop4	**Mat1**	Elem 10:13
Target1		Elem 1:3 6 10:11
Target2		Elem 4 7:9 12:13

Element Centroidal Results. The first illustration is the simple case of results at element centroids. Table 14-1 below lists some scalar values of strain energy at each element centroid. The table is listed by node number with each element and corresponding strain energy value for all contributing elements associated with the particular nodes. The averaging domain columns on the right then list the averaged values for each node based on the averaging domain. Columns with more than one value per node indicate a boundary of the averaging domain and will therefore cause a plot discontinuity across boundaries. See Figure 14-2 for visual effects of averaging domains.

Table 14-1 Averaging at Nodes from Element Centroidal Results

Node	Element	Strain Energy	Averaging Domain					
			All	Property	Material	None	Type	Target
1	1	3.01	3.01	3.01	3.01	3.01	3.01	3.01
2	1	3.01	3.89	3.89	3.89	3.01	3.89	3.89
	2	4.78				4.78		
3	2	4.78	3.97	3.97	3.97	4.78	3.97	3.97
	3	3.16				3.16		
4	3	3.16	3.16	3.16	3.16	3.16	3.16	3.16
5	1	3.01	8.04	3.01		3.01	8.04	3.01
	4	13.06		13.06		13.06		13.06
6	1	3.01	4.24	3.89	2.04	3.01	6.95	2.63
	2	4.78				4.78		
	4	13.06		13.06	13.06	13.06		6.67
	10	0.10		0.19	2.04	0.10	0.19	2.63
	13	0.27				0.27		6.67
7	2	4.78	2.09	3.97	2.04	4.78	3.42	2.09
	3	3.16				3.16		
	6	2.31		2.31	2.04	2.31		
	10	0.10		0.11	2.04	0.10	0.11	
	11	0.11				0.11		
8	3	3.16	2.74	3.16	3.16	3.16	2.74	2.74
	6	2.31		2.31	2.31	2.31		
9	4	13.06	12.10	12.10	12.10	13.06	12.10	12.10
	7	11.13				11.13		

Table 14-1 Averaging at Nodes from Element Centroidal Results (continued)

Node	Element	Strain Energy	Averaging Domain					
			All	Property	Material	None	Type	Target
10	4	13.06	5.95	12.01	12.01	13.06	9.74	5.95
	7	11.13				11.13		
	8	5.02		5.02	5.02	5.02		
	12	0.27		0.27	0.27	0.27	0.27	
	13	0.27				0.27		
11	6	2.31	2.11	3.38	3.38	2.31	3.38	1.21
	8	5.02				5.02		2.70
	9	2.82				2.82		
	11	0.11		0.19	0.19	0.11	0.19	1.21
	12	0.27				0.27		
12	6	2.31	2.57	2.57	2.57	2.31	2.57	2.31
	9	2.82				2.82		2.82
13	7	11.13	11.13	11.13	11.13	11.13	11.13	11.13
14	7	11.13	8.08	11.13	11.13	11.13	8.08	8.08
	8	5.02		5.02	5.02	5.02		
15	8	5.02	3.92	3.92	3.92	5.02	3.92	3.92
	9	2.82				2.82		
16	9	2.82	2.82	2.82	2.82	2.82	2.82	2.82
17	10	0.10	0.19	0.19	0.19	0.10	0.19	0.10
	11	0.11				0.11		
	12	0.27				0.27		0.27
	13	0.27				0.27		

Element Nodal Results. The second illustration is the more complex case of results at element nodes. Table 14-2 below is listed by element number with each node and corresponding von Mises stress for all nodes associated with the particular element. This case is identical to the element centroid case with the exception that each node can have a different value for each contributing element. In this example von Mises stress is derived first and then averaged. See Figure 14-2 for visual effects of averaging domains.

Table 14-2 Averaging at Nodes from Element Nodal Results

Element	Node	von Mises Stress	Averaging Domain					
			All	Property	Material	None	Type	Target
1	1	266353	266353	266353	266353	266353	266353	266353
	2	205495	236621	236621	236621	205495	236621	236621
	6	194627	238950	263404	240096	194627	265783	209085
	5	251128	330989	251128	251128	251128	330989	251128
2	2	267747	236621	236621	236621	267747	236621	236621
	3	269673	247874	247874	247874	269673	247874	247874
	7	288631	213334	254218	199024	288631	259671	213334
	6	287859	238950	263404	240096	287859	265783	209085
3	3	226076	247874	247874	247874	226076	247874	247874
	4	223550	223550	223550	223550	223550	223550	223550
	8	216967	224325	216967	216967	216967	224325	224325
	7	219806	213334	254218	199024	219806	259671	213334
4	5	410849	330989	410849	410849	410849	330989	410849
	6	314864	238950	314864	314864	314864	265783	283747
	10	316307	310705	326528	326528	316307	350090	310705
	9	409360	381243	381243	381243	409360	381243	381243
6	7	270577	213334	270577	270577	270577	259671	213334
	8	231683	224325	231683	231683	231683	224325	224325
	12	231124	264210	264210	264210	231124	264210	231124
	11	269415	265760	311763	311763	269415	311763	206152
7	9	353127	381243	381243	381243	353127	381243	381243
	10	336749	310705	326528	326528	336749	350090	310705
	14	331970	361658	331970	331970	331970	361658	361658
	13	351258	351258	351258	351258	351258	351258	351258

Table 14-2 Averaging at Nodes from Element Nodal Results (continued)

Element	Node	von Mises Stress	Averaging Domain					
			All	Property	Material	None	Type	Target
8	10	397215	310705	397215	397215	397215	350090	310705
	11	389998	265760	311763	311763	389998	311763	305499
	15	384259	346068	346068	346068	384259	346068	346068
	14	391346	361658	391346	391346	391346	361658	361658
9	11	275878	265760	311763	311763	275878	311763	305499
	12	297297	264210	264210	264210	297297	264210	297297
	16	331799	331799	331799	331799	331799	331799	331799
	15	307878	346068	346068	346068	307878	307878	346068
10	6	144769	238950	198700	240096	144769	198700	209085
	7	144769	213334	143829	199024	144769	143829	213334
	17	144769	197728	197728	197728	144769	197728	143829
11	7	142890	213334	143829	199024	142890	143829	213334
	11	142890	265760	196756	196756	142890	196756	206152
	17	142890	197728	197728	197728	142890	197728	143829
12	11	250623	265760	196756	196756	250623	196756	305499
	10	250623	310705	251626	251626	250623	251626	310705
	17	250623	197728	197728	197728	250623	197728	251627
13	10	252631	310705	251626	251626	252631	251626	310705
	6	252631	238950	198700	240096	252631	198700	283747
	17	252631	197728	197728	197728	252631	197728	251627

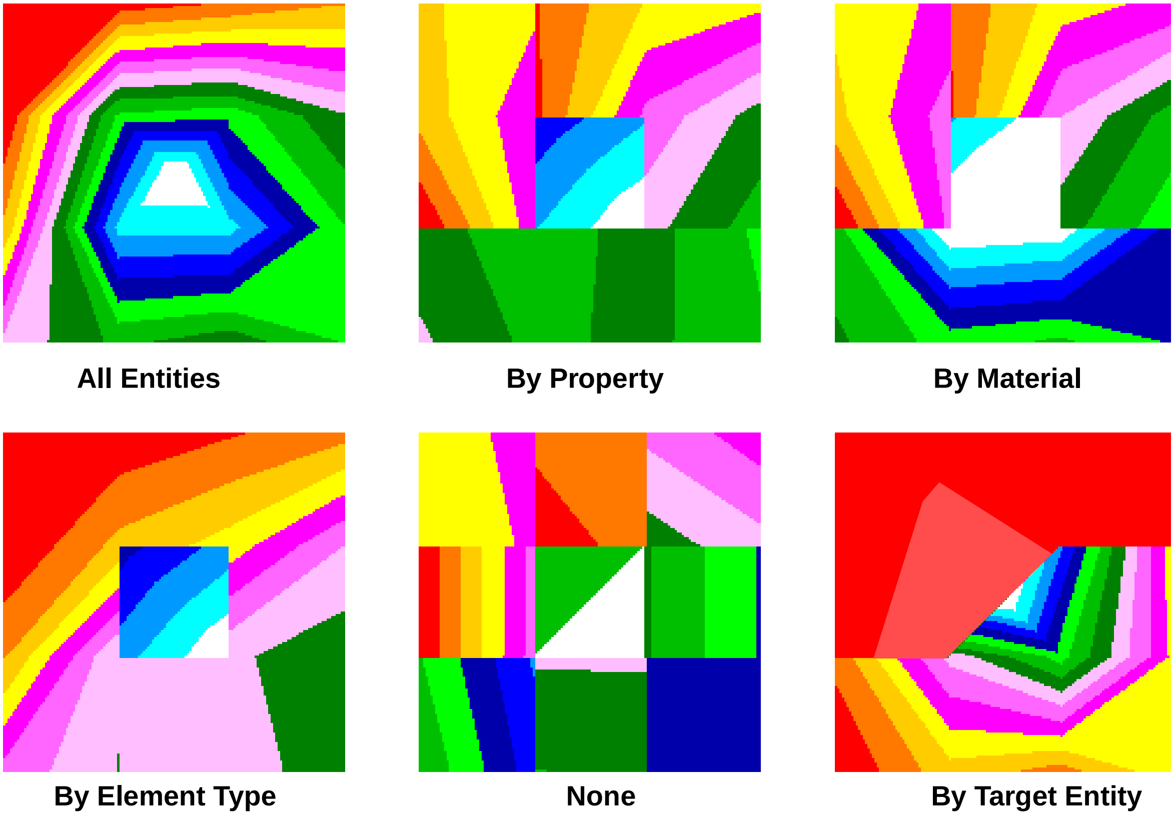


Figure 14-2 Differences in Plots Due to Averaging Domains - Note Discontinuities.

14.5 Extrapolation

When element results are provided to MSC.Patran at quadrature points, it is necessary to extrapolate the results from the quadrature points to the nodes of the element and to the element centroid. Similarly, when results are provided at the element nodes or the centroid, it is necessary to interpolate/extrapolate the results to the centroid or nodes respectively.

MSC.Patran has three basic methods to perform this interpolation/extrapolation:

- By parametric mapping method.
- By solving a set of equations.
- By averaging.

The User Interface allows for four basic methods in which the user can control extrapolation methods. These are explained below and examples given.

Shape Function. If the arrangement of node/quadrature points corresponds to an element type in MSC.Patran, the shape functions are known, and a parametric mapping is used. This is the preferred method, and is the most accurate representation. The parametric mapping method involves mapping the output positions to an element topology that interpolation functions of that topology can be used to compute results at the nodes. As an example, if there are 27 results output at 27 quadrature points inside a hex/20, then these 27 quadrature points can be considered as 27 vertices of a hex/27 element. Results at hex/20 nodes are then computed by the interpolation function of the hex/27, even though these nodes are located outside the element formed by the 27 quadrature points. Once the nodal results of the hex/27 are available, results at the nodes of the hex/20 can be computed by interpolation. These results will be stored in a 20x27 matrix of coefficients. This method only works if there exists an element topology in the library that coincides with the output pattern after being parametrically mapped.

If the arrangement does not correspond to a MSC.Patran element type, a system of equations is constructed and solved for the unknown nodal and centroidal values. The equations are set up such that if the interpolation functions of the element topology are used with the unknown nodal values, they will generate a unit value at each quadrature point. This method only works if there exists an element topology in the library that has the same number of nodes as the number of quadrature points. If Shape Function is set in the User Interface ,the shape functions or a set of equations will be used to extrapolate results as explained above. Only if these two methods fail will averaging take place.

Average. If both previous methods fail, results in the element are averaged and each node of the element will assume this averaged value. Or, alternatively, if the results are provided at nodes, the nodal values would be averaged and assigned to the centroid.

Averaging is also used at element boundaries. In these cases, when extrapolation from different elements yields different result values at the same node, the different results are averaged and assigned to the node.

For degenerate elements, the extrapolation is performed on the parent element topology, and the results at the duplicated nodes in the degenerate element are then averaged.

The User Interface allows for a forced average extrapolation method to be used. The following scenarios can exist.

- Nodal values to centroid
- Gauss values to nodes
- Centroidal values to nodes

Centroid. A forced extrapolation of the analysis results to the element's centroid can also be set in the User Interface which will be performed relative to where the results are initially located. Shown below are several different cases that can occur. Once each centroid value is established it is then used to render the results plot.

- Centroid values to element centroid
- Nodal values to element centroid
- Gauss values to element centroid

Min/Max. The Min / Max method searches each element's results and finds the minimum / maximum value contained within the element. The element then assumes a constant value (including its nodes). For example if the analysis result values are know at the elements Gauss points the minimum / maximum value is used as a constant value across the element. This method has no effect for results that already exist at the element centroid or the nodes.

Examples. Examples are given below for each extrapolation technique using a simple 4 node QUAD element with four interior Gauss points. The Gauss points are located in parametric space at +/- 0.5773502692 (as per theory). In p/q parametric space, where the extrapolation occurs, would look something like Figure 14-3.

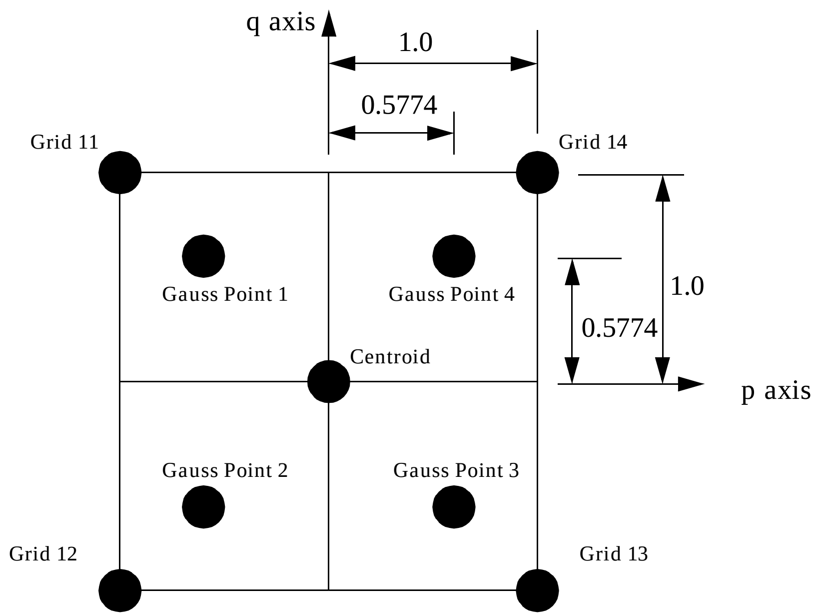


Figure 14-3 Example 4 Noded QUAD with Gauss Points.

The element will have a simple set of linear shape functions described by

$$N_1 = -(p-1)(q+1)$$

$$N_2 = (p-1)(q-1)$$

$$N_3 = -(p+1)(q-1)$$

$$N_4 = (p+1)(q+1)$$

Using these shape functions, the results at any point in the element would be found as

$$Result(p,q) = \sum N_i(p,q) \times Result_i$$

where i runs from 1 to 4 for the four Gauss or grid points.

Note that the shape functions vary by element type and element order. The function shown in these examples are not necessarily the functions used in any particular element formulation; they are to illustrate the extrapolation methods only.

Example 1 - Parametric Mapping (Gauss points to element nodes)

Gauss point results are as follows:

Gauss Point	Stress
1	10.
2	15.
3	20.
4	15.

The stress values at the Gauss points will be extrapolated to the grid locations. To do this, the Gauss points are assigned parametric locations of 1.0. The location of the grids will be at parametric locations of 1/0.5774 or about +/-1.7319 with respect to the Gauss points.

The stress at grid 14, located in parametric space at x/y coordinates of (1.7319, 1.7319) will be calculated as:

$$-\frac{1}{4}(1.7319-1)(1.7319+1)\times 10+\frac{1}{4}(1.7319-1)(1.7319-1)\times 15+-\frac{1}{4}(1.7319+1)(1.7319-1)\times 20 + \frac{1}{4}(1.7319+1)(1.7319+1)\times 15 \;=\; 15.00$$

The stresses at the rest of the grids would be as follows:

Grid	X Location	Y Location	Stress
11	-1.7319	1.7319	6.340499
12	-1.7319	-1.7319	15.00
13	1.7319	-1.7319	23.65950
14	1.7319	1.7319	15.00

Example 2 - Parametric Mapping (Gauss points to element centroid)

The stress at the Gauss points are the same as Example 1. The element centroid would be located in parametric space at (0,0), so interpolation to that point can be accomplished directly:

$$-\frac{1}{4}(0-1)(0+1)\times 10+\frac{1}{4}(0-1)(0-1)\times 15+-\frac{1}{4}(0+1)(0-1)\times 20+\frac{1}{4}(0+1)(0+1)\times 15 \;=\; 15.00$$

Example 3 - Parametric Mapping (Nodal results to element centroid)

In this example the results at the grid points are provided to MSC.Patran. To make an element fill plot, the element centroidal value must be known. The stress values at the element grid points are:

Gauss Point	Stress
1	6.340499
2	15.00
3	23.65950
4	15.00

The value at the centroid is then calculated using the shape functions, just as in Example 2 above:

$$-\frac{1}{4}(0-1)(0+1)\times 10+\frac{1}{4}(0-1)(0-1)\times 15+-\frac{1}{4}(0+1)(0-1)\times 20+\frac{1}{4}(0+1)(0+1)\times 15 = 15.00$$

Note that this gives the same results as in the previous example.

Example 4 - Averaging (Nodal results to element centroid)

The averaging technique simply computes the mathematical average of the nodal stresses and reports this as the centroidal value. So, the centroidal stress would be reported as:

$$(6.340499+15+23.65950+15)/4 = 15.00$$

Example 5 - Averaging (Gauss points to element nodes)

In this case no suitable set of shape functions exists to carry out a proper interpolation. Therefore, the Gauss point stresses are averaged, and the average result distributed to all the grid points:

$$(10+15+20+15)/4 = 15.00$$

The grid point stresses would be reported as:

Grid Point	Stress
11	15.00
12	15.00
13	15.00
14	15.00

Example 6 - Averaging (Centroidal values to element nodes)

In this case there is only one piece of stress data available, so no assumptions about the stress distribution can be made. Therefore, if the element centroid stress is reported as 15.00, the grid point stress will be reported as:

Grid Point	Stress
11	15.00
12	15.00
13	15.00
14	15.00

Example 7 - Averaging (Adjacent element contributions)

In this case the stresses in an adjacent element are included in the reporting of the grid point stress. If two elements have nodal stresses calculated from Gauss points by internal extrapolation as follows:

Element 1		Element 2	
Grid Point	Stress	Grid Point	Stress
11	6.340499	13	27.50
12	15.00	14	17.50
13	23.65950	15	10.00
14	15.00	16	9.50

The nodal stresses calculated by MSC.Patran would be:

Grid Point	Stress
11	6.340499
12	15.00
13	25.5798 = [(23.65650 + 27.50) / 2]
14	16.25 = [(15.00 + 17.50) / 2]
15	10.00
16	9.50

14.6 Coordinate Systems

Results are stored in the MSC.Patran database in a variety of ways. They are also transformed, either automatically or by the user when necessary, to create meaningful plots. It is important to understand each of these coordinate systems and know in what coordinate system results are stored and whether any transformations are being performed prior to graphical display.

Vectors are transformed as:

$$v = [R]U$$

where **v** is a vector referenced in the local coordinate system defined by the rotation matrix [R], each row of which defines a unit vector in the global system. **U** is a vector referenced in the global system. For example, if the global system is rotated $\theta°$ about the z axis, the rotation matrix of the new system is:

$$[R] = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 \\ -\sin(\theta) & \cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The inverse vector transformation is:

$$U = [R]^T v$$

which transforms a vector result defined in the [R] system to the global system, since [R] is an orthonormal matrix by definition.

Similarly, the tensors are transformed as:

$$\sigma = [R]S[R]^T$$

where **S** is a tensor in the global system, and σ is the tensor in the [R] system.

The inverse tensor transformation is:

$$S = [R]^T \sigma [R]$$

which transforms a tensor σ in [R] system to a tensor **S** in the global system.

For nodal results, the coordinate system types are:

Global system	Type=0	ID=0
Nodal system	=1	=0
User system	=3	=Assigned

For element results, the coordinate system types are:

Global system	Type=0	ID=0
Element system	=2	=0
User system	=3	=Assigned
Material system	=4	=0
Ply system	=5	=0

Global System. This is the MSC.Patran global or default rectangular coordinate system. For MSC.Nastran users, this is the same as the MSC.Nastran basic coordinate system. Most alternate coordinate systems use the global system as a basis.

Local Systems. These are MSC.Patran local coordinate systems specifically created by the user within MSC.Patran. They can be either rectangular, cylindrical, or spherical. These are the same as MSC.Nastran global coordinate systems in MSC.Nastran terminology. Do not be confused by this terminology. Just remember that user defined systems in MSC.Patran are called local systems and user defined systems in MSC.Nastran are called global systems. The default coordinate system in MSC.Patran is called the global system and the default system in MSC.Nastran is called the basic system.

Reference Systems. These are local systems or the global system by which geometric definitions are defined. For instance the coordinates locations of a finite element node is defined by referring to a reference system, either local or global.

Analysis Systems. These are the local systems in which results at finite element nodes will be calculated by the analysis solver. Nodes can be defined in one system (the reference system) but results calculated in another (the analysis system). In general, when nodal results are imported into the MSC.Patran database, they will be stored in the analysis systems.

Unknown Systems. These are systems which are unknown to MSC.Patran and therefore must remain in these systems when postprocessing. No transformation are allowed.

Element Systems. These are coordinate systems local to each specific element. There are many types of element coordinate systems. Suffice it to say here, that when elemental based results calculated in an elemental system are imported into the MSC.Patran database, the coordinate systems in which they are stored vary from element to element. This makes meaningful graphical visualization of these results quite difficult. Many times a coordinate transformation is required to convert all results into a consistent coordinate system. Once this is done then operations such as nodal averaging and scalar results derivations (von Mises) can be performed correctly and meaningfully.

Projected Global System. This is one system used to convert and display element based shell and plate data stored in an element systems into a consistent, meaningful plot.

The projected global system is defined as follows: First, the normal to the shell surface is calculated. This varies for curved elements and is constant for flat elements. If the angle between the normal and the global x-axis is greater than 0.01 radians, the global x-axis is projected onto the shell surface as the local x-axis. If the angle is less than 0.01 radians, either the global y-axis or the z-axis (whichever makes the largest angle with the normal) is defined to be the local x-axis. The local y-axis is perpendicular to the plane defined by the normal and the local x-axis. The projected z-axis will align with the element normal.

For one dimensional (1D) and three dimensional (3D) elements, the projected global system is the global system and therefore no projection is performed.

This system has been set as the default for viewing fringe and other plots of element based vector and tensor components on two dimensional (2D) elements. It provides a system with real-world significance which is consistent from element to element.

Projected Systems. These are systems like the projected global system but instead project other coordinate systems other than the global onto the elements. An example is the shell p-elements of MSC.Nastran which use a convective system which is a project of a coordinate system onto the element (plus an optional flip and rotation):

$$\vec{k} = \text{element normal at poing of projection}$$

$$\vec{j} = \vec{k} \times \vec{P}_{axis}$$

$$\vec{i} = \vec{j} \times \vec{k}$$

If projected axis is parallel to element normal, the axis of greatest projection is used.

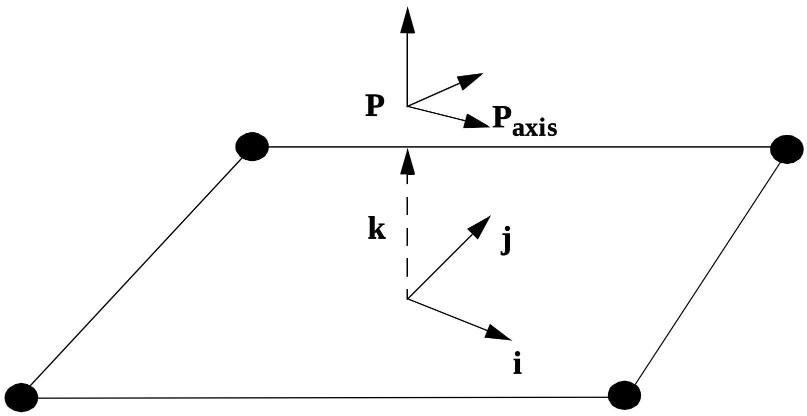


Figure 14-4 Projected Coordinate System Definition.

MSC.Patran Element IJK. These are MSC.Patran defined element coordinate systems. Many analysis translators will convert results from code specific element coordinate systems to a consistent MSC.Patran element IJK coordinate system. These systems differ from element topology to element topology.The IJK system is defined as follows:

$$\vec{i} = \vec{V_1}$$

$$\vec{k} = \vec{V_1} \times \vec{V_2}$$

$$\vec{j} = \vec{k} \times \vec{i}$$

$$\vec{V_1} = V_{1-2}$$

$$\vec{V_2} = V_{1-3}$$

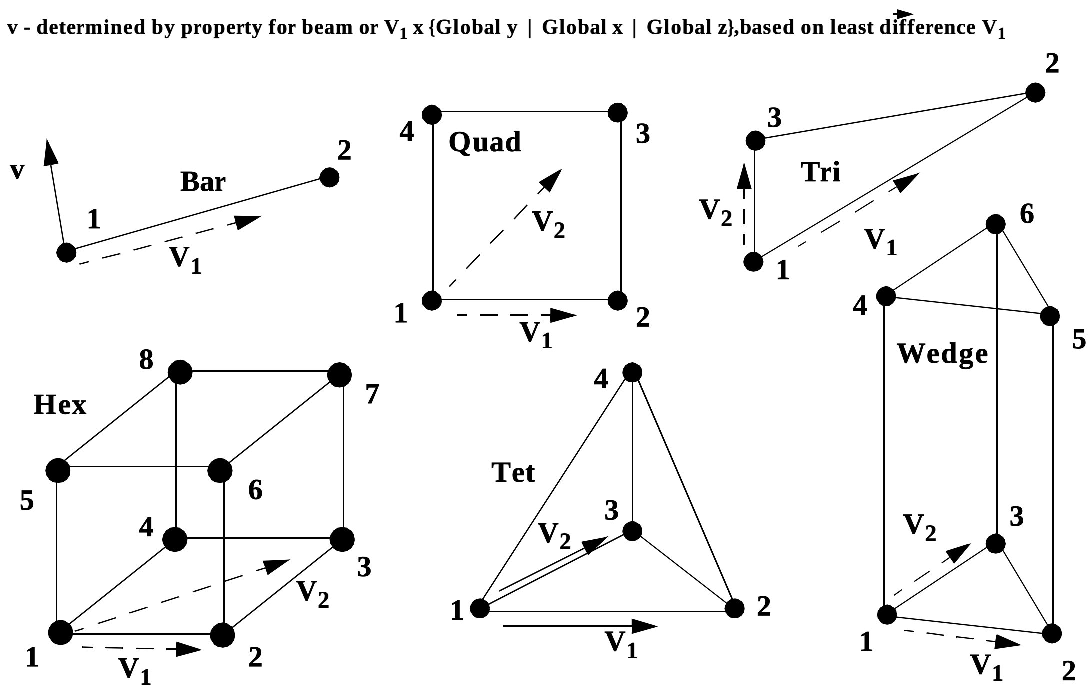


Figure 14-5 MSC.Patran Element IJK Coordinate System Definitions.

Element Bisector (CQUAD4). This element coordinate systems, supported by MSC.Patran, is specific to the MSC.Nastran CQUAD4 element. Other element types default to the IJK system. The definition of the bisector system is as follows:

$$\vec{i} = \vec{V_1} + \vec{V_2}$$

$$\vec{j} = \vec{V_1} - \vec{V_2}$$

$$\vec{k} = \vec{i} \times \vec{j}$$

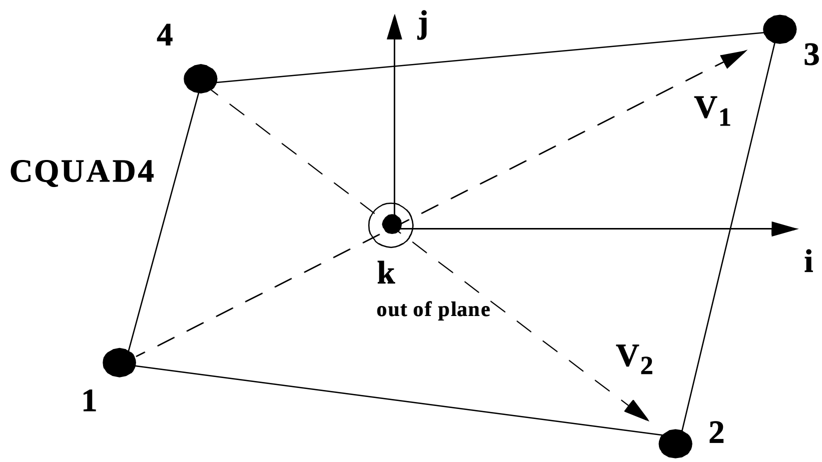


Figure 14-6 CQUAD4 Bisector Coordinate System Definition.

Material Systems. These are element coordinate systems based on a material definition and angle. These exist for QUAD and TRI elements only. Material coordinate systems are defined as follows:

$$\vec{i} = \vec{V_1} \ \text{rotated} \ \alpha \ \text{about} \ \vec{k}$$

which is rotated around a degrees about k. a is from the material property record.

$$\vec{j} = \vec{k} \times \vec{i}$$

The k vector is the same as that for bisector (QUAD element) or IJK (TRI element).

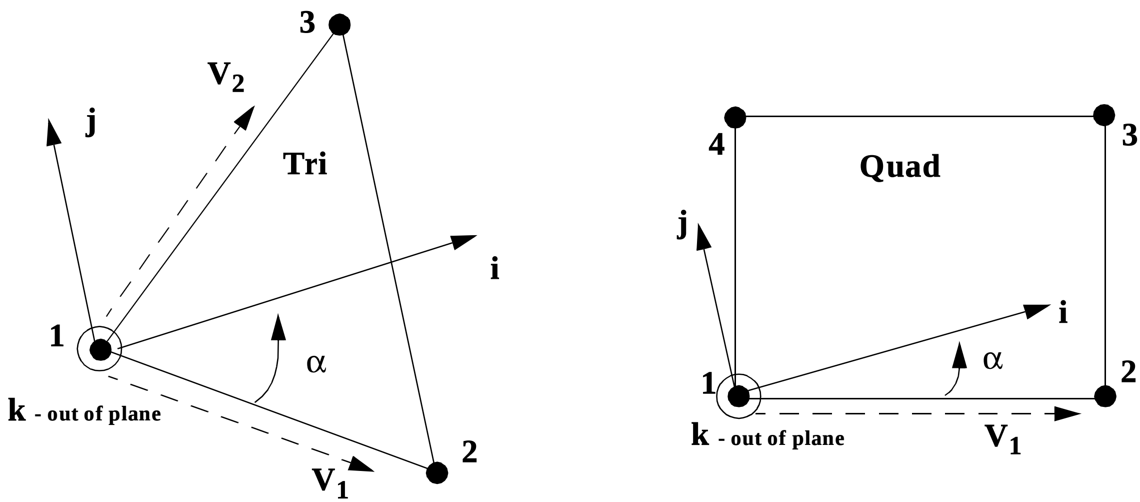


Figure 14-7 MSC.Patran Element IJK Coordinate System Definitions.

MSC.Nastran CQUAD8 System. This element coordinate systems, supported by MSC.Patran, is specific to the MSC.Nastran CQUAD8 element. Other element types default to the IJK system. This coordinate system is position dependent. The definition of this system is as follows:

$$\vec{n} = \text{element normal at a position}$$

$$\vec{t}_1 = \text{tangent 1}, \vec{t}_2 = \text{tangent 2}$$

Use bisections:

$$\vec{b}_1 = \vec{t}_1 + \vec{t}_2$$

$$\vec{b}_2 = \vec{n} + \vec{b}_1$$

So the element system is:

$$\vec{i} = \vec{j} \times \vec{k}$$

$$\vec{j} = \vec{b}_1 + \vec{b}_2$$

$$\vec{k} = \vec{n}$$

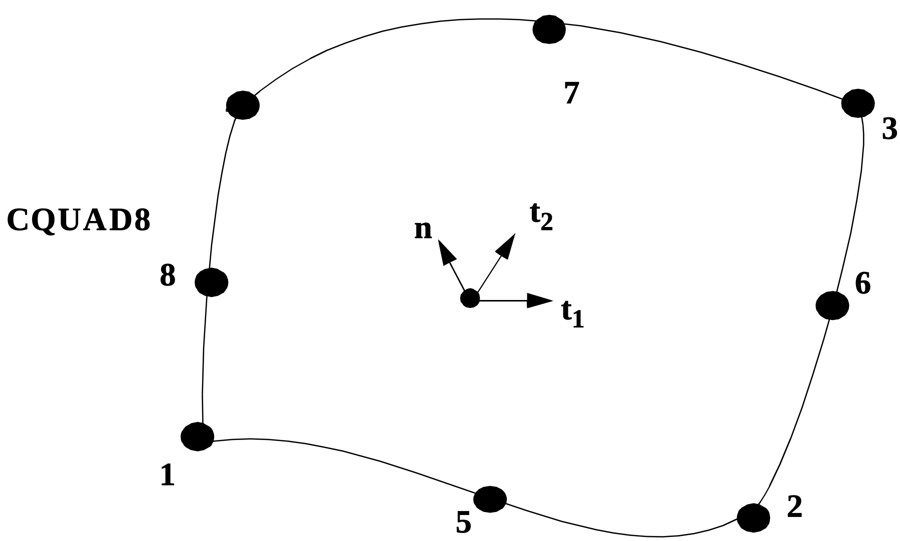


Figure 14-8 MSC.Nastran CQUAD8 Coordinate System Definition.

MSC.Nastran CTRIA6 System. This element coordinate systems, supported by MSC.Patran, is specific to the MSC.Nastran CTRIA6 element. Other element types default to the IJK system. This coordinate system is position dependent. The definition of this system is as follows:

$$\vec{n} = \text{element normal at a position}$$

$$\vec{t}_1 = \text{tangent 1}$$

So the element system is:

$$\vec{i} = \vec{t}_1$$

$$\vec{j} = \vec{n} \times \vec{i}$$

$$\vec{k} = \vec{n}$$

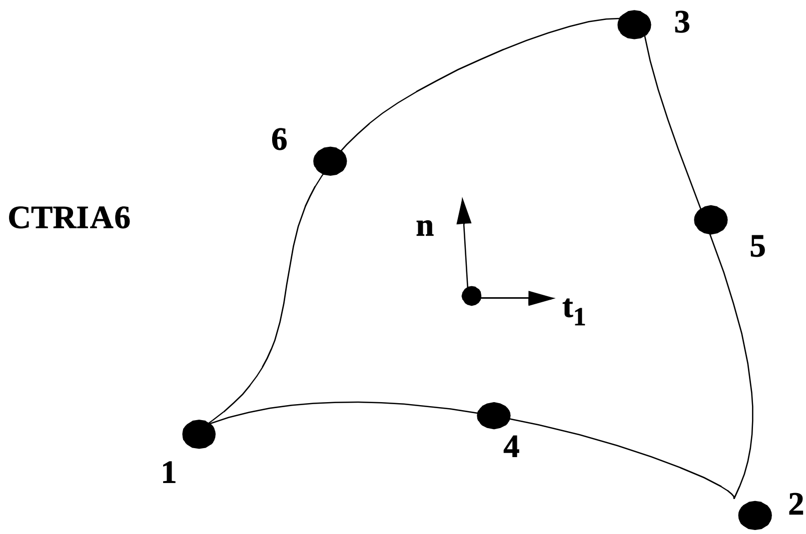


Figure 14-9 MSC.Nastran CTRIA6 Coordinate System Definition.

CHAPTER 15

Verification and Validation

- Overview
- Validation Problems

15.1 Overview

The purpose of this chapter is to demonstrate the correctness of result postprocessing plots, graphs, reports, and result combinations and derivations in MSC.Patran. A number of representative models are used to illustrate this validation and verification. In most cases very simple models are used for simplicity and clarity. Many of these models have closed form solution which are compared to the finite element solutions as displayed in MSC.Patran. The main purpose of this chapter, however, is to show that MSC.Patran correctly displays results as reported and calculated by the finite element solver. It is not to prove that any particular finite element solver calculates the correct answers.

Care has been taken to ensure that as much functionality as possible is covered by these examples. As new postprocessing capabilities are added to MSC.Patran from release to release, this chapter will increase its scope. Each problem gives a description of pertinent information that any engineer should be aware of when postprocessing results. These include:

- ❑ Solver and solution type
- ❑ Element type
- ❑ Model and result descriptions
- ❑ Postprocessing features covered

All problem files have been provided with the MSC.Patran delivery and are indicated with each problem if the user wishes to investigate them personally. below gives a brief description of each problem. These files are located in a directory in the main MSC.Patran installation directory called:

```
<install_dir>/results_vv_files
```

Table 15-1. V&V Problems

Problem	Description	Features Validated
Problem 1: Linear Statics, Rigid Frame Analysis	MSC.Nastran, Solution 101, Linear Statics, Deformation and Fringe Plots of Displacement Results.	prob001.bdf prob001.op2
Problem 2: Linear Statics, Cross-Ply Composite Plate Analysis	MSC.Nastran, Solution 101, Linear Statics, Fringe Plots of Cross-Ply Stress Results with Coordinate Transformation to Global System.	prob002.bdf prob002.op2
Problem 3: Linear Statics, Principal Stress and Stress Transformation	MSC.Nastran, Solution 101, Linear Statics, Fringe and Tensor Plots of Stress Tensor and Principal Stresses in Beam.	prob003.bdf prob003.op2
Problem 4: Linear Statics, Plane Strain with 2D Solids	MSC.Nastran, Solution 101, Linear Statics, Displacement Plots and Fringe Plots of Stress and Displacement with Local Coordinate Transformations.	prob004.bdf prob004.op2

Table 15-1. V&V Problems

Problem	Description	Features Validated
Problem 5: Linear Statics, 2D Shells in Spherical Coordinates	MSC.Nastran, Solution 101, Linear Statics, Deformation and Fringe Plots of Stress with Spherical Coordinate Transformations.	prob005.bdf prob005.op2
Problem 6: Linear Statics, 2D Axisymmetric Solids	MSC.Nastran, Solution 101, Linear Statics, Deformation and Fringe Plots of Stress Using Axisymmetric Elements.	prob006.bdf prob006.op2
Problem 7: Linear Statics, 3D Solids and Cylindrical Coordinate Frames	MSC.Nastran, Solution 101, Linear Statics, Repeat of Problem 6 Using Solid Elements and Cylindrical Frame.	prob007.bdf prob007.op
Problem 8: Linear Statics, Pinned Truss Analysis	MSC.Nastran, Solution 101, Linear Statics, Displacement Plots and Fringe Plots of Deflection, Stress and Rod Forces.	prob008.bdf prob008.op2
Problem 9: Nonlinear Statics, Large Deflection Effects	MSC.Nastran, Solution 106, Non-Linear Statics, Large Displacements, Displacement Fringe Plots of Deflection of a Plate Modeled with CQUAD4 and CQUAD8, CTRIA3 and CTRIA6 Element Types.	prob009_Q4.bdf prob009_Q4.op2 prob009_Q8.bdf prob009_Q8.op2 prob009_T3.bdf prob009_T3.op2 prob009_T6.bdf prob009_T6.op2
Problem 10: Linear Statics, Thermal Stress with Solids	MSC.Nastran, Solution 101, Linear Statics, Fringe Plots Stress due to Thermal Gradient.	prob010.bdf prob010.op2
Problem 11: Superposition of Linear Static Results	MSC.Nastran, Solution 101, Linear Statics, Fringe Plots of Deflection and Linear Superposition.	prob011_A.bdf prob011_A.op2 prob011_B.bdf prob011_B.op2 prob011_C.bdf prob011_C.op2
Problem 12: Nonlinear Statics, Post-Buckled Column	MSC.Nastran, Solution 106, Non-Linear Statics, Displacement, Fringe and Vector Plots of Deflection.	prob012.bdf prob012.op2
Problem 13: Nonlinear Statics, Beams with Gap Elements	MSC.Nastran, Solution 106, Non-Linear Statics, Displacement and Fringe Plots of Deflection.	prob013.bdf prob013.op2
Problem 14: Normal Modes, Point Masses and Linear Springs	MSC.Nastran, Solution 103, Normal Modes, Vector Plots of Mode Shapes.	prob014.bdf prob014.op2

Table 15-1. V&V Problems

Problem	Description	Features Validated
Problem 15: Normal Modes, Shells and Cylindrical Coordinates	MSC.Nastran, Solution 103, Normal Modes, Deformation and Fringe Plots of Mode Shapes using h-Element Formulation.	prob015.bdf prob015.op2
Problem 16: Normal Modes, Pshells and Cylindrical Coordinates	MSC.Nastran, Solution 103, Normal Modes, Deformation and Fringe Plots of Mode Shapes using p-Element Formulation.	prob016.bdf prob016.op2
Problem 17: Buckling, shells and Cylindrical Coordinates	MSC.Nastran, Solution 105, Buckling, Deformation and Fringe Plot of Buckled Cylinder.	prob017.bdf prob017.op2
Problem 18: Buckling, Flat Plates	MSC.Nastran, Solution 105, Buckling, Deformation and Fringe Plot of Buckled Plate.	prob018.bdf prob018.op2
Problem 19: Direct Transient Response, Solids and Cylindrical Coordinates	MSC.Nastran, Solution 109, Direct Transient Response, Graph and Fringe Plots of Responses.	prob019_T.bdf prob019_T.op2 prob019_S.bdf prob019_S.op2
Problem 20:Modal Transient Response with Guyan Reduction and Bars, Springs, Concentrated Masses and Rigid Body Elements	MSC.Nastran, Solution 112, Modal Transient Response, Graph and Plots of Responses.	prob020.bdf prob020.op2
Problem 21: Direct Nonlinear Transient, Stress Wave Propagation with 1D Elements	MSC.Nastran, Solution 129, Nonlinear Transient Response, Graph Plots of Responses.	prob021.bdf prob021.op2
Problem 22: Direct Nonlinear Transient, Impact with 1D, Concentrated Mass and Gap Elements	MSC.Nastran, Solution 129, Nonlinear Transient Response, Graph Plots of Responses.	prob022.bdf prob022.op2
Problem 23: Direct Frequency Response, Eccentric Rotating Mass with Variable Damping	MSC.Nastran, Solution 108 Direct Frequency Response, Graph Plots of Responses.	prob023_1.bdf prob023_1.op2 prob023_2.bdf prob023_2.op2 prob023_3.bdf prob023_3.op2
Problem 24: Modal Frequency Response, Enforced Base Motion with Modal Damping and Rigid Body Elements	MSC.Nastran, Solution 103 & 111, Modal Frequency Response, Fringe Plots of Mode Shapes, Graph Plots of Responses.	prob024_1.bdf prob024_1.op2 prob024_2.bdf prob024_2.op2
Problem 25:Modal Frequency Response, Enforced Base Motion with Modal Damping and Shell P-Elements	MSC.Nastran, Solution 111, Modal Frequency Response, Graph Plots of Responses.	prob025.bdf prob025.op2

Table 15-1. V&V Problems

Problem	Description	Features Validated
Problem 26: Complex Modes, Direct Method	MSC.Nastran, Solution 107, Complex Modes, Direct Method, Fringe and Deformation Plots of Modes.	prob026.bdf prob026.op2
Problem 27: Steady State Heat Transfer, Multiple Cavity Enclosure Radiation	MSC.Nastran, Solution 153, Steady State Heat Transfer, Fringe Plots of Temperature and Radiation Heat Flux.	prob027.bdf prob027.op2
Problem 28: Transient Heat Transfer with Phase Change	MSC.Nastran, Solution 159, Transient Thermal Analysis, Graph Plots of Temperature Responses.	prob028.bdf prob028.op2
Problem 29: Steady State Heat Transfer, 1D Conduction and Convection	MSC.Nastran, Solution 153, Steady State Heat Transfer, Fringe Plots of Temperatures.	prob029.bdf prob029.op2
Problem 30: Freebody Loads, Pinned Truss Analysis	MSC.Nastran, Solution 101, Linear Statics, Freebody Plots.	prob030.bdf prob030.op2

15.2 Validation Problems

Problem 1: Linear Statics, Rigid Frame Analysis

Solution/Element Type:

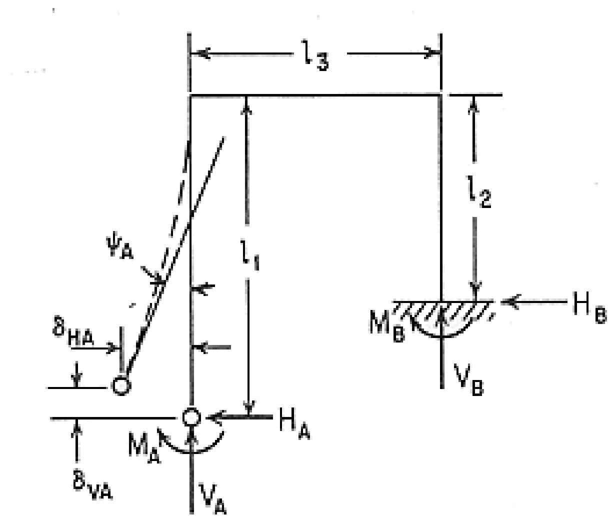
MSC.Nastran, Solution 101, Linear Statics, CBAR Bar Elements with Standard Formulation

Reference:

Roark, R.J., and Young, W.C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, pp. 122-126.

Problem Description:

A rigid frame with uniform properties is subjected to a concentrated force midspan of the top horizontal member. One end of the frame is fixed while the other is free. Find the horizontal and vertical displacements as well as the rotation at the free end of the frame.



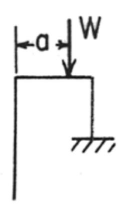
Engineering Data:

$\ell_1 = 7.0in$ $\quad E = E_1 = E_2 = E_3 = 10^7 psi$

$\ell_2 = 10.0in$ $\quad v = 0.3$

$\ell_3 = 4.0in$ $\quad I = I_1 = I_2 = I_3 = 0.0052083in^4$

$a = 2.0in$ $\quad W = 100.0lbf$



Theoretical Solution:

Upon substitution of the engineering data into the equations below, the following results are obtained:

$$\delta_{HA} = -W\left[\frac{\ell_2}{2EI}(2\ell_1 - \ell_2)(\ell_3 - a) + \frac{\ell_1}{2EI}(\ell_3 - a)^2\right] = -0.10368 in$$

$$\delta_{VA} = -W\left(\frac{\ell_2 \ell_3^2}{EI} + \frac{\ell_3^3}{3EI} - a\left(\frac{\ell_2 \ell_3}{EI} - \frac{\ell_3^2}{2EI}\right) + \frac{a^3}{6EI}\right) = -0.16640 in$$

$$\Psi_A = -W\left[\frac{\ell_1}{EI}(\ell_3 - a) + \frac{1}{2EI}(\ell_3 - a)^2\right] = -0.0422403 radians$$

MSC.Nastran Results:

To determine the deflections at the free end of the frame, the model shown in **Figure 15-1** was created. This model consisted of 14 MSC.Nastran CBAR elements. Each bar was assumed to have a square cross-section of 0.5 x 0.5 inches. This gives a cross-sectional moment of inertia of 0.0052083 in^4 that is identical to what was assumed in the preceding calculations. Using this model, the following results were obtained.

Table 15-2 Rigid Frame Analysis Results

Source	δ_{HA}	δ_{VA}	Ψ_A
MSC.Nastran*	0.10368	-0.16706	0.042243
Theory	-0.10368	-0.16664	-0.042240
%, Difference	0.0%	0.252%	0.007%

*MSC.Nastran results have opposite sign due to the reversal of the direction of x and z axes of the global coordinate frame relative to the theoretical results coordinates.

The corresponding deformed shape plot that was made for the rigid frame is shown in **Figure 15-2** where the deformed shape has been superimposed upon the undeformed mesh. Fringe plots for the x, y, and z components of the translational displacements that were generated with MSC.Patran are shown in **Figure 15-3**, **Figure 15-4** and **Figure 15-5**, respectively. The fringe plot for the rotation about the z axis is shown plotted in **Figure 15-6**. All fringe plots are displayed on the fully deformed structure. Examination of these fringe plots near the free end of the frame clearly shows that the MSC.Patran results are identical to the preceding MSC.Nastran results.

File(s): <install_dir>/results_vv_files/prob001.bdf, prob001.op2

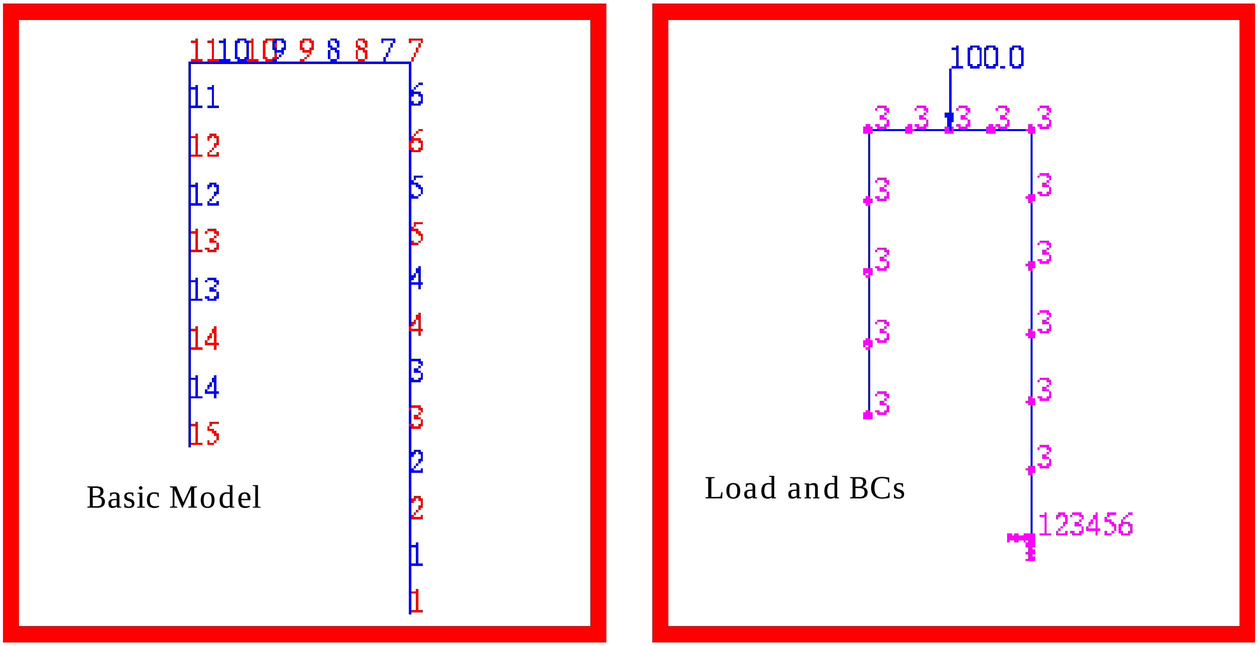


Figure 15-1 Basic FE Model of Rigid Frame with Load and BCs.

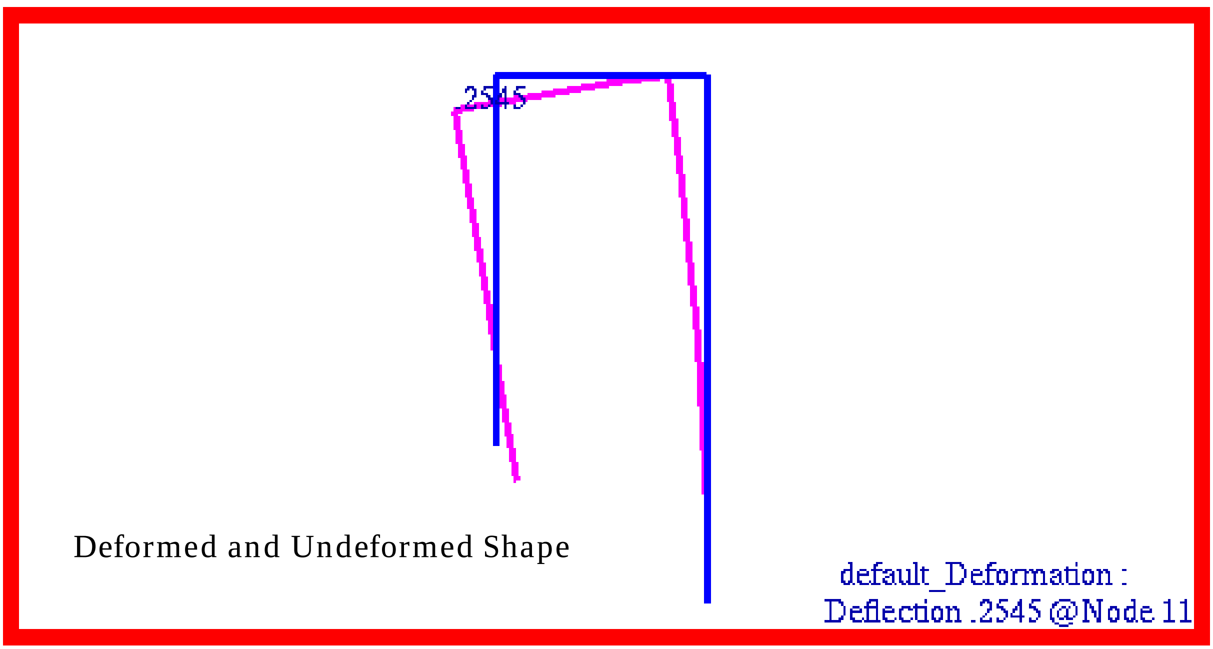


Figure 15-2 Deformed and Undeformed Shape of Rigid Frame.

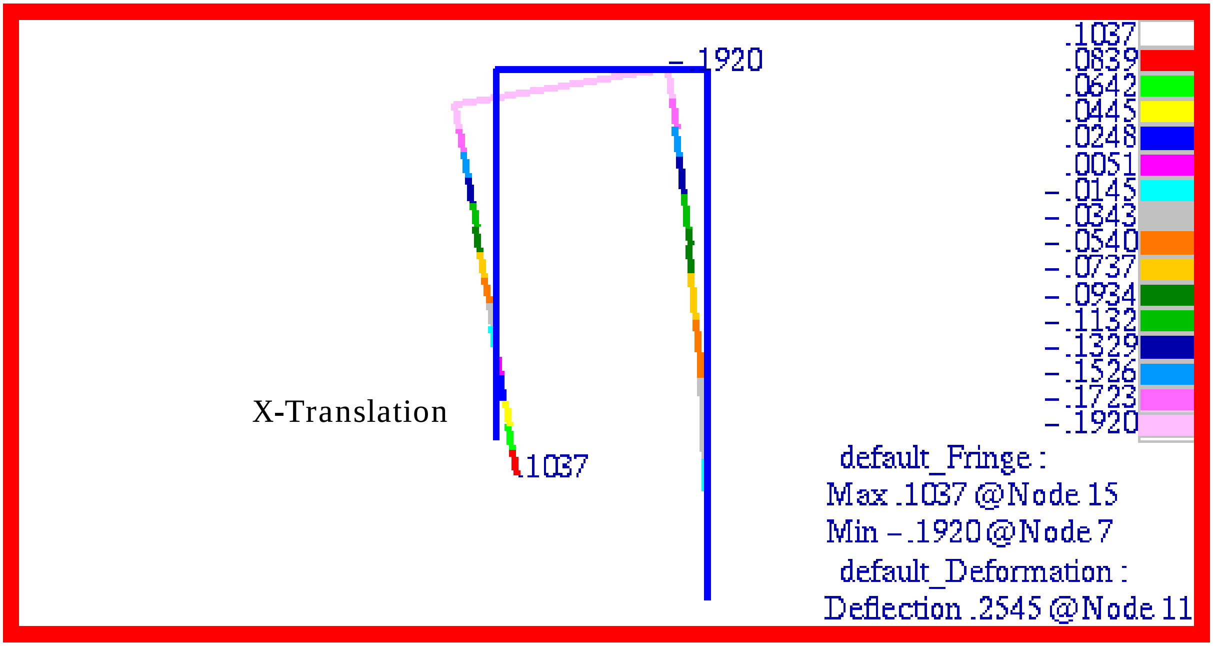


Figure 15-3 X-Translational Deformation of Rigid Frame.

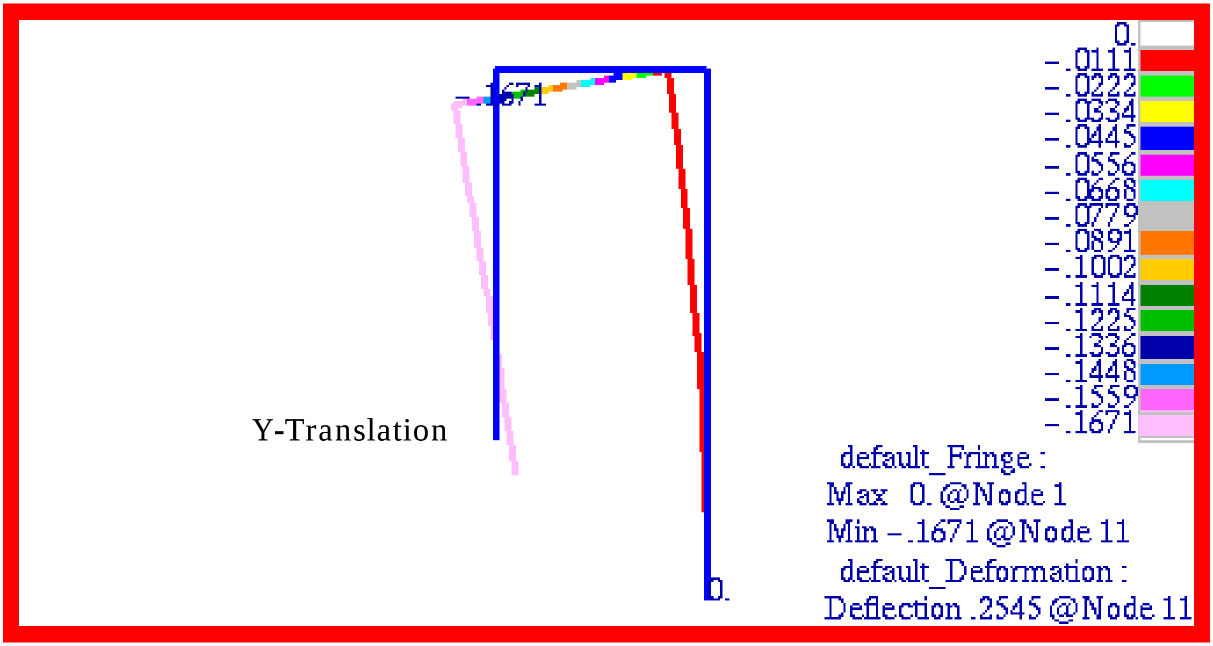


Figure 15-4 Y-Translational Deformation of Rigid Frame.

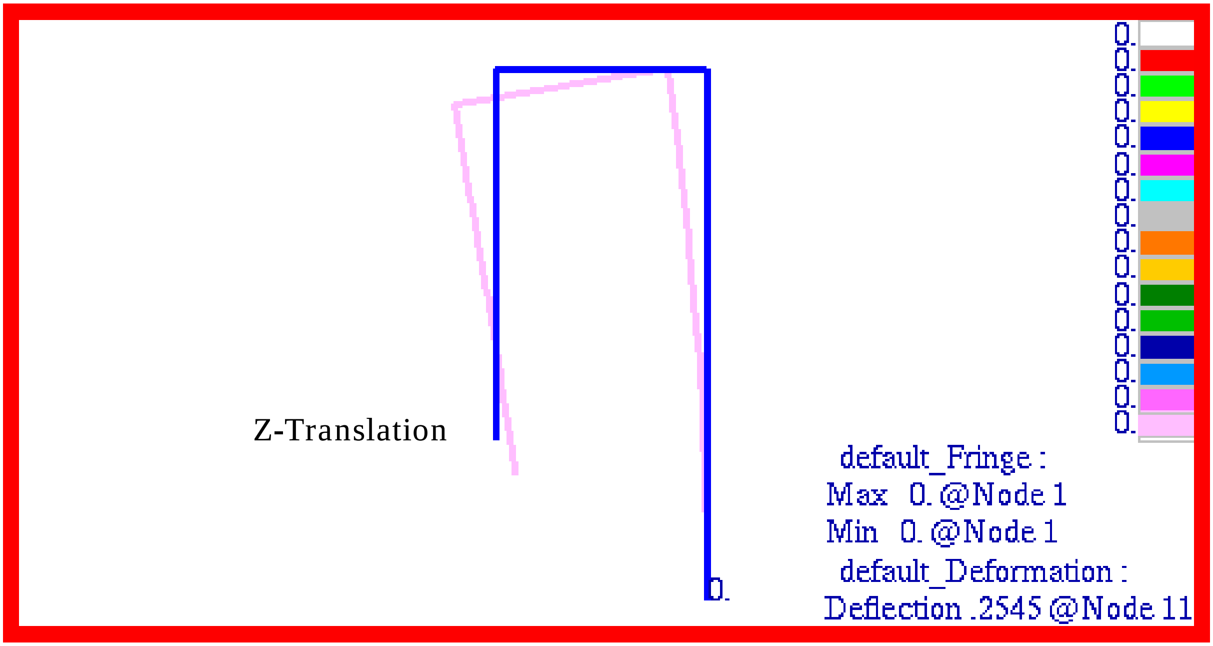


Figure 15-5 Z-Translational Deformation of Rigid Frame.

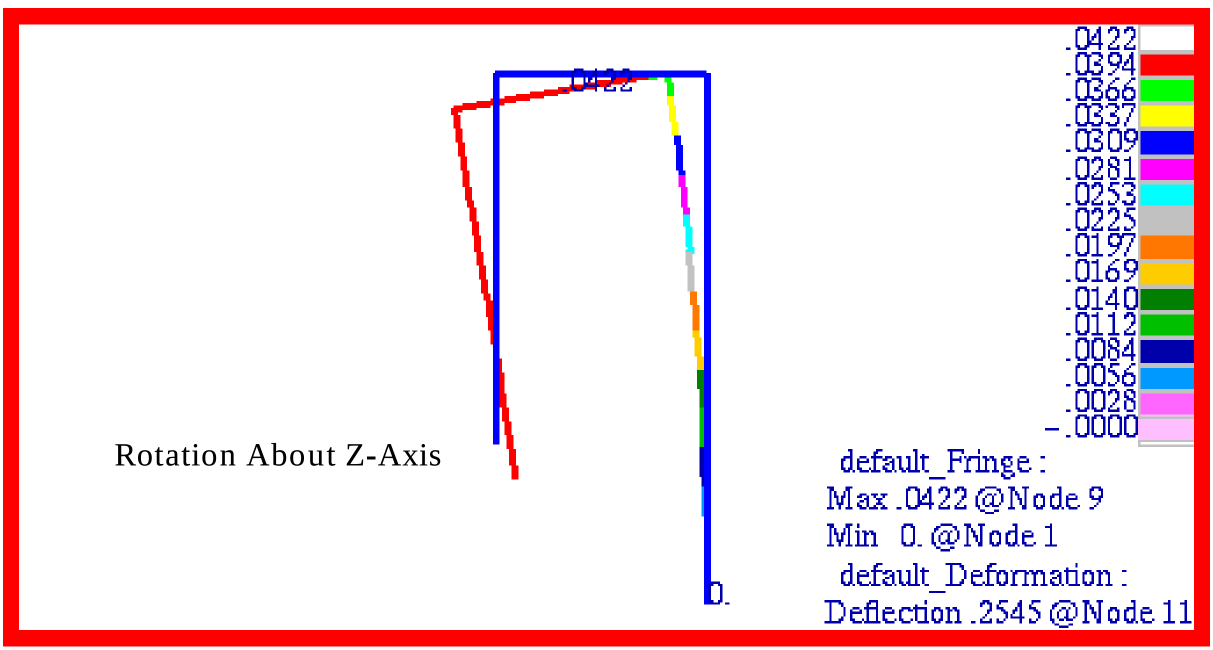


Figure 15-6 Rotational Displacement about Z-axis of Rigid Frame.

Problem 2: Linear Statics, Cross-Ply Composite Plate Analysis

Solution/Element Type:

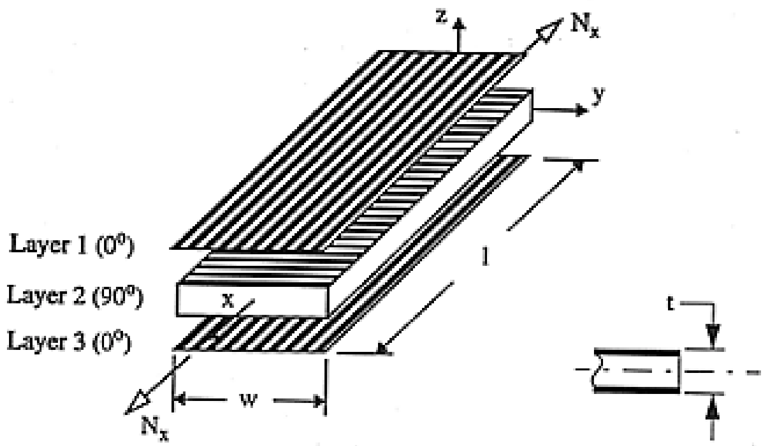
MSC.Nastran, Linear Statics, Solution 101, CQUAD4 Elements with Standard Formulation and Laminated Material Properties.

Reference:

Jones, R. M., *Mechanics of Composite Materials*, Hemisphere Pub. Corp., 1975, pp. 198-201.

Problem Description:

A flat rectangular plate is made from a cross-ply 0/90/0 degree ply stacking. Assuming that the plate is cured at 270^o F, find the stress in each of the plies once the plate is cooled down to 70^o F.



Engineering Data:

$length = l = 3.0 in$

$width = w = 2.0 in$

$laminate\ thickness = t = 0.55 in$

$outer\ ply\ thickness = t_o = 0.0458 in$

$inner\ ply\ thickness = t_i = 0.4583 in$

$N_x = 200 lb_f / in$

$E_1 = 7.8 \times 10^6 psi$

$E_2 = 2.6 \times 10^6 psi$

$G_{12} = 1.25 \times 10^6 psi$

$\nu_{12} = 0.25$

$\alpha_1 = 3.5 \times 10^{-6} / (°F)$

$\alpha_2 = 11.4 \times 10^{-6} / (°F)$

Theoretical Solution:

Only the stresses in plies 1 and 2 are shown since the laminate is symmetric about its midplane. Hence, ply 1 is identical to ply 3. In the expressions below, ΔT refers to the temperature difference between the curing temperature for the laminate and its final operational temperature. In this particular example, ΔT would equal -200^o F. Because of the absence of any localized restraint to thermal shrinkage coupled with isotropic thermal expansions, the stresses in each ply are uniform. The predicted stresses are as follows for $N_x = 200 lbf / in$, $\Delta T = -200°F$ and $t = 0.55$ in:

$$\sigma_x^{(1)} = 2.27\left(\frac{N_x}{t}\right) + 35.5\Delta T\ \frac{\text{psi}}{°F} = -6274.55psi$$

$$\sigma_y^{(1)} = 0.12\left(\frac{N_x}{t}\right) - 16.0\Delta T\ \frac{\text{psi}}{°F} = 3243.64psi$$

$$\sigma_x^{(2)} = 0.75\left(\frac{N_x}{t}\right) - 7.1\Delta T\ \frac{\text{psi}}{°F} = 1692.73psi$$

$$\sigma_y^{(2)} = -0.024\left(\frac{N_x}{t}\right) + 3.2\Delta T\ \frac{\text{psi}}{°F} = -648.73psi$$

$$\tau_{xy}^{(1)} = \tau_{xy}^{(2)} = 0$$

MSC.Nastran Results:

To predict the stresses in the cross-ply laminate for this problem, a model was created that consisted of CQUAD4 elements with laminate properties specified by a MSC.Nastran PCOMP data entry. This permitted stress recovery to be performed on a ply by ply basis. Due to symmetry, any motion along the vertical and horizontal planes of symmetry was restrained out. In addition, all out of plane deformations were restrained since a balanced cross-ply laminate should remain flat during cool down. Application of these boundary conditions avoided any potential problems with unrestrained rigid body motion. The applied loads and boundary conditions are also shown superimposed upon the finite element mesh in **Figure 15-7**.

Using this model, the following results were obtained with MSC.Nastran.

Table 15-3 Ply Stress, σ_x

Source	$\sigma_x^{(1)}$	$\sigma_x^{(2)}$	$\sigma_x^{(3)}$
MSC.Nastran	-6280.51	1691.68	-6280.51
Theory	-6274.55	1692.73	6274.55
%, Difference	0.095%	-0.062%	0.095%

Table 15-4 Ply Stress, σ_y

Source	$\sigma_y^{(1)}$	$\sigma_y^{(2)}$	$\sigma_y^{(3)}$
MSC.Nastran	3228.55	-645.29	3228.55
Theory	3243.64	-648.73	3243.64
%, Difference	-0.465%	-0.530%	-0.465%

Table 15-5 Shear Stresses, $\tau_{xy,\ max}$

Source	$\tau_{xy}^{(1)}$	$\tau_{xy}^{(2)}$	$\tau_{xy}^{(3)}$
MSC.Nastran	-0.002043	0.000551	-0.002043
Theory	0.0	0.0	0.0
%, Difference	-	-	-

The corresponding fringe plots that were generated with MSC.Patran for the x-, y- and xy shear stress components in each of the plies are shown in **Figure 15-8** through **Figure 15-16**. A comparison of these plots with the preceding MSC.Nastran results clearly shows that the two are identical. Layer 1 and 3 were plotted in the element coordinate system; layer 2 was plotted in the Global coordinate system. You must read in the .bdf file below to be able to perform the correct coordinate transformation to reproduce the layer 2 plots.

Files:`<install_dir>/results_vv_files/prob002.bdf, prob002.op2`

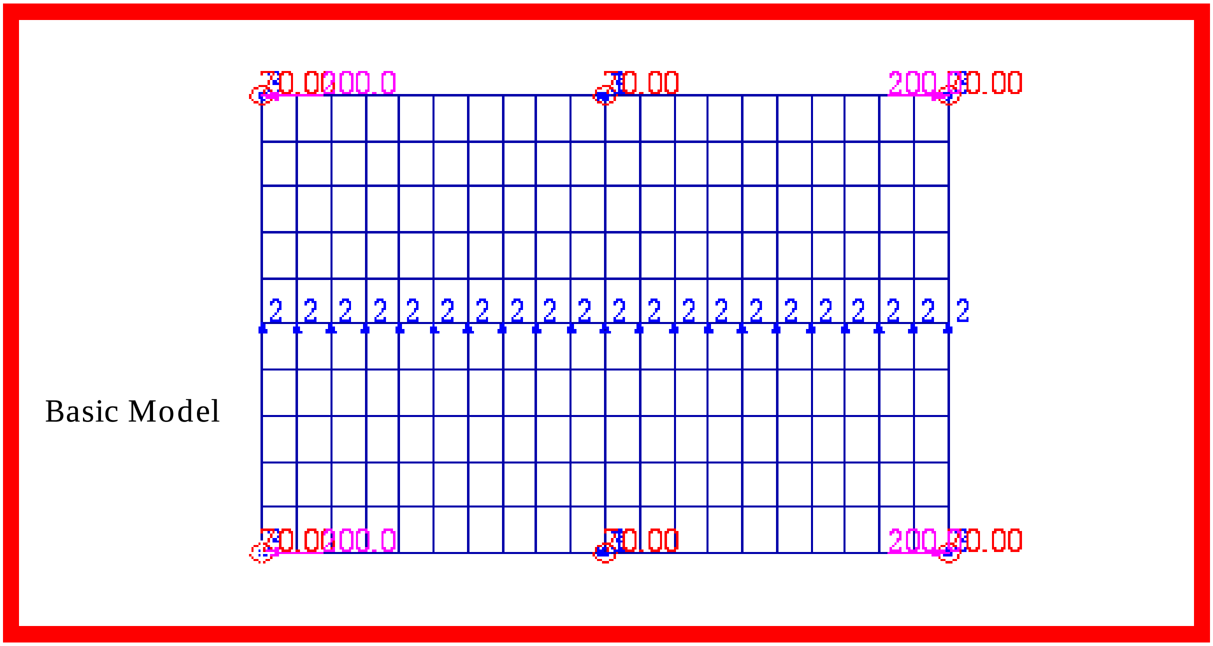


Figure 15-7 Basic Cross-Ply FE Composite Plate Model.

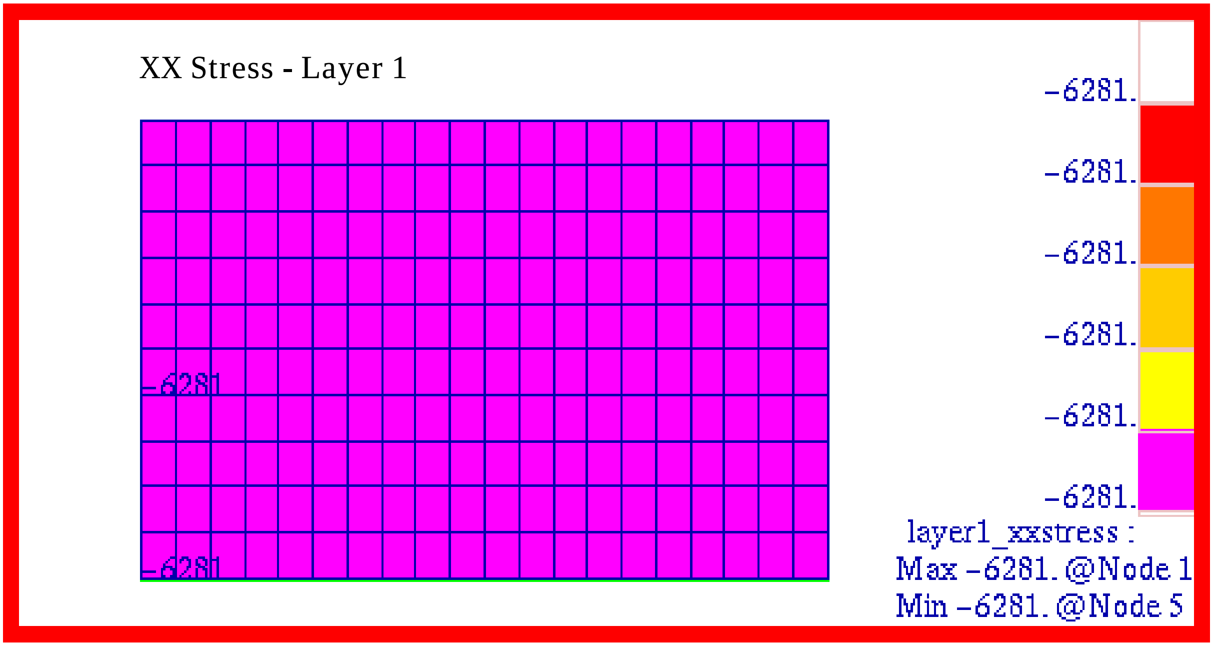


Figure 15-8 Cross-Ply Composite Plate, σ_x Stress, Layer 1.

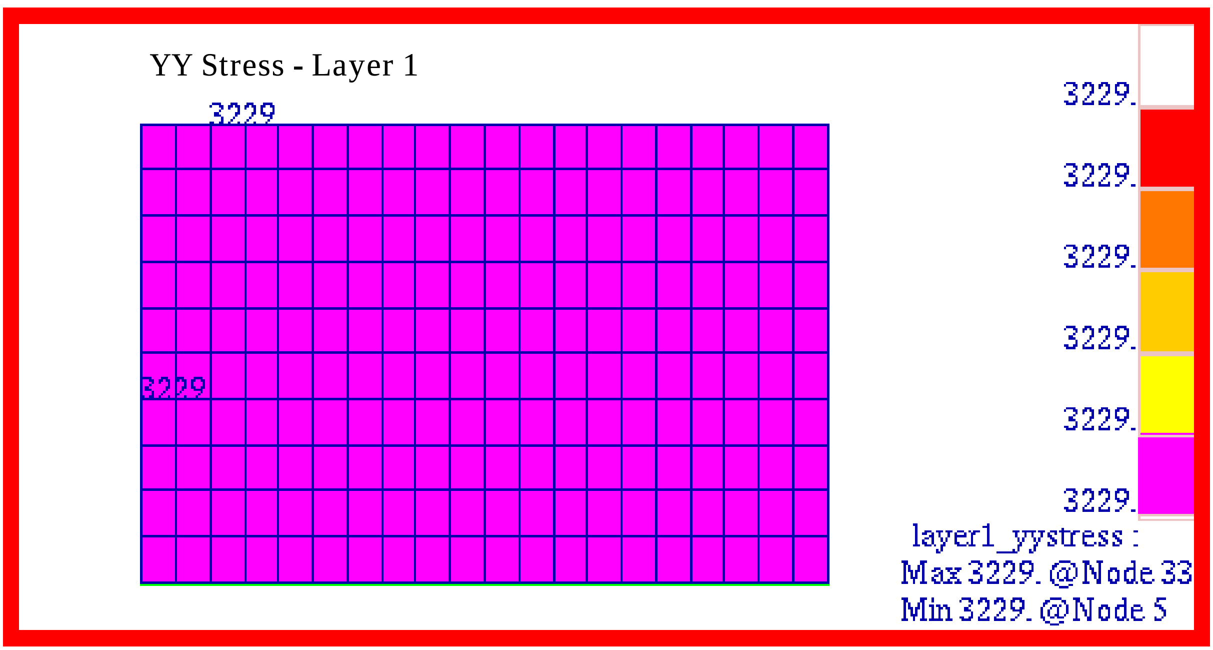


Figure 15-9 Cross-Ply Composite Plate, σ_y Stress, Layer 1.

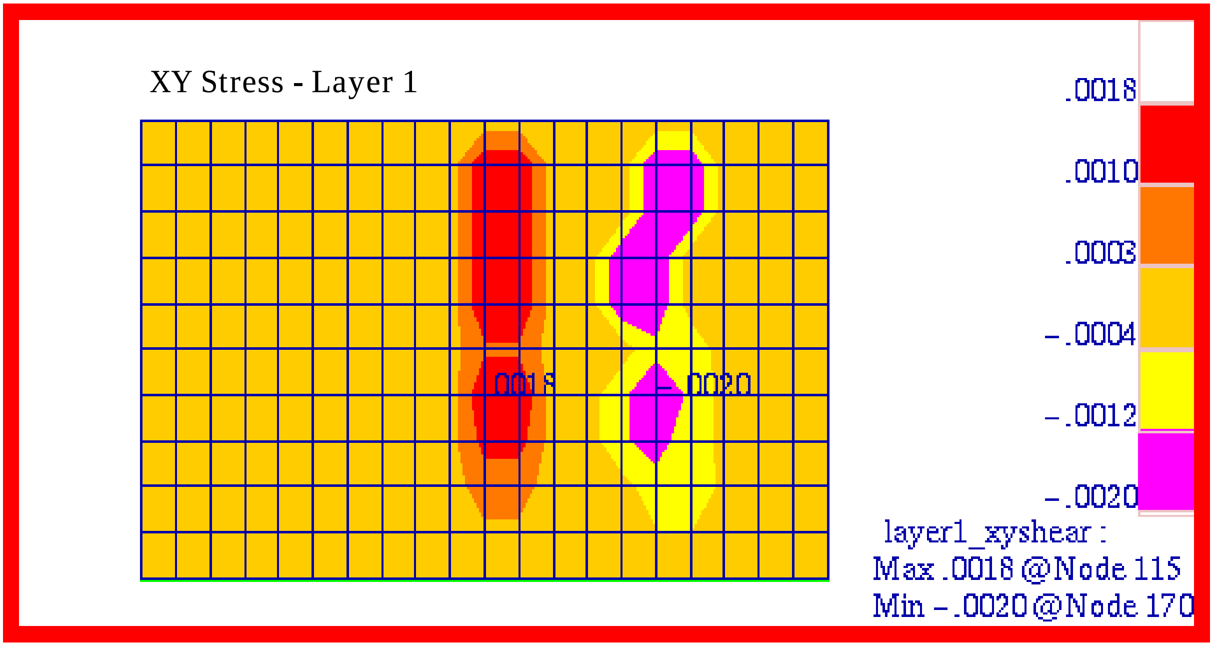


Figure 15-10 Cross-Ply Composite Plate, τ_{xy} Stress, Layer 1.

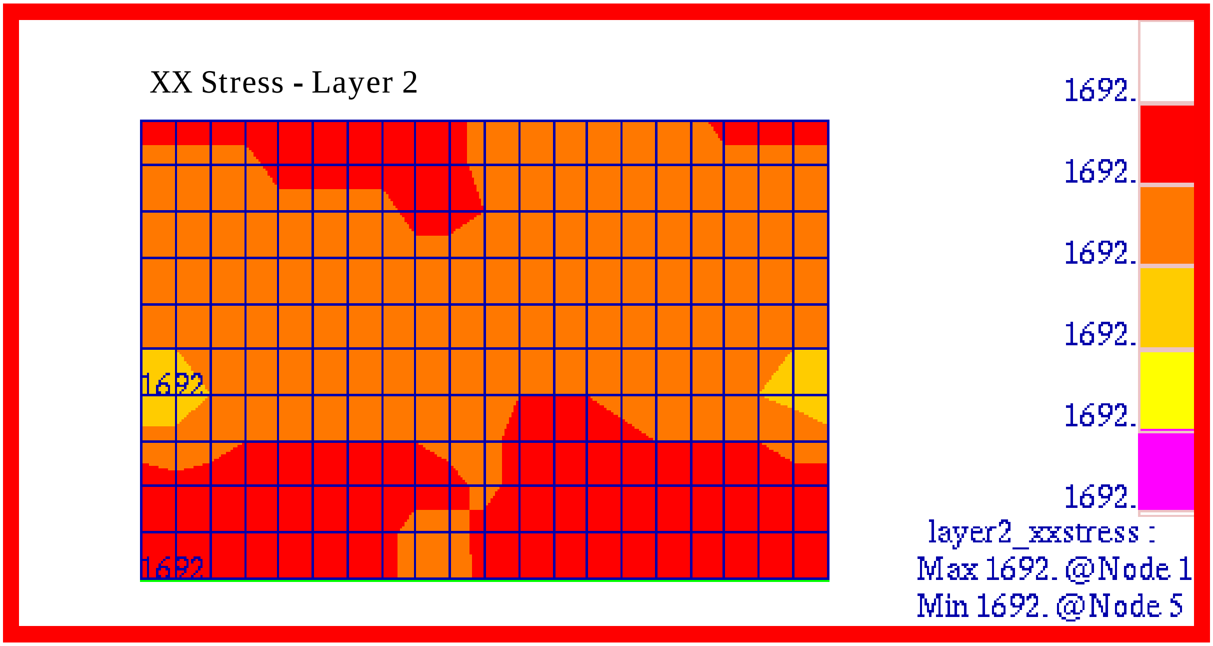


Figure 15-11 Cross-Ply Composite Plate, σ_x Stress, Layer 2.

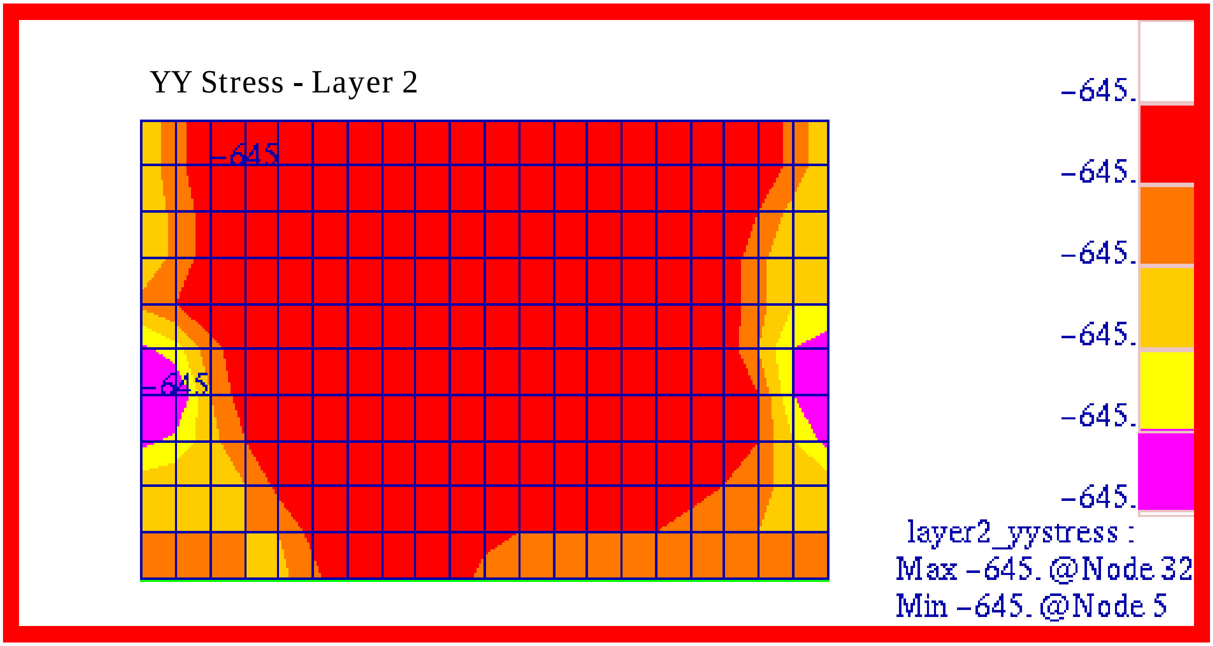


Figure 15-12 Cross-Ply Composite Plate, σ_y Stress, Layer 2.

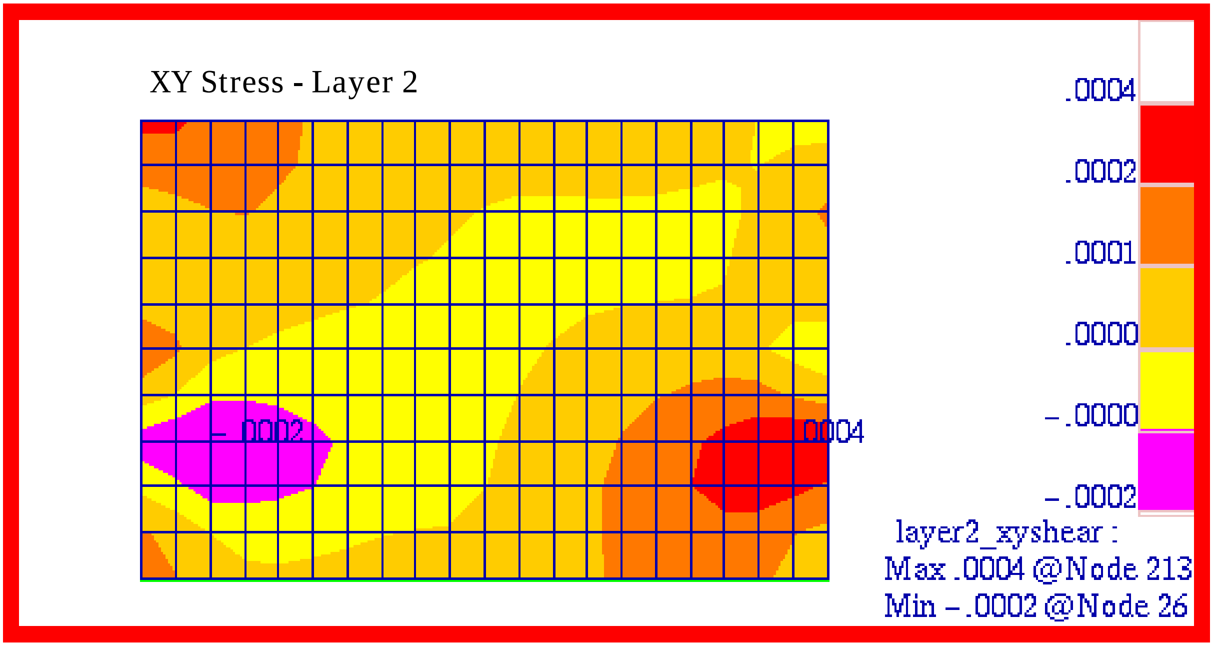


Figure 15-13 Cross-Ply Composite Plate, τ_{xy} Stress, Layer 2.

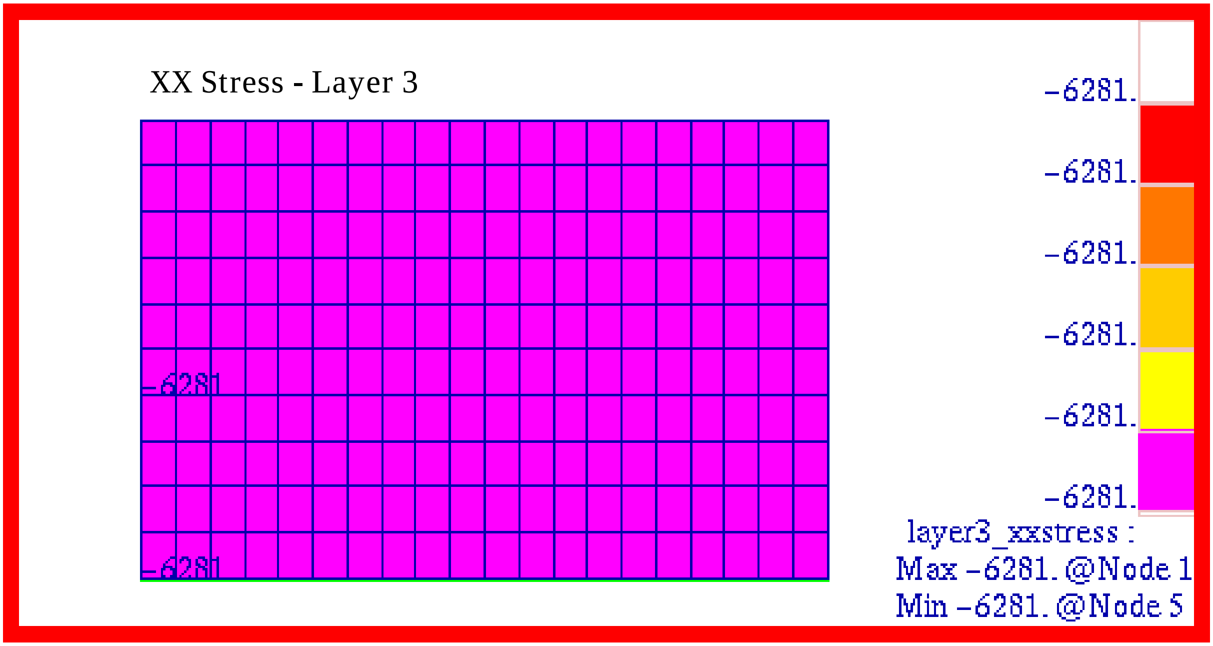


Figure 15-14 Cross-Ply Composite Plate, σ_x Stress, Layer 3.

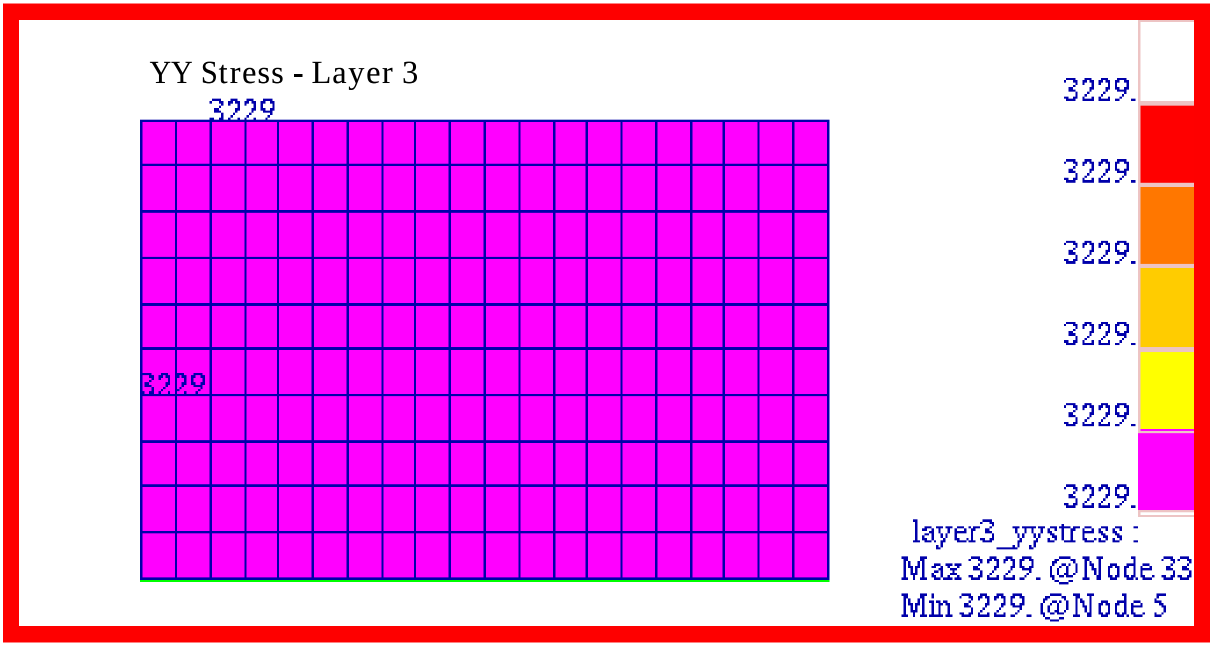


Figure 15-15 Cross-Ply Composite Plate, σ_y Stress, Layer 3.

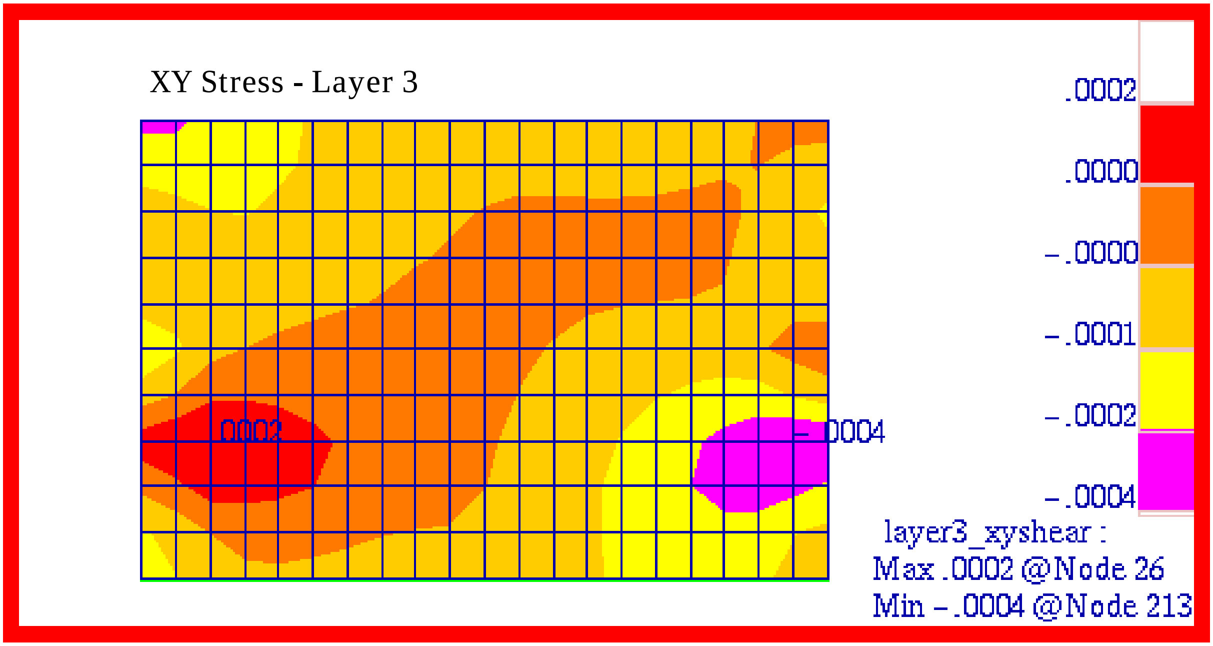


Figure 15-16 Cross-Ply Composite Plate, τ_{xy} Stress, Layer3.

Problem 3: Linear Statics, Principal Stress and Stress Transformation

Solution/Element Type:

MSC.Nastran, Solution 101, Linear Statics, CQUAD4 Elements with Standard Formulation.

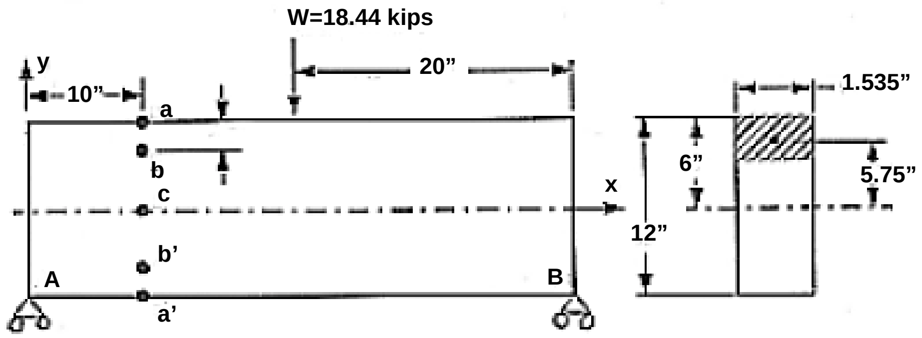
Reference:

Popov, E.P., *Introduction to Mechanics of Solids*, Prentice-Hall, Inc., 1968, pp. 337-340.

Roark, R.J., and Young, W.C. , *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, pp. 62, 93 - 96, 101.

Problem Description:

A weightless rectangular beam spans 40 inches and is loaded with a vertical downward force W = 18.44 kips at midspan. Find the principal stresses at points a, b, c, b' and a'.



Engineering Data:

$Length = l = 40in$

$Height = h = 12in$

$Depth = d = 1.535in$

$v = 0.318$

$I_1 = dh^3/12 = 221.04in^4$

$I_2 = hd^3/12 = 3.617in^4$

$E = 3 \times 10^7 psi$

Theoretical Solution:

Reaction Loads:

$$R_A = \frac{W}{l}(x-a) = \frac{18.44}{40}(40-20) = 9.22kips$$

$$R_B = \frac{Wa}{l} = \frac{18.44(20)}{40} = 9.22kips$$

Transverse Shear Force:

$$V = R_A - W\langle x-a\rangle^0 = R_A = 9.22kips$$

$$\text{where } \langle x-a\rangle^0 = \begin{cases} 0 & x < a \\ 1 & x > a \\ undefined & \text{x=a} \end{cases}$$

Moments:

$$M_A = M_B = 0$$

$$M(x) = R_A x - W(x-a)\langle x-a\rangle^0$$

$$M(10) = R_A(10) = 92.2Kips - in$$

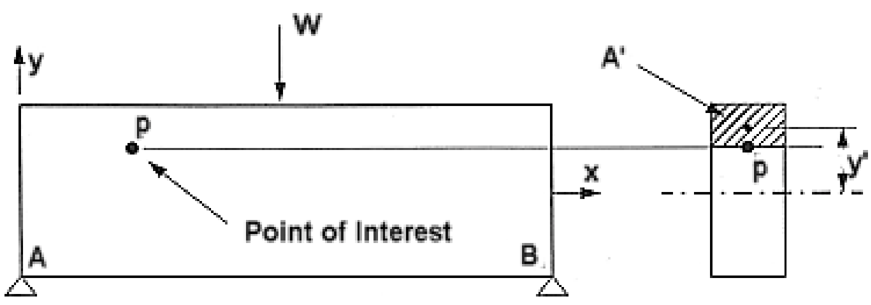
$$where\langle x-a\rangle^0 = 0 \text{ since } (x = 10) < (a = 20)$$

Fiber and Shear Stresses:

$$\sigma(x, y) = \frac{M(y)}{I} = \frac{[R_A - W(x-a)\langle x-a\rangle^0]y}{I}$$

$$\tau_{xy} = \frac{VA'y'}{Id}$$

Here A' is the area of that part of the section above (or below if the point of interest is located below the beam's neutral axis) the point of interest and y' is the distance from the neutral axis to the centroid of A' as shown below.



Stress at a:

$$\sigma(10, 6) = \frac{-R_A xy}{I} = \frac{-(9.22)(10)(6)}{221.04} = -2.50ksi$$

$$\tau_{xy}(10, 6) = 0.0$$

Stress at b:

$$\sigma(10,\,5.5) \;=\; \frac{-R_A xy}{I} \;=\; \frac{-(9.22)(10)(5.5)}{221.04} \;=\; -2.29ksi$$

$$\tau_{xy}(10,\,5.5) \;=\; -\frac{VA'y'}{Id} \;=\; -\frac{(9.22)(10)(0.5)(5.75)}{(221.04)(1.535)} \;=\; -0.12ksi$$

Stress at c:

$$\sigma(10,\,0) \;=\; 0.0$$

$$\tau_{xy} \;=\; -\frac{VA'y'}{Id} \;=\; \frac{(9.22)(1.535)(6)(3)}{(221.04)(1.535)} \;=\; -0.75ksi$$

Stress at b':

$$\sigma(10,\,-5.5) \;=\; \frac{-R_A xy}{I} \;=\; \frac{-(9.22)(10)(-5.5)}{221.04} \;=\; 2.29ksi$$

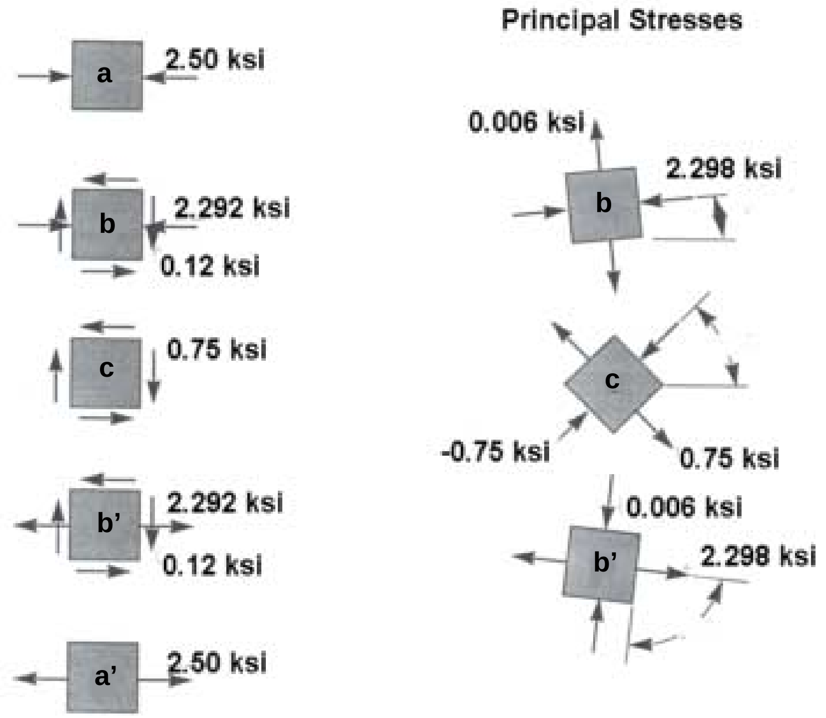
$$\tau_{xy}(10,\,-5.5) \;=\; \frac{VA'y'}{Id} \;=\; \frac{(9.22)(1.535)(0.5)(-5.75)}{(221.04)(1.535)} \;=\; -0.12ksi$$

Stress at a':

$$\sigma(10,\,-6) \;=\; \frac{-R_A xy}{I} \;=\; \frac{-(9.22)(10)(-6)}{221.04} \;=\; 2.50ksi$$

$$\tau_{xy} \;=\; 0.0$$

Based upon these fiber and shear stresses, the following principal stresses are calculated:



MSC.Nastran Results:

To determine the stress state in the beam, a MSC.Nastran model was generated using MSC.Patran. To maximize accuracy, a mesh density was chosen so that nodes would be precisely situated at every designated stress recovery point. In addition, to prevent any out of plane deformation, all displacements normal to the plane of the model were fully restrained, thereby imposing a plane strain condition. Furthermore, to prevent any longitudinal rigid body translation, a symmetry boundary condition was imposed at midspan the entire height of the beam. The imposed boundary conditions and applied loads are shown in **Figure 15-17** superimposed upon the mesh, which consisted solely of CQUAD4 elements using a standard formulation. Using this model, the following stresses were predicted at points **a**, **b**, **c**, **b'** and **a'**.

Table 15-6 Stresses* at Position a

Source	σ_x	τ_{xy}	Max Principal	Min Principal
MSC.Nastran	-2354.	-64.04	-29. 75	-2356.
Theory	-2500.	0.0	0.0	-2500.
%, Difference	-5.85%	-	-	-5.76%

Table 15-7 Stresses* at Position b

Source	σ_x	τ_{xy}	Max Principal	Min Principal
MSC.Nastran	-2151.	-124.4	12.65	-2158.
Theory	-2290.	-120.0	6.00	-2298.
%, Difference	-6.07%	3.67%	110.8%	-6.09%

Table 15-8 Stresses* at Position c

Source	σ_x	τ_{xy}	Max Principal	Min Principal
MSC.Nastran	-122.9	-760.1	756.5	-779.9
Theory	0.0	-750.0	750.0	-750.0
%, Difference	-	1.35%	0.867%	3.99%

Table 15-9 Stresses* at Position b'

Source	σ_x	τ_{xy}	Max Principal	Min Principal
MSC.Nastran	2437.	-106.5	2442.	1.557
Theory	2290.	-120.0	2298.	6.00
%, Difference	6.42%	-11.3%	6.27%	-74. 05%

Table 15-10 Stresses* at Position a'

Source	σ_x	τ_{xy}	Max Principal	Min Principal
MSC.Nastran	2671.	-54.92	2672.	40.86
Theory	2500.	0.0	2500.	0.0
%, Difference	6.64%	-	6.88%	-

* Based upon nodal average of adjacent elements.

The MSC.Patran fringe plots that were made of the x-, y- and xy- stress components are shown in Figure 15-18, Figure 15-19, and Figure 15-20 respectively. In addition, the orientation angle for the max principal stress, or zero shear angle, has been plotted in Figure 15-21. To better show the stress state at each of the designated stress recovery points, tensor plots were made of the fiber stress as well as the principal stress at each position, starting with **a** and proceeding onto **a'**. These are shown in Figure 15-22 through Figure 15-26. Examination of the MSC.Patran tensor and fringe plots reveals that they are identical to the preceding MSC.Nastran results.

File(s):<install_dir>/results_vv_files/prob003.bdf, prob003.op2

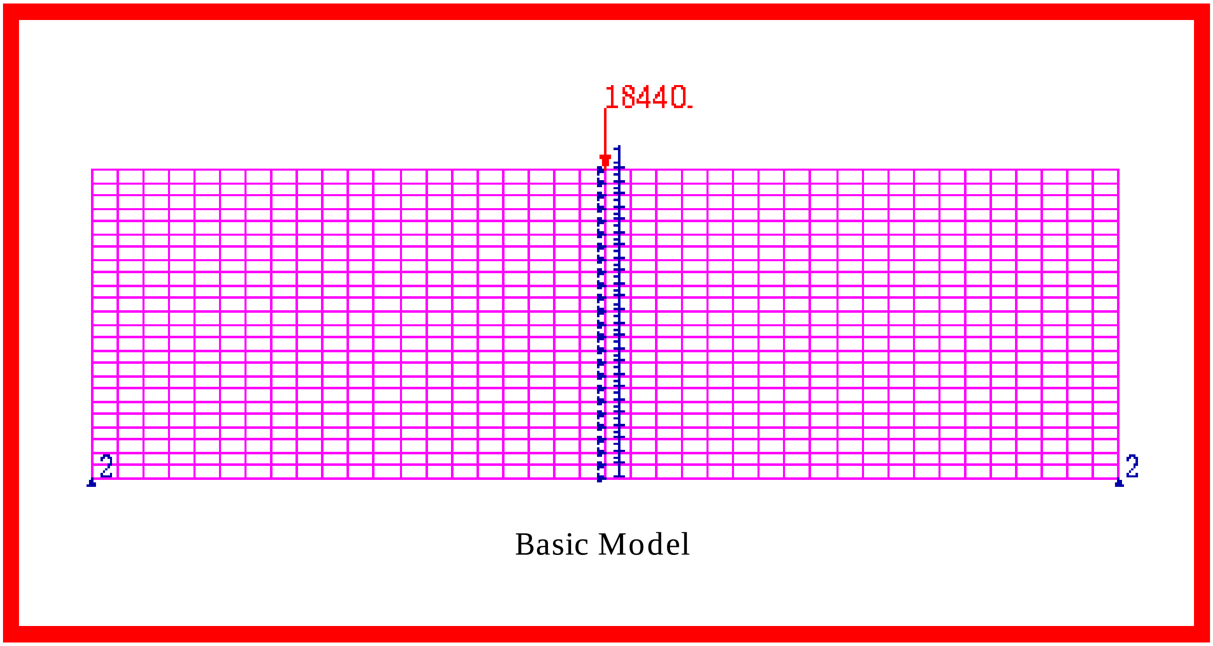


Figure 15-17 Basic Beam Model with Loads and Boundary Conditions.

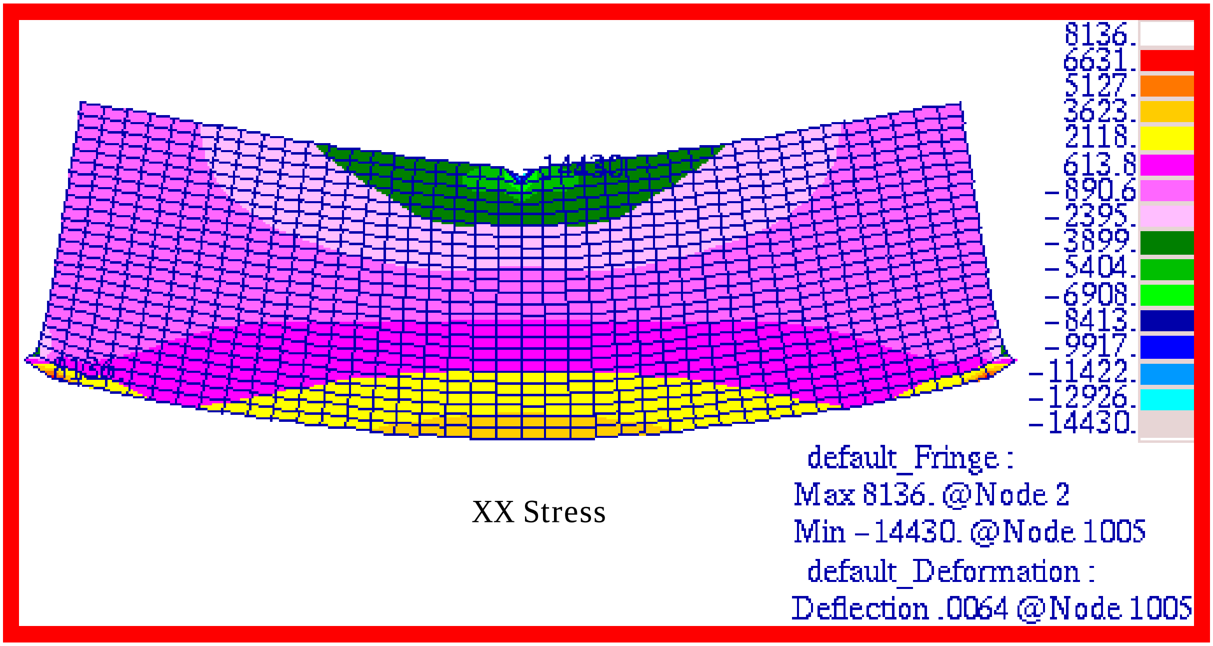


Figure 15-18 σ_x **Stress of Beam Model.**

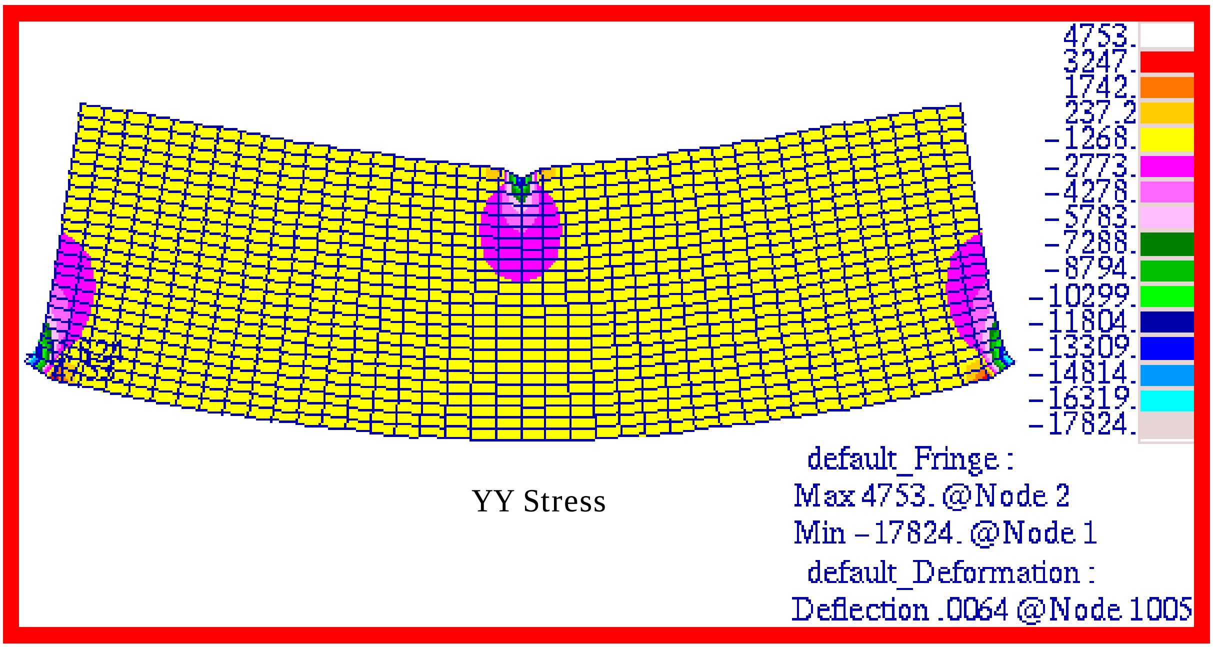


Figure 15-19 σ_y **Stress of Beam Model.**

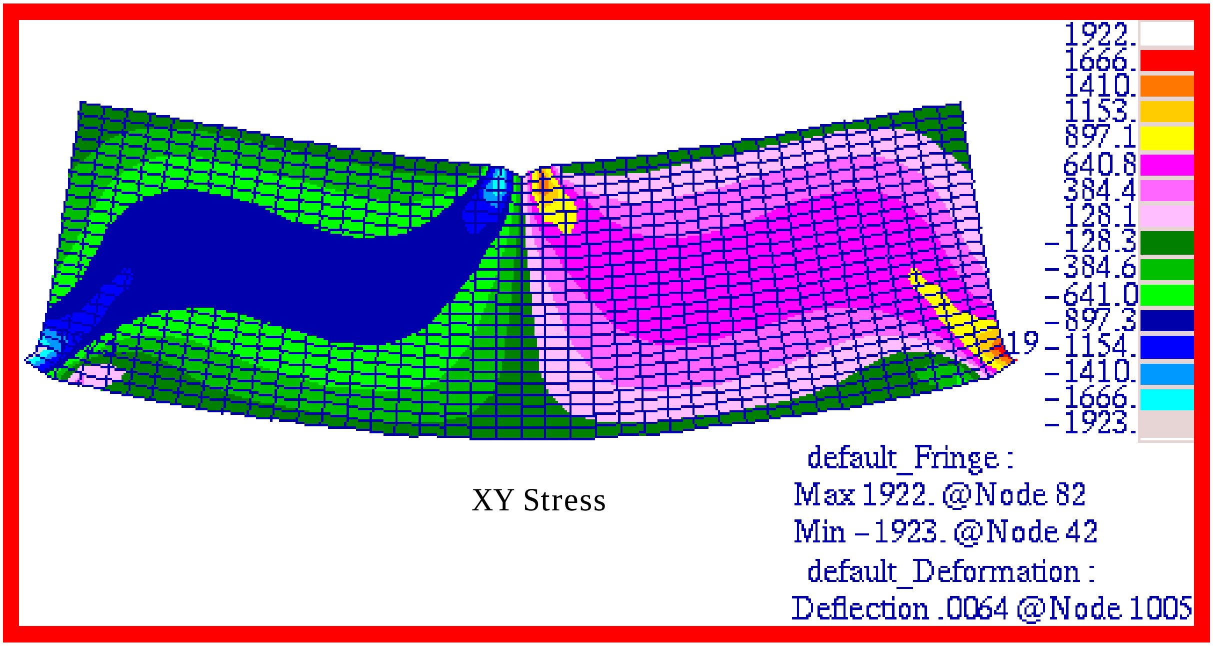


Figure 15-20 τ_{xy} **Stress of Beam Model.**

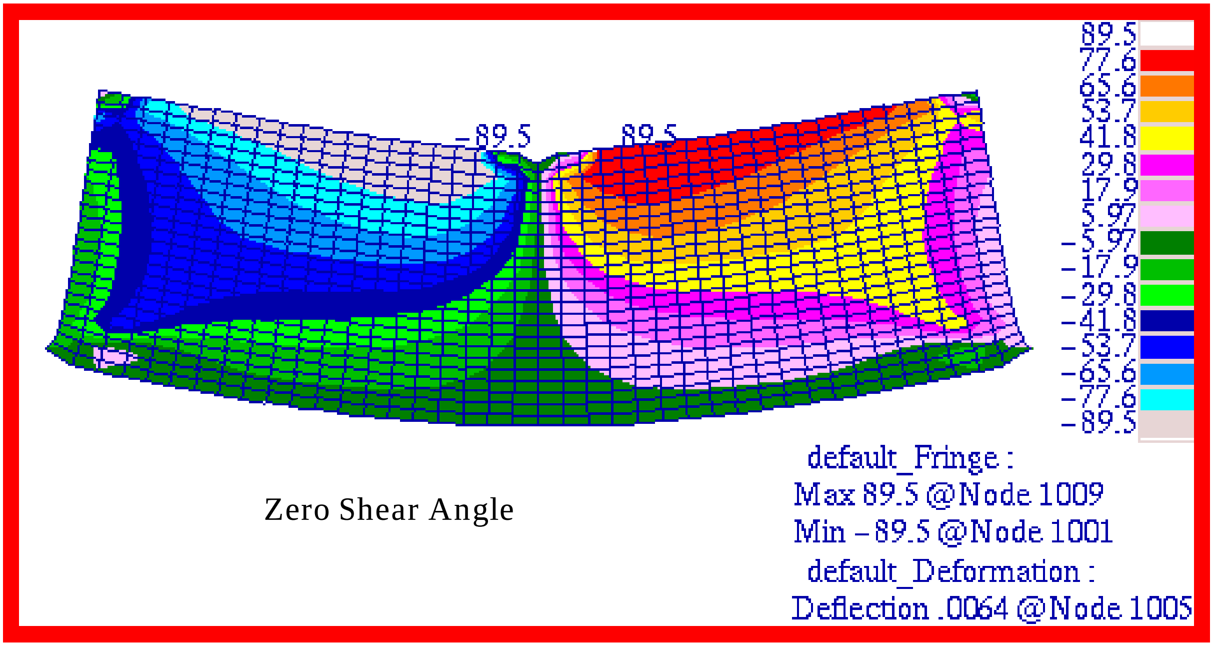


Figure 15-21 Orientation Angle of Maximum Principal Stress.

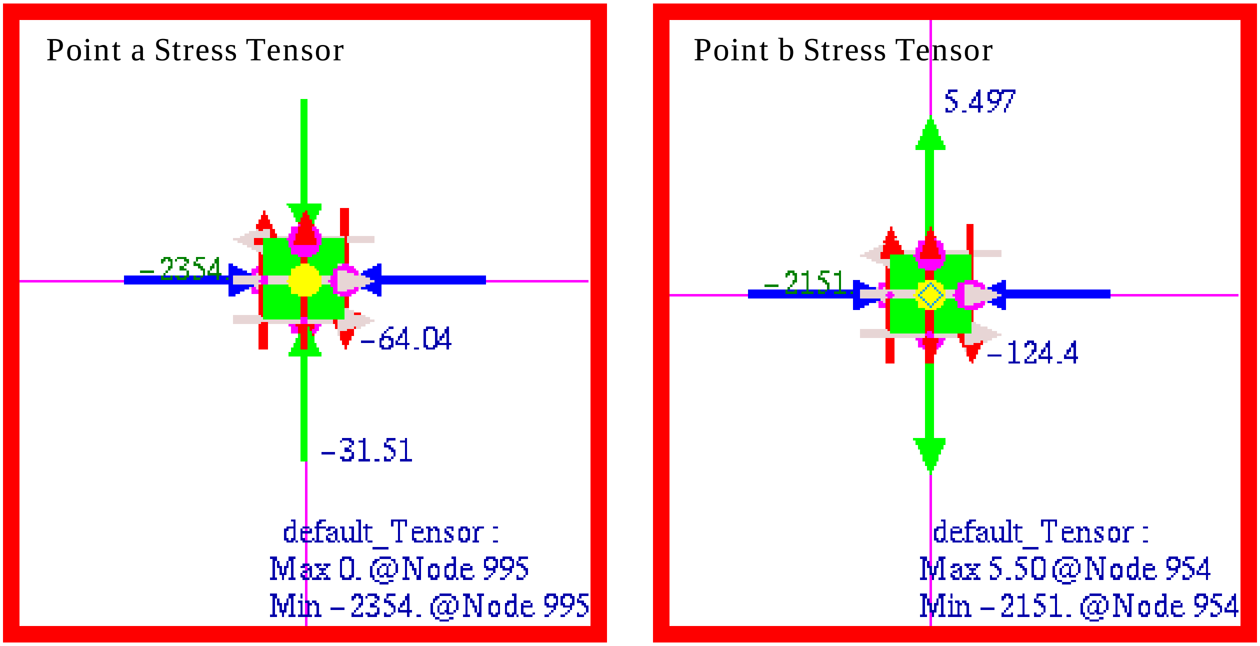


Figure 15-22 Stress Tensor at Points a and b.

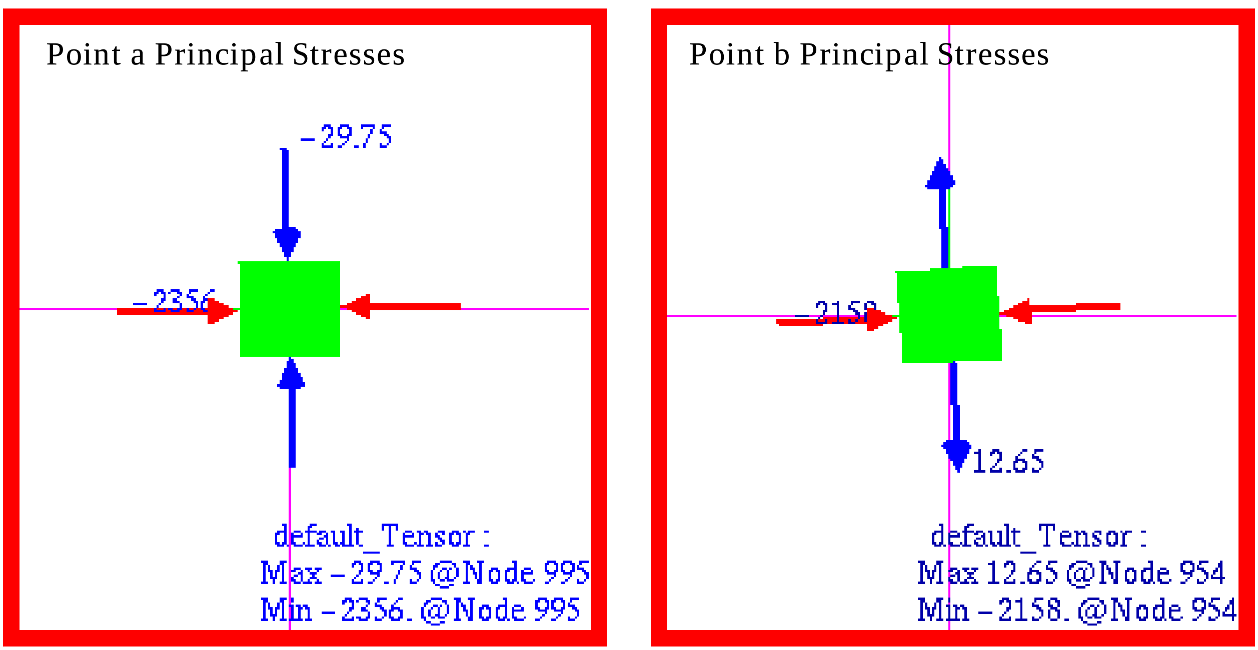


Figure 15-23 Principal Stresses at Points a and b.

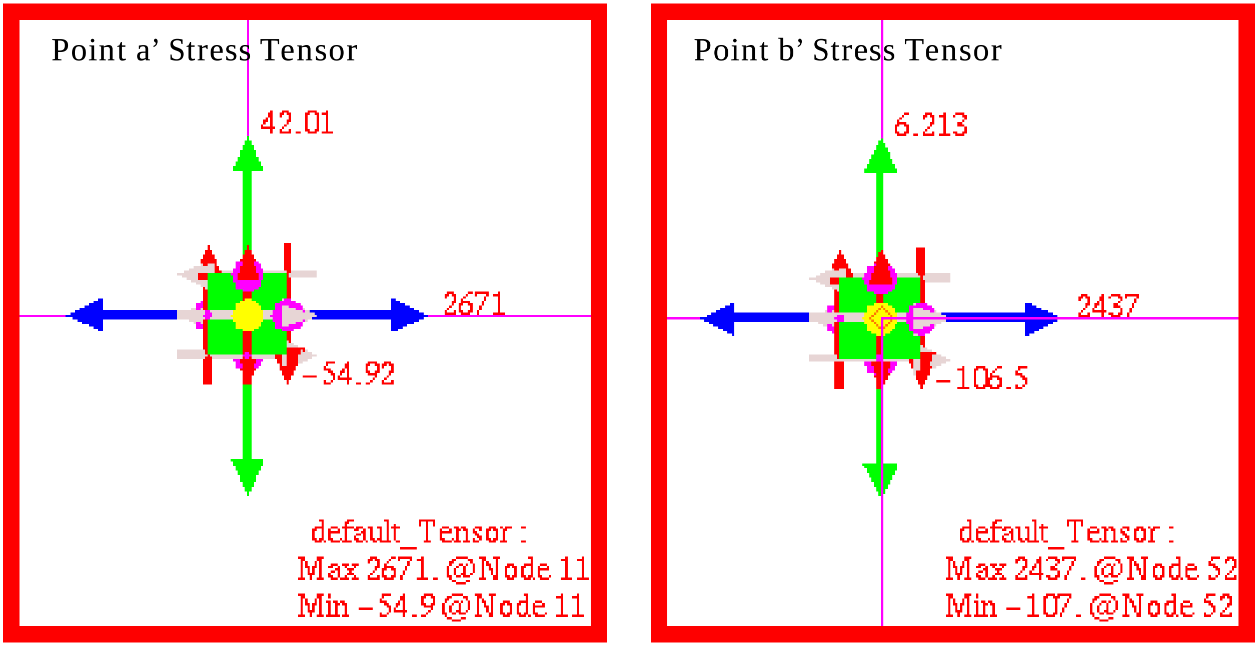


Figure 15-24 Stress Tensor at Points a’ and b’.

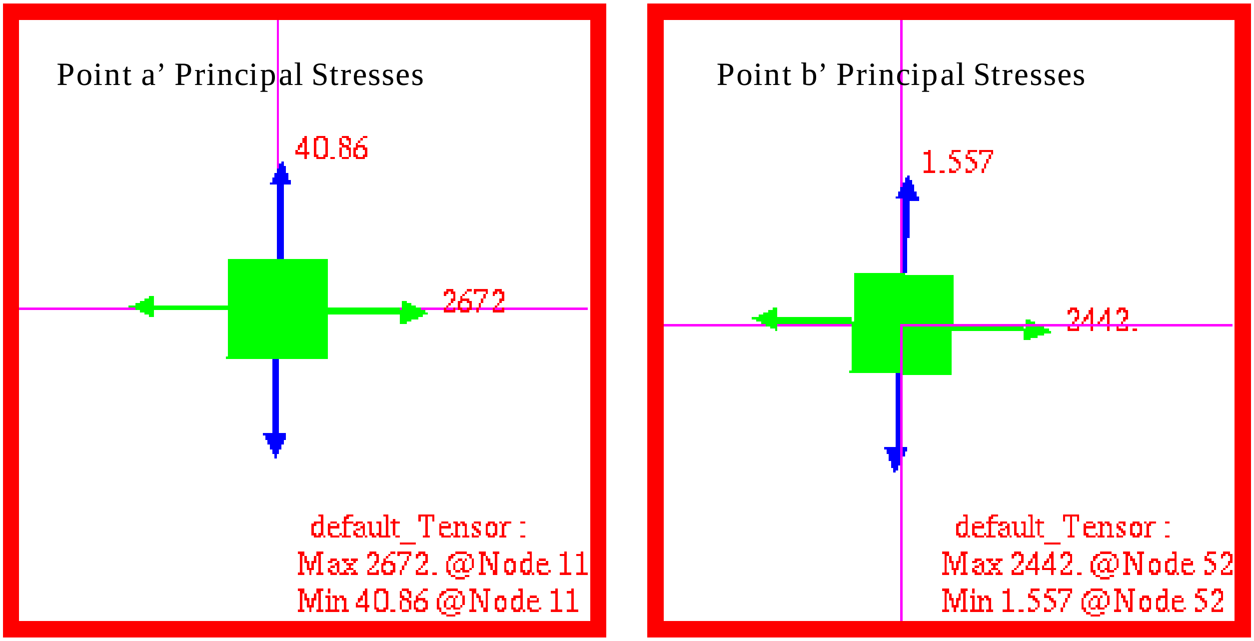


Figure 15-25 Principal Stresses at Points a’ and b’.

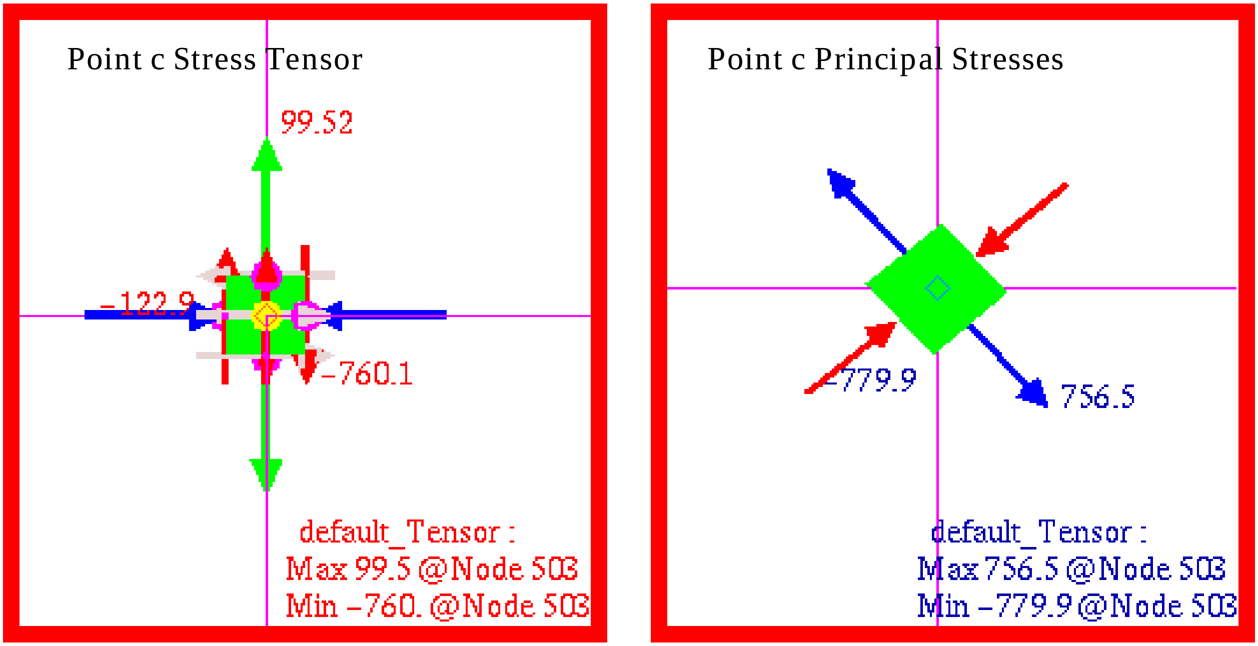


Figure 15-26 Stress Tensor and Principal Stresses at Point c.

Problem 4: Linear Statics, Plane Strain with 2D Solids

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR and CTRIAR Elements with Standard and Revised Formulations

Reference:

Roark, R. J., and Young, W. C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, p. 504.

Problem Description:

An infinitely long, thick walled, cylinder is subjected to a uniform internal pressure. Assuming near incompressible material behavior, find the radial displacement as well the radial and hoop stress at the inner diameter (ID) and outer diameter (OD) of the cylinder.

Engineering Data:

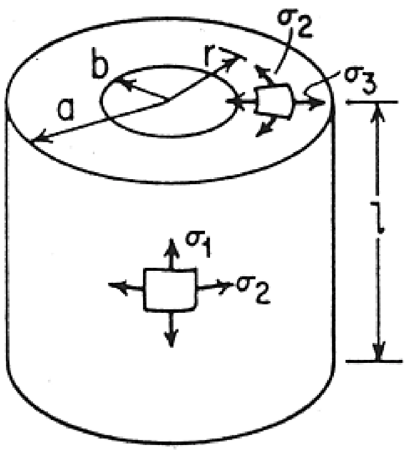
E=1000. psi

$\nu = 0.4999$

a = 9.0 inches

b = 3.0 inches

q = 100. psi



Theoretical Solution:

For the case of an internally pressurized thick walled cylinder, the displacement and stresses at the inner and outer radius are given by the equations below. Upon substituting the assumed values for E, υ, a, b, and q, the following displacements and stresses are calculated:

$$\sigma_1 = 0$$

$$\sigma_2 = \frac{qb^2(a^2 + r^2)}{r^2(a^2 - b^2)} = \begin{pmatrix} 25.00psi \text{ , r=a} \\ 125.00psi \text{ , r=b} \end{pmatrix}$$

$$max\ \sigma_2 = q\frac{a^2 + b^2}{a^2 - b^2} \text{ at r=b}$$

$$\sigma_3 = \frac{-qb^2(a^2 - r^2)}{r^2(a^2 - b^2)} = \begin{pmatrix} 0 \text{ , r=a} \\ -100.0psi \text{ , r=b} \end{pmatrix}$$

$$max\ \sigma_3 = -q \text{ at r=b}$$

$$\Delta a = \frac{q}{E}\frac{2ab^2}{a^2 - b^2} = 0.2250 inches$$

$$\Delta b = \frac{qb}{E}\left(\frac{a^2 + b^2}{a^2 - b^2} + v\right) = 0.5250 inches$$

MSC.Nastran Results:

For the purposes of this problem, a 15 degree segment was modeled of the cross-section of the pipe with the appropriate axisymmetric boundary conditions being applied to the lateral edges of the model. In addition, five individual segments were modeled that were meshed with either CQUAD4, CQUAD8, CTRIA3 or CTRIA6 elements using a standard formulation or CQUADR and CTRIAR elements with a revised formulation. This was done to assess how variations in element topology and formulation affect overall accuracy of the results when performing a plane strain analysis with a high Poisson's ratio. The actual model that was generated with MSC.Patran is shown in **Figure 15-27** where the individual element types associated with each segment have been identified. The loading and boundary conditions that were applied to the model are shown in **Figure 15-28**.

Using the aforementioned model, the following results were obtained for each of the various element types.

Table 15-11 Displacement, Δa

	CQUAD4	CQUAD8	CTRIA3	CTRIA6	CQUADR	CTRIAR
MSC.Nastran*	.1370	.1690	.1680	.1689	.1755	.1726
Theory	.2250	.2250	.2250	.2250	.2250	.2250
%, Difference	-39.11%	-24.89%	-25.33%	-24.93%	-22.00%	-23.29%

Table 15-12 Displacement, Δb

	CQUAD4	CQUAD8	CTRIA3	CTRIA6	CQUADR	CTRIAR
MSC.Nastran*	.4102	.5060	.5031	.5058	.5357	.5256
Theory	.5250	.5250	.5250	.5250	.5250	.5250
%, Difference	-21.87%	-3.62%	-4.17%	-3.66%	2.04%	0.11%

Table 15-13 Stresses, $\sigma_{2,\, r\, =\, a}$

	CQUAD4	CQUAD8	CTRIA3	CTRIA6	CQUADR	CTRIAR
MSC.Nastran*	-165.1	25.32	25.00	25.01	25.30	26.44
Theory	25.00	25.00	25.00	25.00	25.00	25.00
%, Difference	-760.40%	1.28%	0.0%	0.04%	1.20%	5.76%

Table 15-14 Stresses, $\sigma_{2,\, r\, =\, b}$

	CQUAD4	CQUAD8	CTRIA3	CTRIA6	CQUADR	CTRIAR
MSC.Nastran*	1766.	129.4	124.0	124.3	131.7	129.3
Theory	125.00	125.00	125.00	125.00	125.00	125.00
%, Difference	1312.8%	3.52%	-0.80%	-0.56%	5.36%	3.44%

Table 15-15 Stresses, $\sigma_{3,\, r\, =\, a}$

	CQUAD4	CQUAD8	CTRIA3	CTRIA6	CQUADR	CTRIAR
MSC.Nastran*	-186.0	.3775	.7289	.0476	3.809	-3.039
Theory	0.0	0.0	0.0	0.0	0.0	0.0
%, Difference	-	-	-	-	-	-

Table 15-16 Stresses, $\sigma_{3,\, r\, =\, b}$

	CQUAD4	CQUAD8	CTRIA3	CTRIA6	CQUADR	CTRIAR
MSC.Nastran*	1591.0	-94.52	-91.00	-99.47	-52.96	-119.6
Theory	-100.0	-100.0	-100.0	-100.0	-100.0	-100.0
%, Difference	1691%	- 6.48%	-9.00%	-.53%	-47.04%	19.60%

* Represents average of all edge nodal values with nodal values averaged across adjacent elements.

The fringe plot that was generated for the radial displacement is shown in **Figure 15-29**. Fringe plots for the radial and hoop stress are shown in **Figure 15-30** and **Figure 15-31**. Here the stresses have been plotted with an adjusted scale that better shows the stress gradient present in each of the segments. All plots have been transformed into the cylindrical coordinate system defined in the problem. A comparison with the preceding tabular results clearly shows that MSC.Patran is accurately reproducing the MSC.Nastran results.

The preceding results clearly demonstrate the wide degree of variability in the result attributable to element topology as well as formulation. Consistently, the higher order CQUAD8 and CTRIA6 elements gave superior performance compared to their linear counterparts. Similarly, in this application where cylindrical geometry was involved, a triangular element gave far more accurate results then compared to a quadrilateral element. Only by adopting the revised formulation of a CQUADR could acceptable results be obtained with a quadrilateral element. This would not be unexpected since the removal of any membrane-bending coupling produces far less sensitivity to extreme values in Poisson’s ratio as in this example.

File(s):<install_dir>/results_vv_files/prob004.bdf, prob004.op2

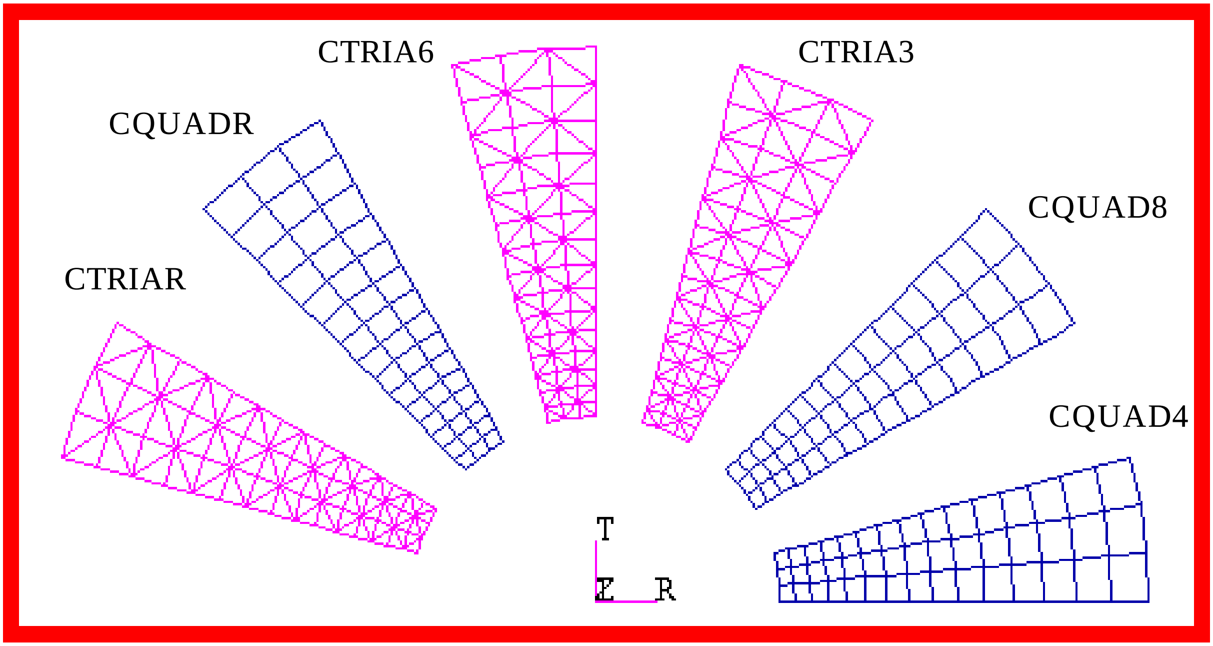


Figure 15-27 Basic Models Using 2D Solid Elements.

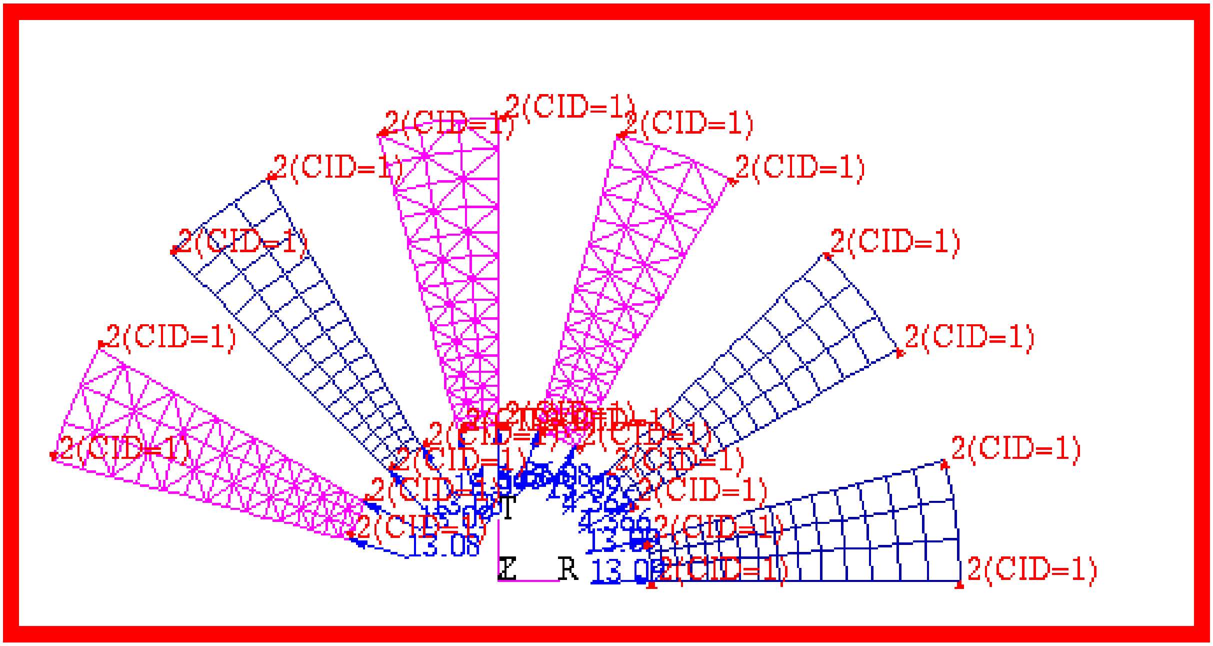


Figure 15-28 Loads and Boundary Conditions of 2D Solid Models.

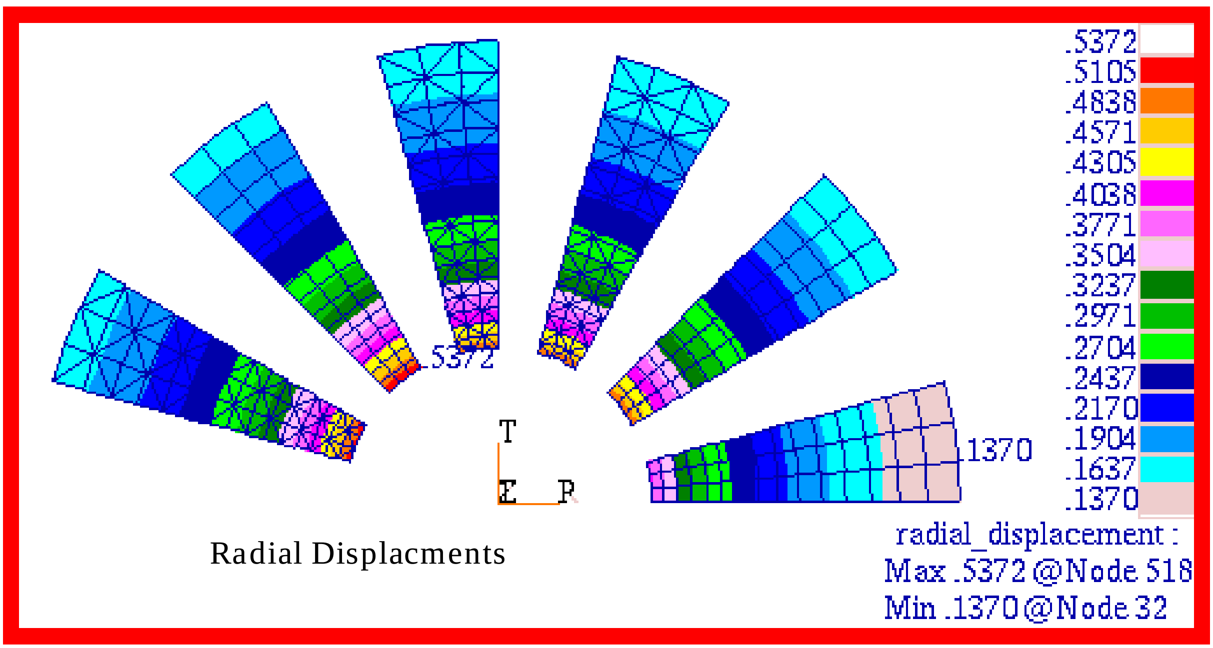


Figure 15-29 Fringe Plot of Radial Displacement on 2D Solids.

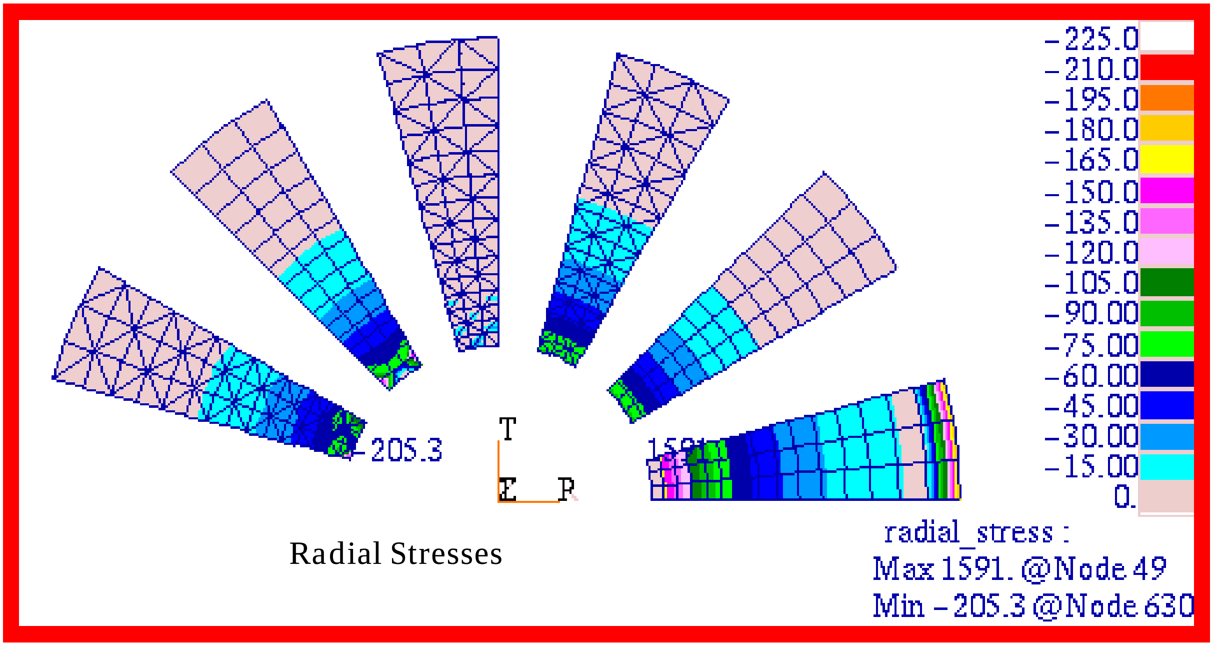


Figure 15-30 Fringe Plot of Radial Stress, σ_3, on 2D Solid Elements.

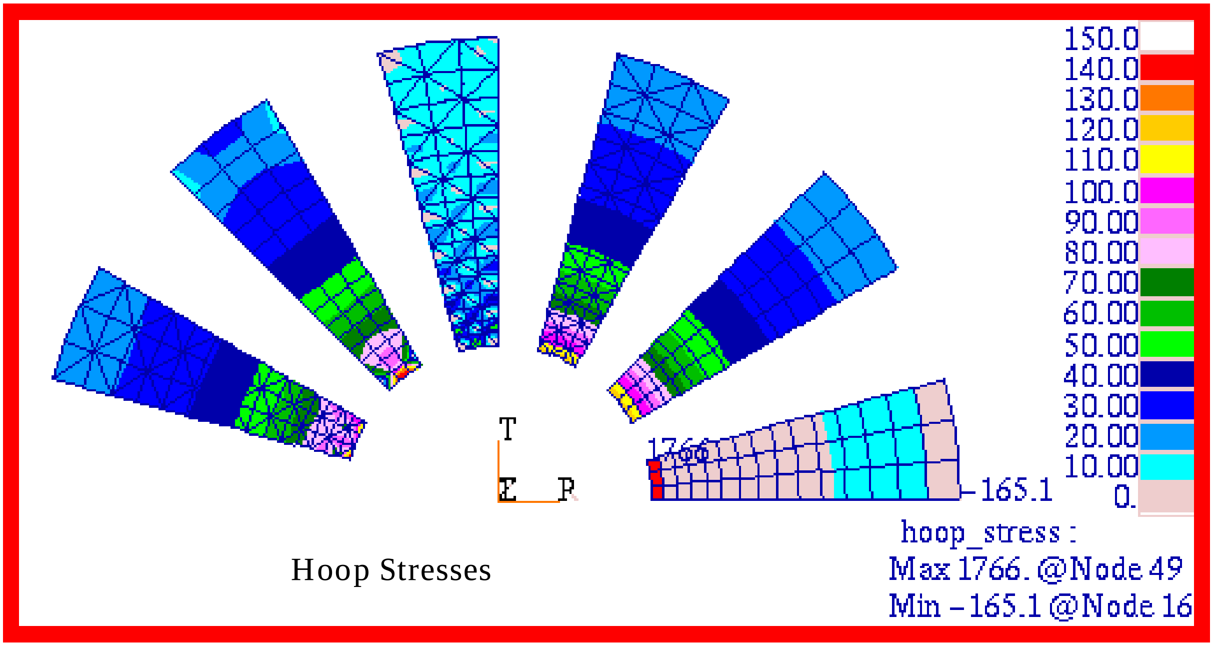


Figure 15-31 Fringe Plot of Hoop Stress, σ_2, on 2D Solid Elements.

Problem 5: Linear Statics, 2D Shells in Spherical Coordinates

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, CQUAD4 and CTRIA3 with Standard Formulation

Reference:

Roark, R. J., and Young, W.C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, p. 451.

Problem Description:

A section of a thin, spherical shell is internally pressurized. The free edge of the shell is restrained in the meridional direction. Find the stresses and displacements of the shell.

Engineering Data:

$t = 0.05$ in

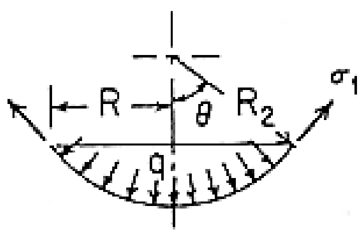
$R_2 = 10.0$ in

$q = 200.0$ psi

$\theta = 45.0$ degrees

$E = 1.0 \times 10^7$ psi

$\upsilon = 0.333$



Theoretical Solution:

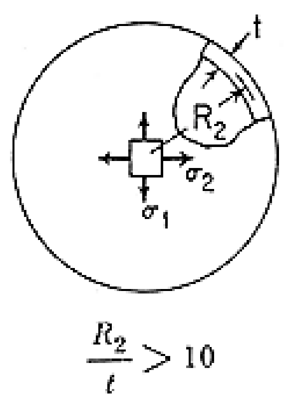
For the case of an internally pressurized spherical shell tangentially supported, the predicted stresses and displacements are, after substituting the assumed values for E, υ, t, and R_2:

$$\sigma_1 = \sigma_2 = \frac{qR_2}{2t} = 20,000.psi$$

$$\Delta R_2 = \frac{qR_2^2(1-v)}{2Et} = 0.013340in$$

$$\Delta y = \frac{qR_2^2(1-v)(1-\cos\theta)}{2Et} = 0.003907in$$

$$\Delta R = \frac{qR_2^2(1-v)\sin\theta}{2Et} = 0.009433in$$



MSC.Nastran Results:

A model of the spherical shell was created using MSC.Patran. Due to symmetry, only one fourth of the shell was modeled with the appropriate axisymmetric boundary conditions applied to the free edges. The model that was created is shown in Figure 15-32. The majority of the model was meshed with CQUAD4 elements, with the mesh becoming increasingly refined near the center of the shell. Only at the very apex of the shell are CTRIA3 elements used. However, every attempt was made to minimize having high aspect ratio triangular elements that would otherwise cause a loss of accuracy. Using this model, the following results were obtained.

Table 15-17 2D Shell Displacements

Source	ΔR_2		ΔR	Δy
	max	min		
Theory	.01334	.01334	.009433	.003907
MSC.Nastran	.01462	.01383	.010330	.003890
%, Difference	9.60%	3.67%	9.51%	0.435%

Table 15-18 2D Shell Stresses*

Source	σ_1		σ_2	
	max	min	max	min
Theory	20,000.	20,000.	20,000.	20,000.
MSC.Nastran	20,034.	19,881.	20,563.	19,877.
%, Difference	0.170%	0.595%	2.82%	0.615%

* Based upon nodal averaging of results between adjacent elements.

The corresponding fringe plots for the radial displacement and the x-displacement in global coordinates are shown in **Figure 15-33** and **Figure 15-34**. The radial displacement plot gives ΔR_2 whereas the max value shown on the x-displacement plot corresponds to ΔR. In **Figure 15-35**, the y-displacement in global coordinates is plotted. The difference between the maximum and minimum y-displacements corresponds to Δy above, or the change in vertical height for the shell. Fringe plots for the radial, meridional and circumferential stresses are shown in **Figure 15-36** through **Figure 15-38**. A comparison of the MSC.Patran fringe plots with the MSC.Nastran results shows an exact correlation between the two.

File(s):<install_dir>/results_vv_files/prob005.bdf, prob005.op2

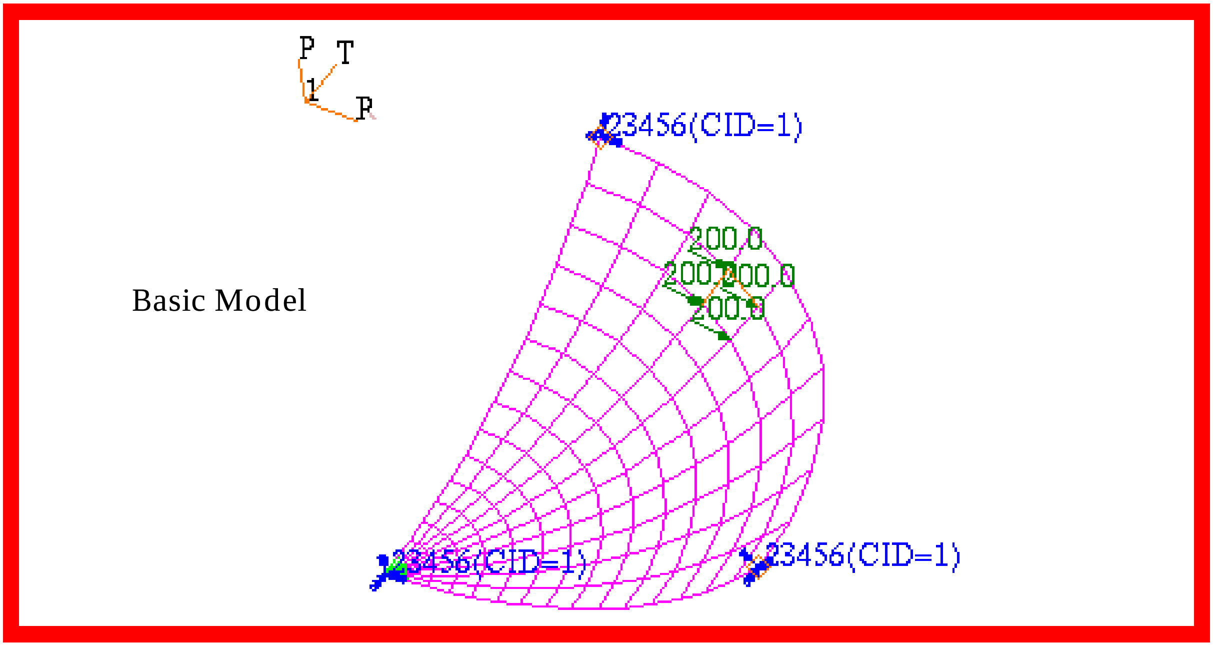


Figure 15-32 Basic Model of 2D Shell in Spherical Coordinates.

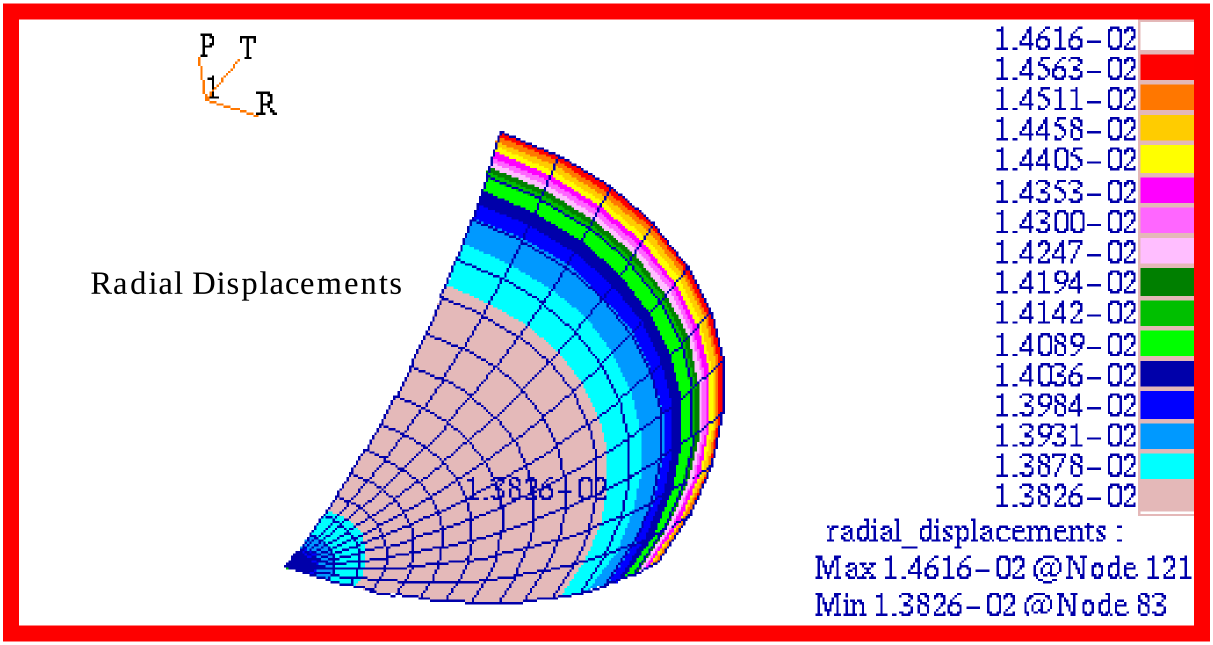


Figure 15-33 Radial Displacement of Spherical Membrane.

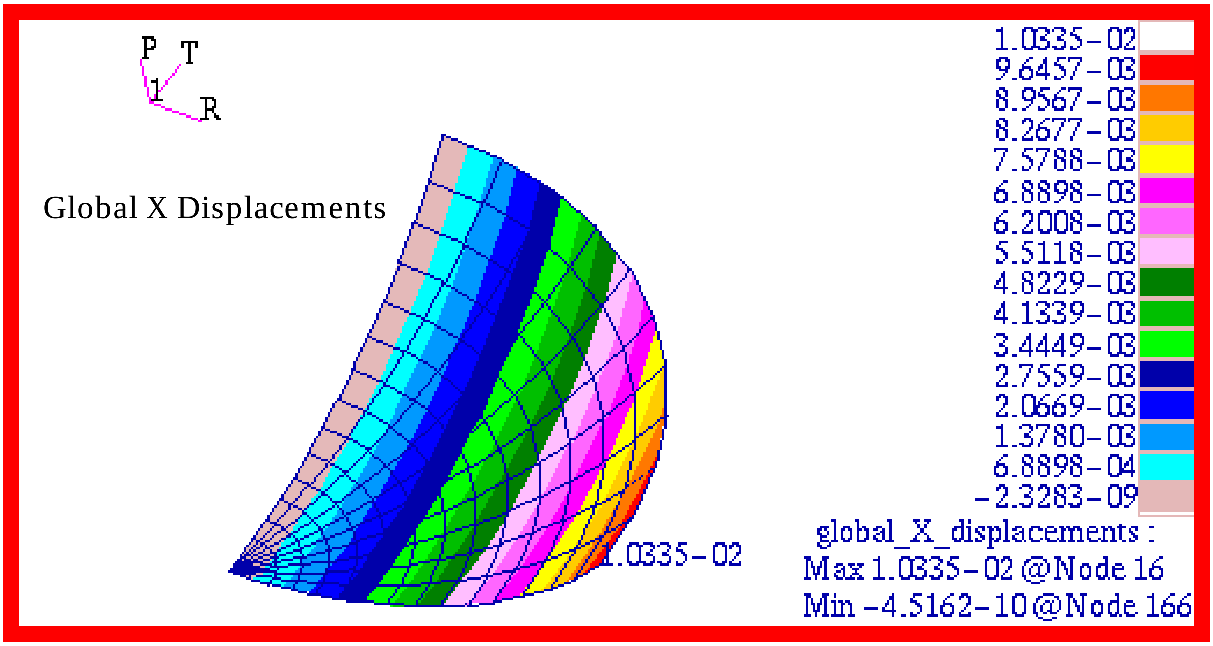


Figure 15-34 Global Translational X Displacement of Membrane.

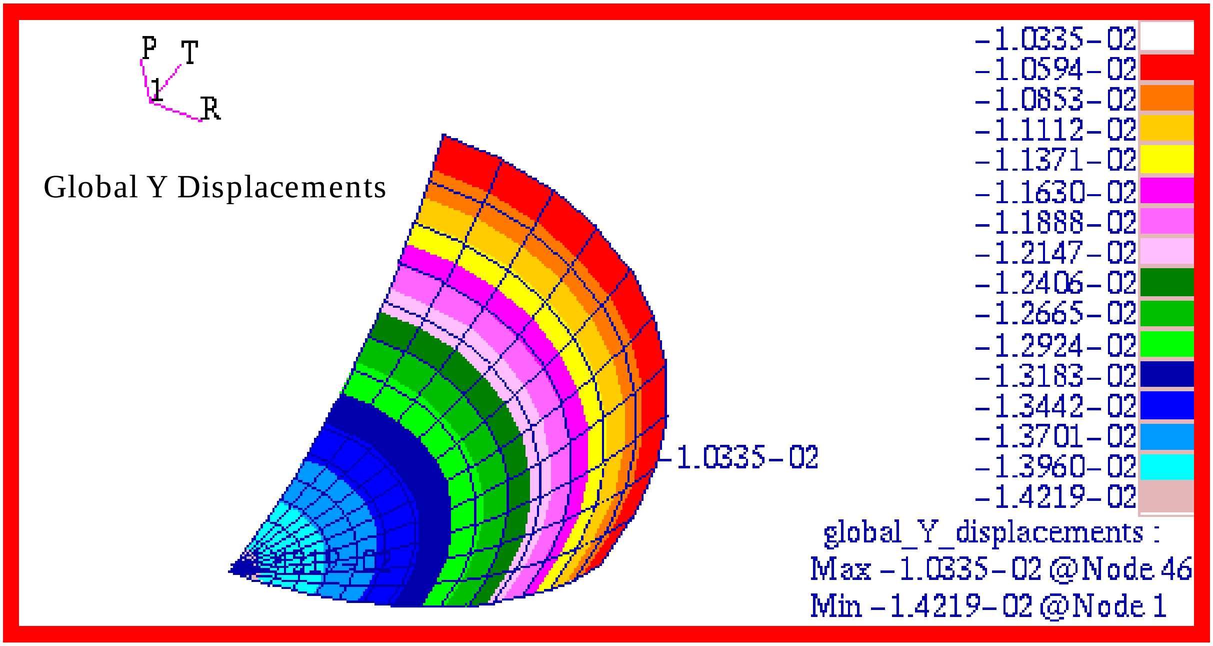


Figure 15-35 Global Translational Y Displacement of Membrane.

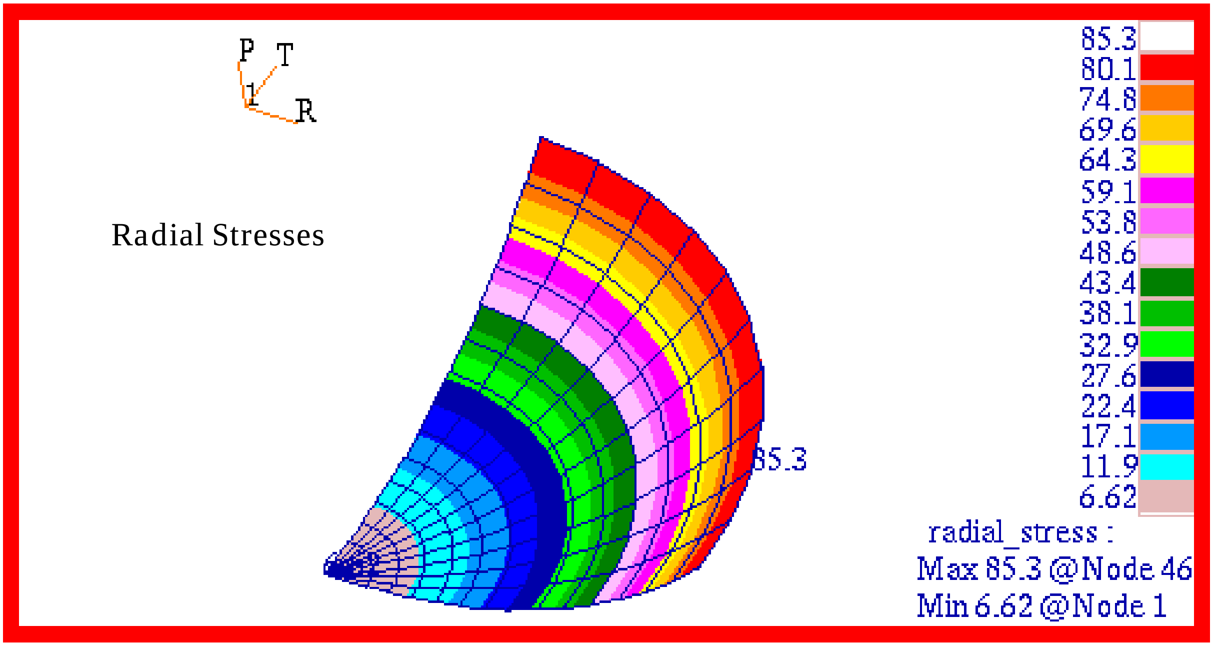


Figure 15-36 Radial Stresses of Spherical Membrane.

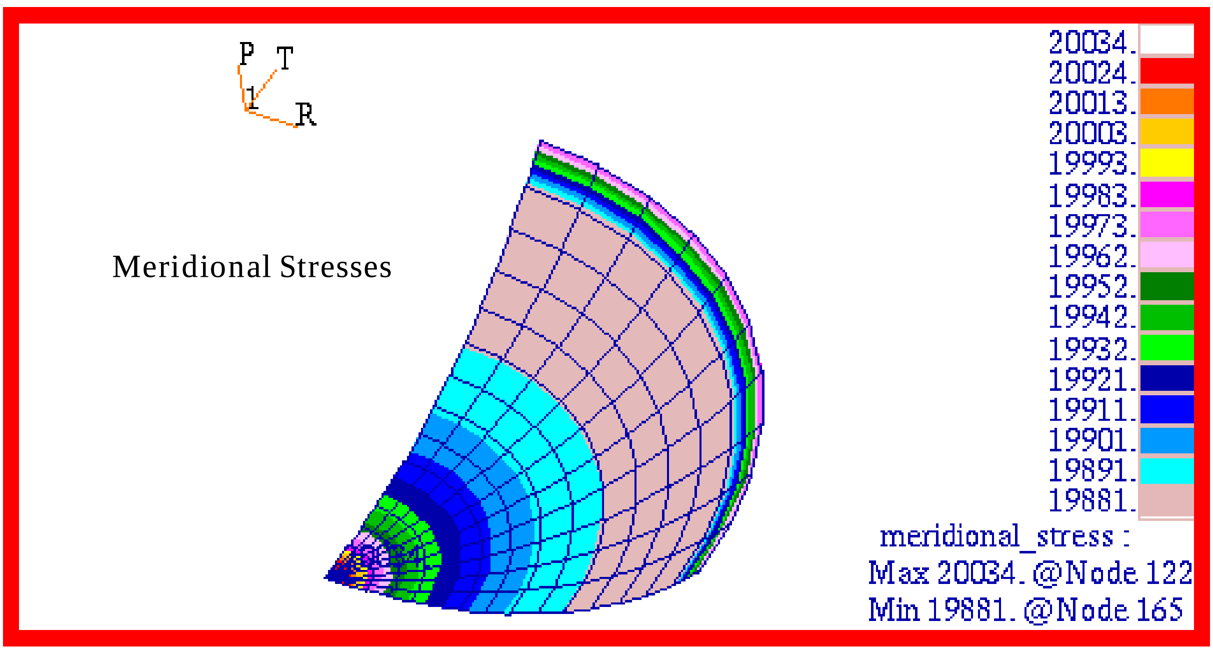


Figure 15-37 Meridional Stresses of Spherical Membrane.

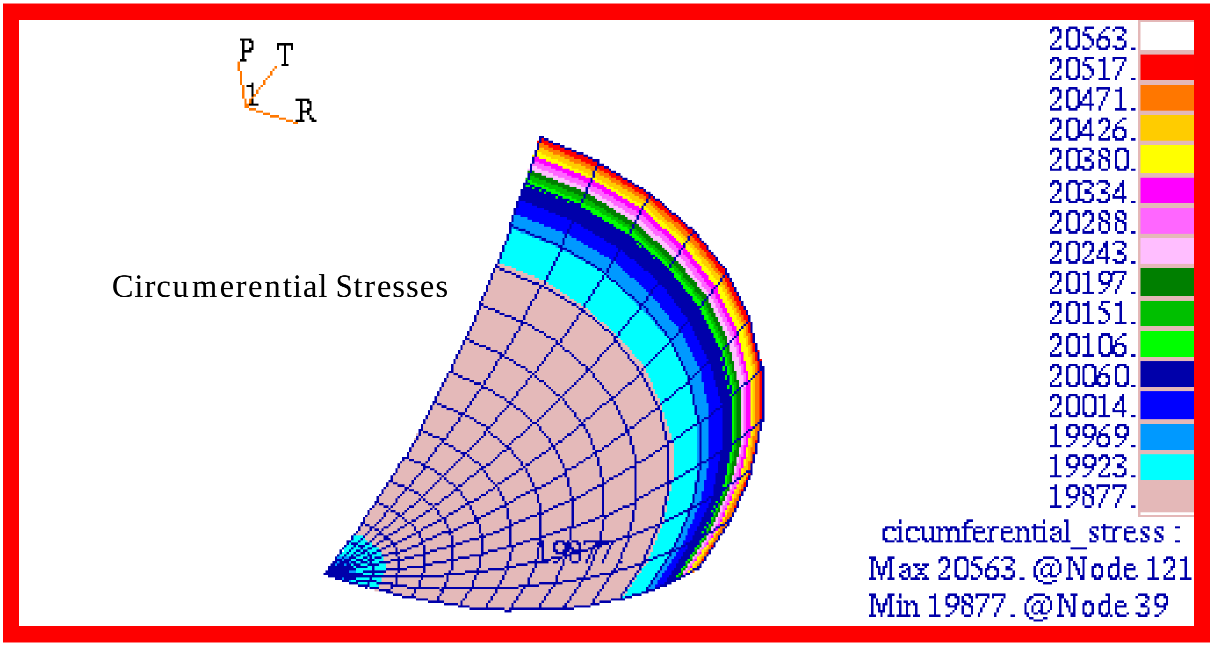


Figure 15-38 Circumferential Stresses of Spherical Membrane.

Problem 6: Linear Statics, 2D Axisymmetric Solids

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, CTRIAX6 Elements, 2D Axisymmetric Solids

Reference:

Roark, R. J., and Young, W. C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, pp. 504-505.

Problem Description:

A thick walled cylinder with thick end caps is subjected to a uniform external pressure. Determine both the radial and hoop displacement and stress state at the mid plane of the cylinder.

Engineering Data:

$E = 1.0 \times 10^7$ psi

$\upsilon = 0.333$

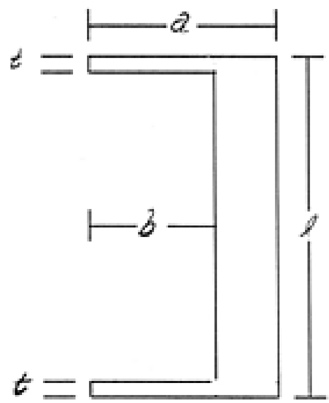
a = 6.0 in

b = 4.0 in

l = 11.0 in

t = 0.5 in

$q = 1000.0\ \text{lb}/\text{in}^2$



Theoretical Solution:

For the case of an externally pressurized thick walled cylinder with thick end caps, there is no closed form solution. The nearest solution corresponds to a capped thick walled cylinder where the restraint afforded by the end caps to radial and hoop contraction is ignored. The presence of the thick end caps would necessarily reduce the radial and hoop displacement at the mid plane of the cylinder, resulting in lower axial and hoop stresses. Only the radial stress would be left unaltered at the mid plane. However, a substantial gradient in all of the stress components should be observed near the ends of the cylinder due to bending of the end caps. Nevertheless, the classical closed form solution is useful in bounding the actual displacements and stresses and is as follows. For the assumed values for E, υ, a, b, ℓ and q, the following results are obtained for an externally pressurized thick walled cylinder with negligible end caps:

$$\sigma_1 = \frac{-qa^2}{a^2 - b^2} = -1800 psi$$

$$\sigma_2 = \frac{qb^2(a^2 + r^2)}{r^2(a^2 - b^2)} = -2600\ psi \text{ at } r = a = 6.0 inches$$

$$\max \sigma_2 = q\frac{a^2 + b^2}{a^2 - b^2} = -3600\ psi \text{ at } r = b = 4.0 inches$$

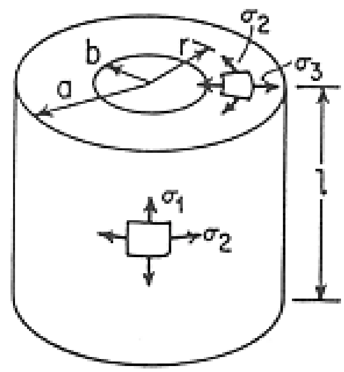
$$\sigma_3 = \frac{-qb^2(a^2 - r^2)}{r^2(a^2 - b^2)} = 0\ psi \text{ at } r = b = 4.0\,inches$$

$$\max \sigma_3 = -q = -1000\ psi \text{ at } r = a = 6.0\,inches$$

$$\Delta a = \frac{-qa}{E}\frac{a^2(1 - 2v) + b^2(1 + v)}{a^2 - b^2} = -0.001006\,inches$$

$$\Delta b = \frac{-qb}{E}\frac{a^2(2 - v)}{a^2 - b^2} = -0.001202\,inches$$

$$\Delta l = \frac{-ql}{E}\frac{a^2(1 - 2v)}{a^2 - b^2} = -0.000673\ inches$$



MSC.Nastran Results:

Two models were created with MSC.Patran of the thick walled cylinder. One featured 3 noded and the other 6 noded CTRIAX6 elements. Both models used the same number of elements. The models with the applied loading are shown in Figure 15-39, which was assumed to be a uniform external ambient pressure of 1000.0 psi. Using these models, results were obtained and the following errors occurred relative to the theoretical values for each of the element topologies that were examined. The results clearly show the expected behavior; namely, the presence of the end caps reduces the inward radial contraction at the center of the cylinder. This necessarily reduces the predicted hoop stress relative to the theoretical value. The only stress components that should correlate with the theoretical values are the radial stress at the ID and OD of the cylinder ad the axial stress at the mean radius of the cylinder.

Table 15-19 Radial Stress, σ_3*

	3 Noded CTRIAX		6 Noded CTRIAX	
	r* = *a	***r* = *b***	***r* = *a***	***r* = *b***
Theory	-1000.0	0.0	-1000.0	0.0
MSC.Nastran	-903.576	-268.901	-1000.697	-9.847213
% Difference	-9.64%	-	0.0697%	-

Table 15-20 Hoop Stress, σ_2*

	3 Noded CTRIAX		6 Noded CTRIAX	
	r* = *a	***r* = *b***	***r* = *a***	***r* = *b***
Theory	-2600.0	-3600.0	-2600.0	-3600.0
MSC.Nastran	-2363.289	-2967.001	-2467.246	-2740.299
% Difference	-9.104%	-17.58%	-5.12%	-23.88%

Table 15-21 Axial Stress at Mean Diameter, σ_1*

	3 Noded CTRIAX	6 Noded CTRIAX
Theory	-1800.0	-1800.0
MSC.Nastran	-1871.255	-1876.067
% Difference	3.96%	4.23%

Table 15-22 Displacements*

	3 Noded CTRIAX			6 Noded CTRIAX		
	Δa	Δb	$\Delta \ell$	Δa	Δb	$\Delta \ell$
Theory	-1.006E-3	-1.202E-3	-6.73E-4	-1.006E-3	-1.202E-3	-6.73E-4
MSC.Nastran	-7.986E-4	-9.586E-4	-6.340E-4	-8.104E-4	-9.814E-4	-6.297E-4
% Difference	-25.97%	-20.247%	-6.145%	-19.441%	-18.354%	-6.873%

* Results based upon nodal average of adjacent elements.

The fringe plots created of the radial displacement are shown in **Figure 15-40** and **Figure 15-41**. The range has been adjusted to better show the displacement at the center of the cylinder. Similarly, the radial stress distribution is shown in **Figure 15-42** and **Figure 15-43**; the hoop stress in **Figure 15-44** and **Figure 15-45**; and the axial stress in **Figure 15-46** and **Figure 15-47**. A comparison of the MSC.Patran fringe plots with the preceding tabular results clearly reveals that the MSC.Nastran results are being accurately displayed.

File(s):<install_dir>/results_vv_files/prob006.bdf, prob006.op2

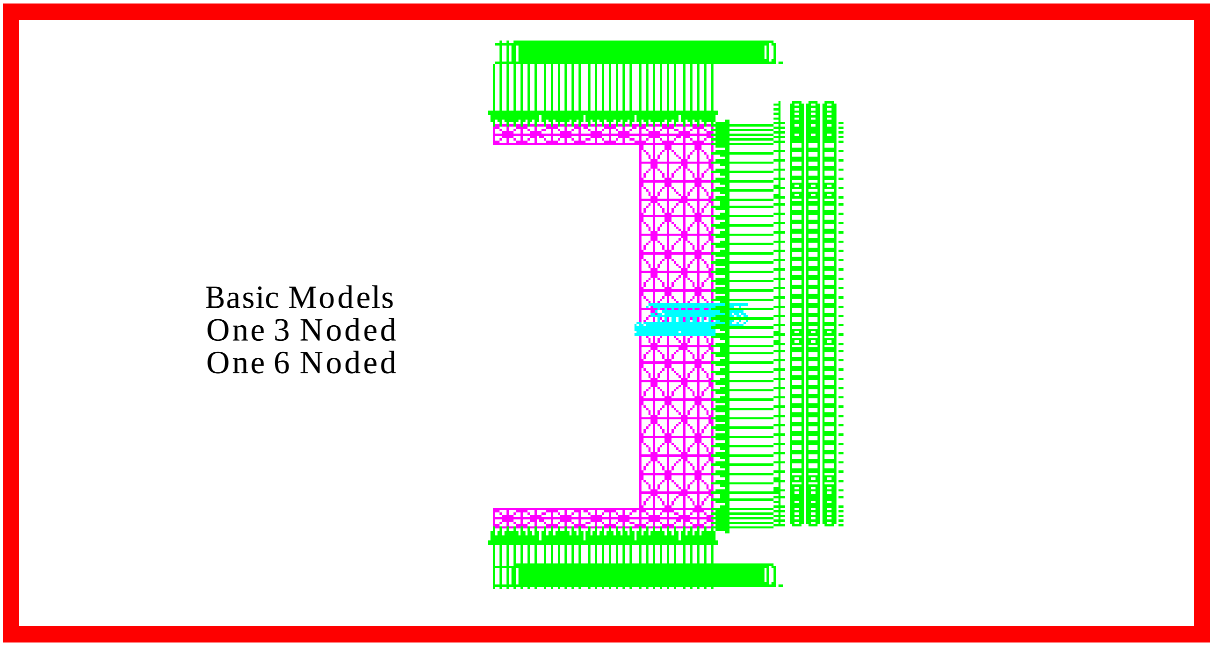


Figure 15-39 Axisymmetric Models of Thick Walled Cylinder.

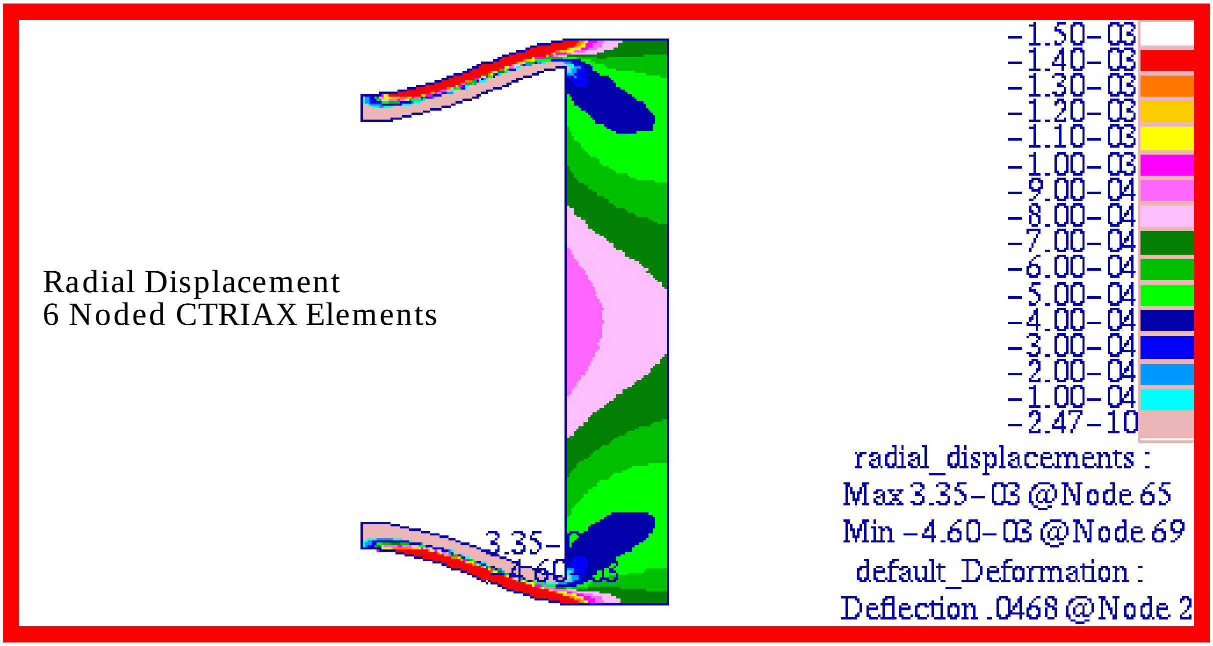


Figure 15-40 Radial Displacement of 6 Noded Axisymmetric Model.

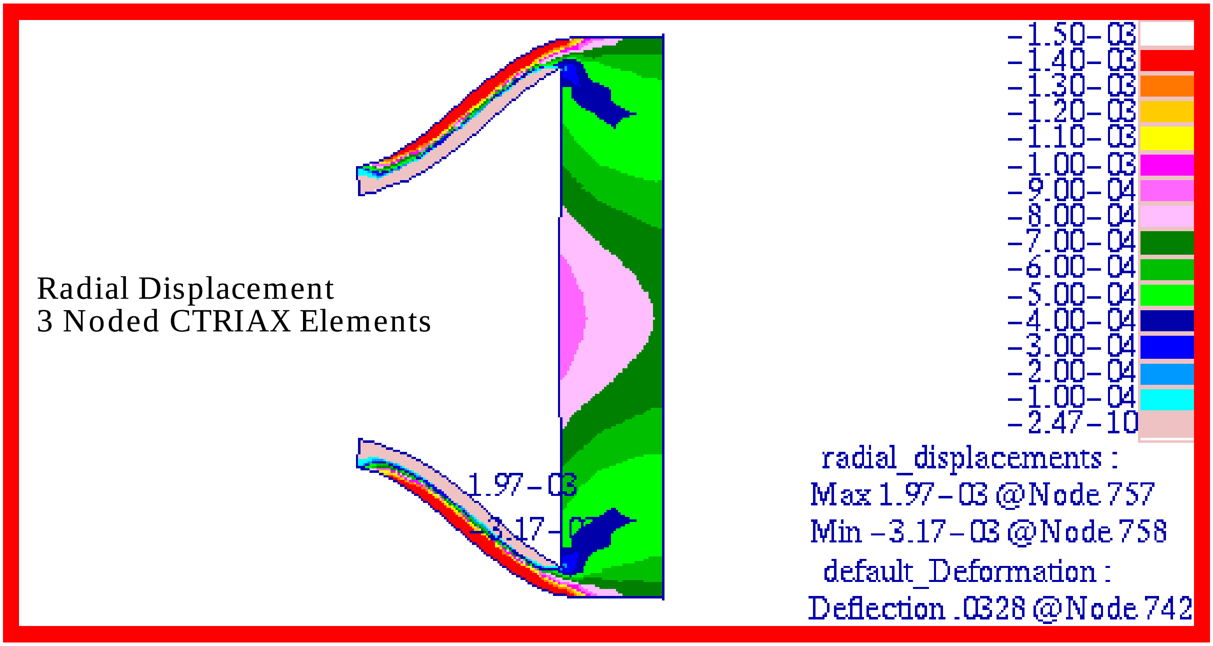


Figure 15-41 Radial Displacement of 3 Noded Axisymmetric Model.

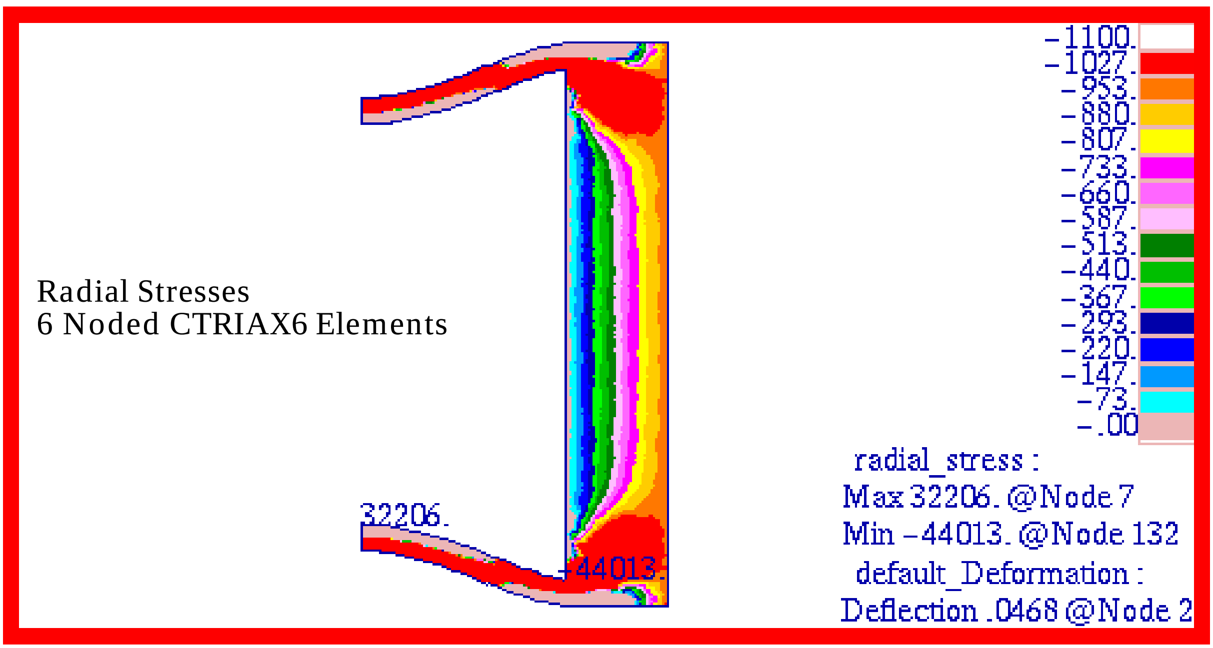


Figure 15-42 Radial Stress in 6 Noded Axisymmetric Model.

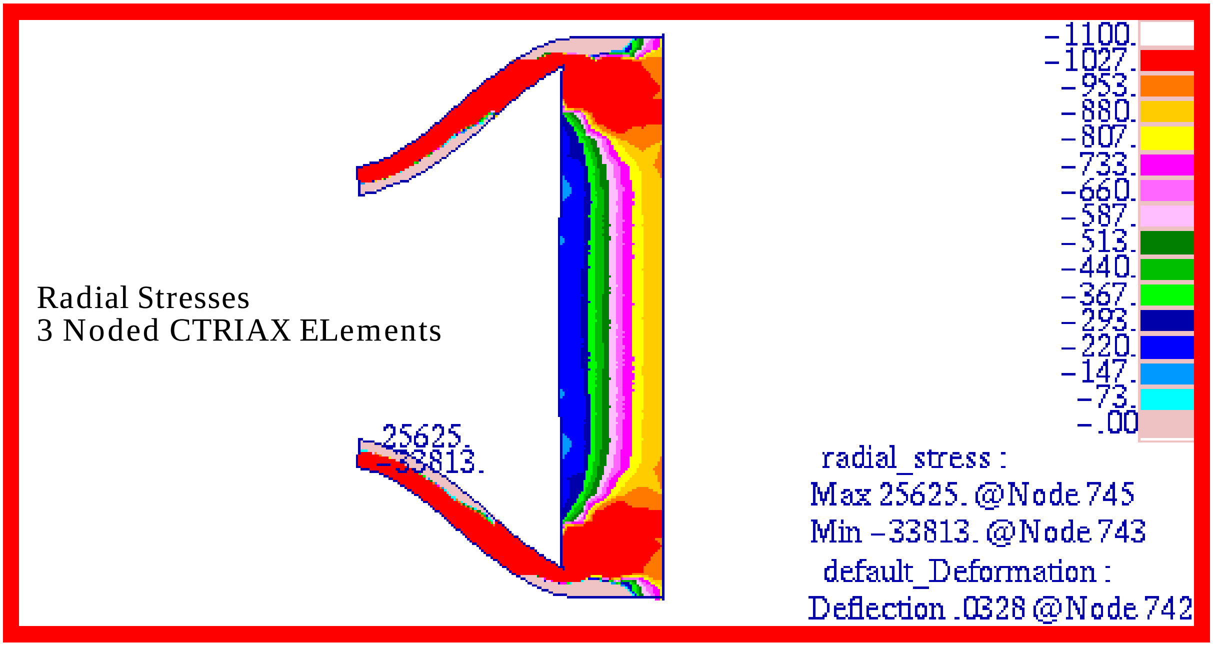


Figure 15-43 Radial Stress in 3 Noded Axisymmetric Model.

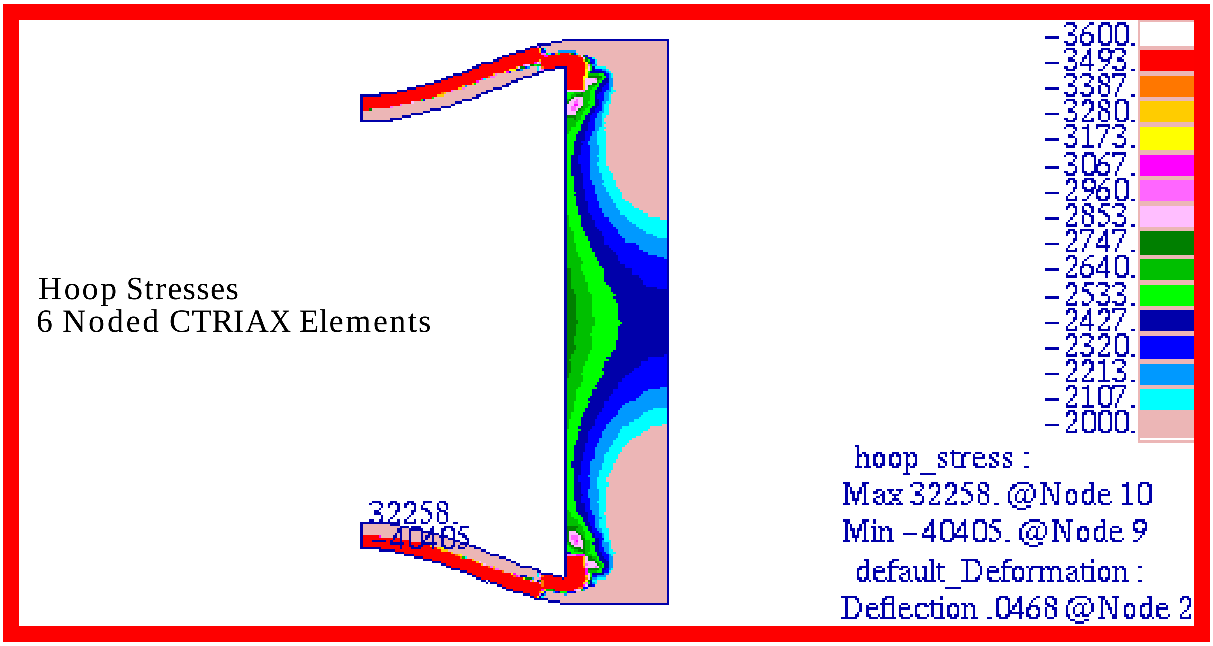


Figure 15-44 Hoop Stress in 6 Noded Axisymmetric Model.

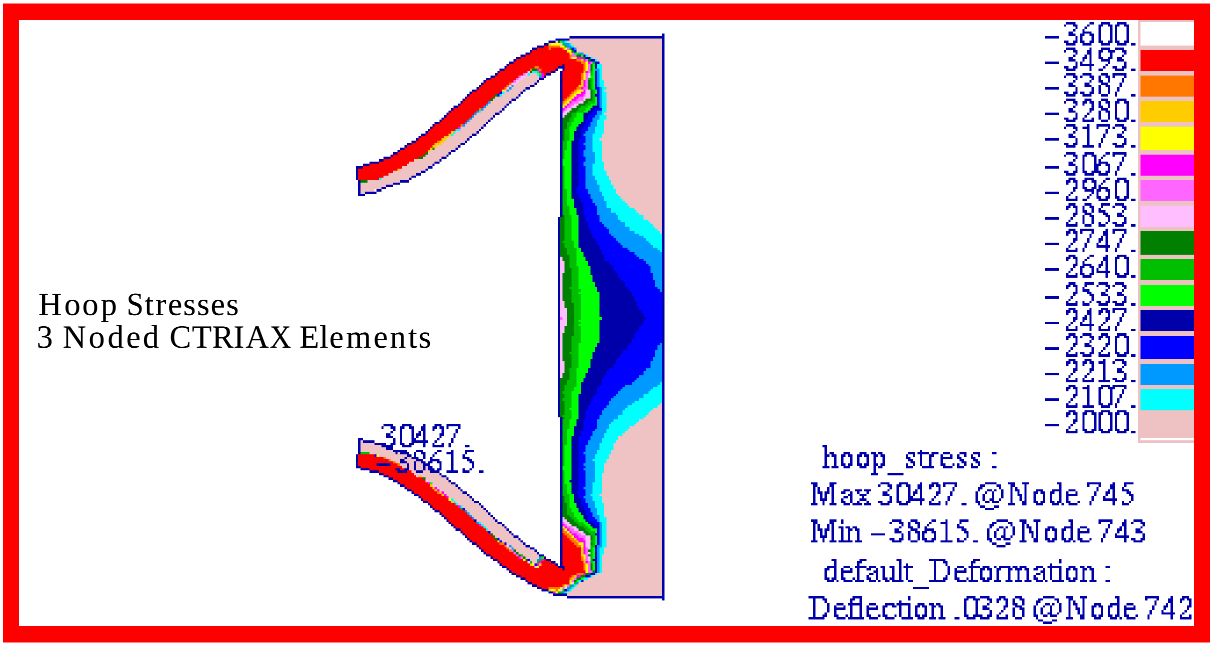


Figure 15-45 Hoop Stress in 3 Noded Axisymmetric Model.

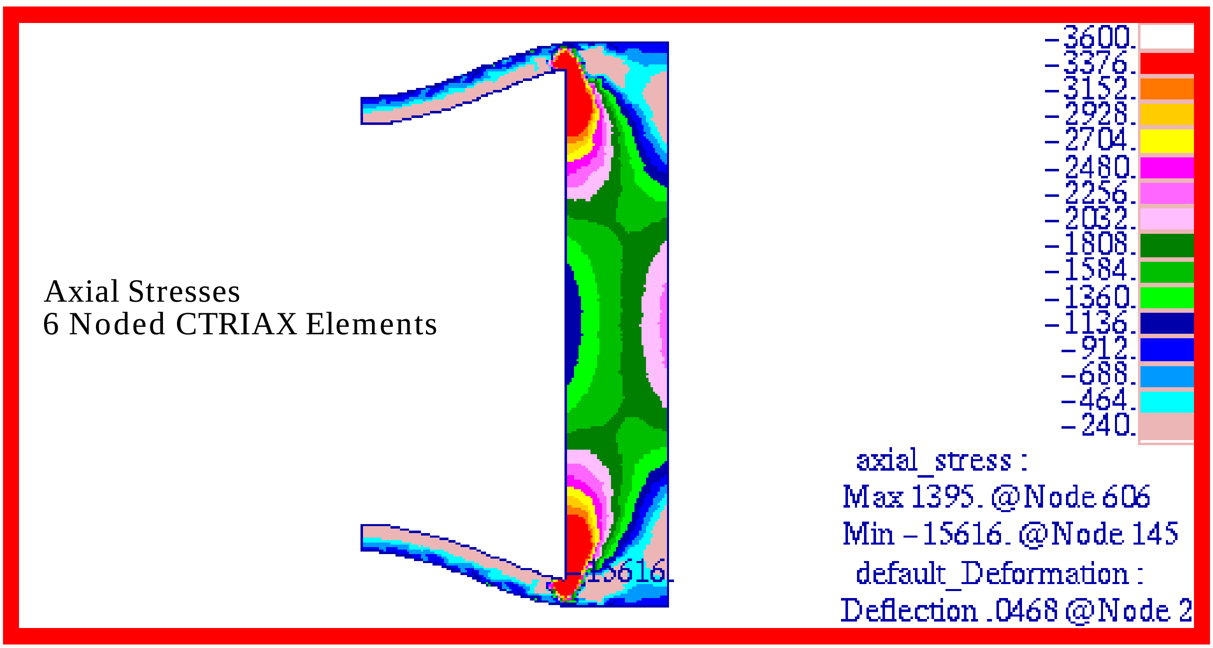


Figure 15-46 Axial Stress in 6 Noded Axisymmetric Model.

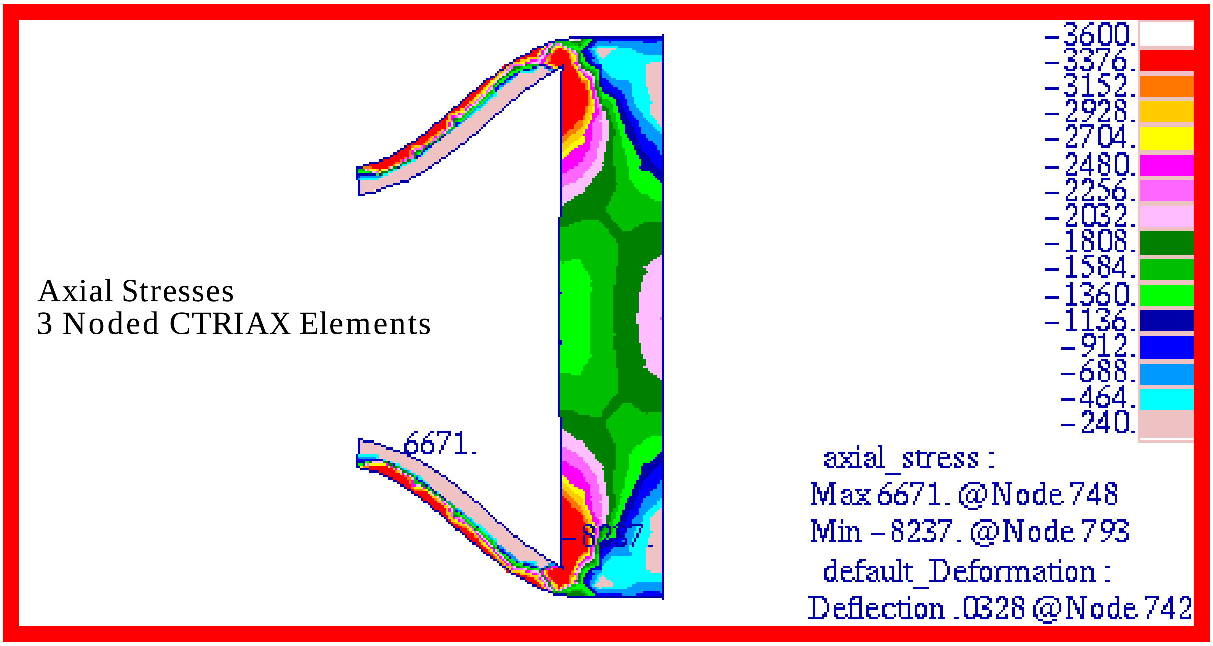


Figure 15-47 Axial Stress in 3 Noded Axisymmetric Model.

Problem 7: Linear Statics, 3D Solids and Cylindrical Coordinate Frames

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, CHEX8 Elements - 3D Solids

Reference:

Roark, R.J., and Young, W. C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, pp. 504-505.

Problem Description:

This is a repeat of **Problem 6: Linear Statics, 2D Axisymmetric Solids** (p. 351) with the exception that the cylinder and end caps are entirely modeled with MSC.Nastran 3D solid HEX8 elements. In addition, all analytical results are recovered in a global cylindrical coordinate frame. This should produce results that are identical to those recovered with CTRIAX6 2D axisymmetric solid elements.

Engineering Data:

$E = 1.0 \times 10^7$ psi

$\upsilon = 0.333$

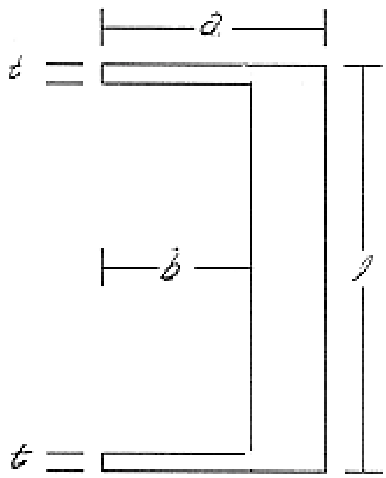
$a = 6.0$ in

$b = 4.0$ in

$l = 11.0$ in

$t = 0.5$ in

$q = 1000.0$ psi



Theoretical Solution:

See **Problem 6: Linear Statics, 2D Axisymmetric Solids** (p. 351) for derivation and theoretical results.

MSC.Nastran Results:

A model was created of the thick walled cylinder with the end caps using just MSC.Nastran 3D solid CHEX8 elements. Due to symmetry, only one eighth of the cylinder was modeled, with the appropriate axisymmetric and symmetry boundary conditions applied to the free faces of the model. The model that was created is shown in **Figure 15-48** and the applied boundary conditions in **Figure 15-49**. Using this model, the results below were computed at the middle of the cylinder.

Once again, the analytical results demonstrate the expected behavior. Namely, the presence of the end caps reduces the inward radial contraction at the center of the cylinder. This reduces the predicted hoop stress relative to the theoretical value. The only stress components that are unaffected are the radial stress at the ID and OD of the cylinder and the axial stress at the mean radius of the cylinder. The percent error that occurred relative to the theoretical values for the radial and axial stress are shown below. For comparative purposes, the results that were obtained for a three noded CTRIAX6 element have been included, illustrating the excellent degree of correlation that exists when using either element type to model axisymmetric structures.

Table 15-23 Radial Stresses, σ_3*

	CHEX8		CTRIAX6, 3 Noded	
	r* = *a	***r* = *b***	***r* = *a***	***r* = *b***
Theory	-1000.0	0.0	-1000.0	0.0
MSC.Nastran	-918.731	-184.862	-903.576	-268.901
% Difference	-8.13%	-	-9.64%	-

Table 15-24 Hoop Stresses, σ_2*

	CHEX8		CTRIAX6, 3 Noded	
	r* = *a	***r* = *b***	***r* = *a***	***r* = *b***
Theory	-2600.0	-3600.0	-2600.0	-3600.0
MSC.Nastran	-2902.487	-2418.523	-2467.246	-2740.299
% Difference	11.634%	-32.819%	-5.12%	-23.88%

Table 15-25 Axial Stresses, σ_1*

	CHEX8	CTRIAX6, 3 Noded
Theory	-1800.0	1800.0
MSC.Nastran	-1760.235	1871.255
% Difference	-2.21%	3.96%

Table 15-26 Displacements*

	CHEX8			CTRIAX6		
	Δa	Δb	$\Delta \ell$	Δa	Δb	$\Delta \ell$
Theory	-1.006E-3	-1.202E-3	-6.73E-4	-1.006E-3	-1.202E-3	-6.73E-4
MSC.Nastran	-8.095E-4	-9.980E-4	-6.043E-4	-8.104E-4	-9.814E-4	-6.297E-4
% Difference	-19.535%	-18.346%	-10.212%	-19.441%	-18.354%	-6.873%

* Results based upon nodal average of adjacent elements.

The corresponding fringe plots that were generated for the radial and axial displacement are shown in **Figure 15-50** and **Figure 15-51**. Similarly, fringe plots for the radial, hoop and axial stress are shown in **Figure 15-52**, **Figure 15-53**, and **Figure 15-54**, respectively. In each of the fringe plots, the range has been purposely adjusted to better show the gradient in displacement and stress that exists at the middle of the cylinder. In addition, all stresses and displacements have been transformed to a cylindrical coordinate frame. If the boundary conditions were properly applied, then an axisymmetric response should be present, which is evident in each of fringe plots.

File(s):<install_dir>/results_vv_files/prob007.bdf, prob007.op2

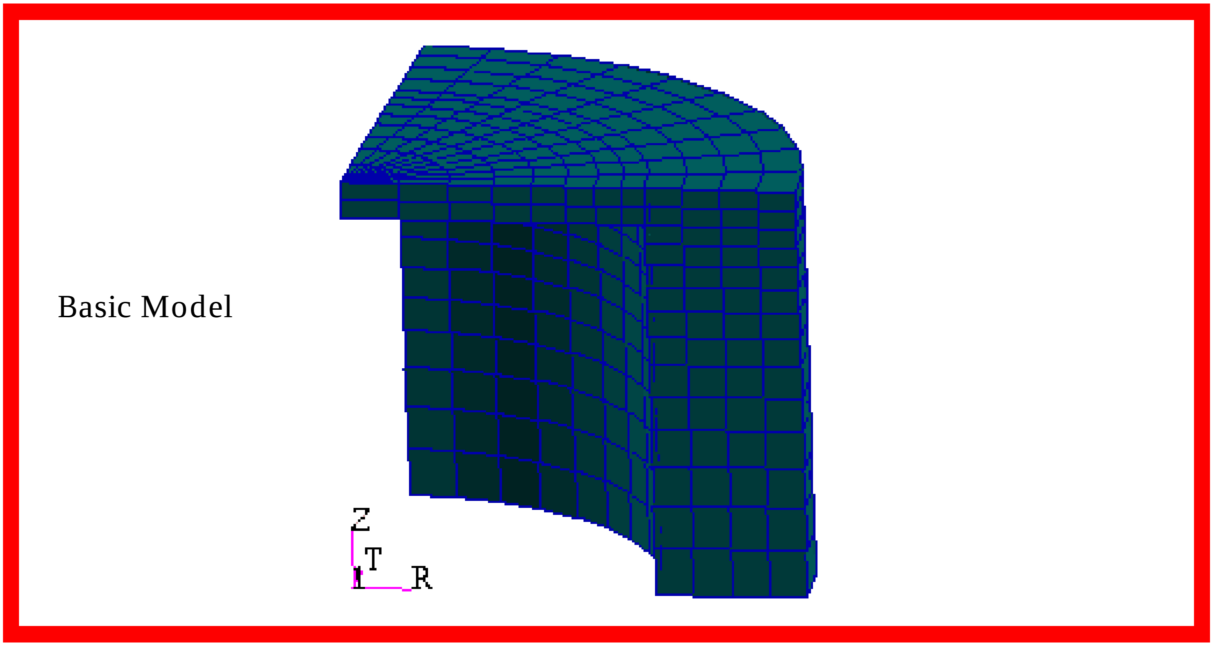


Figure 15-48 Solid Model of Thick Walled Cylinder with End Caps.

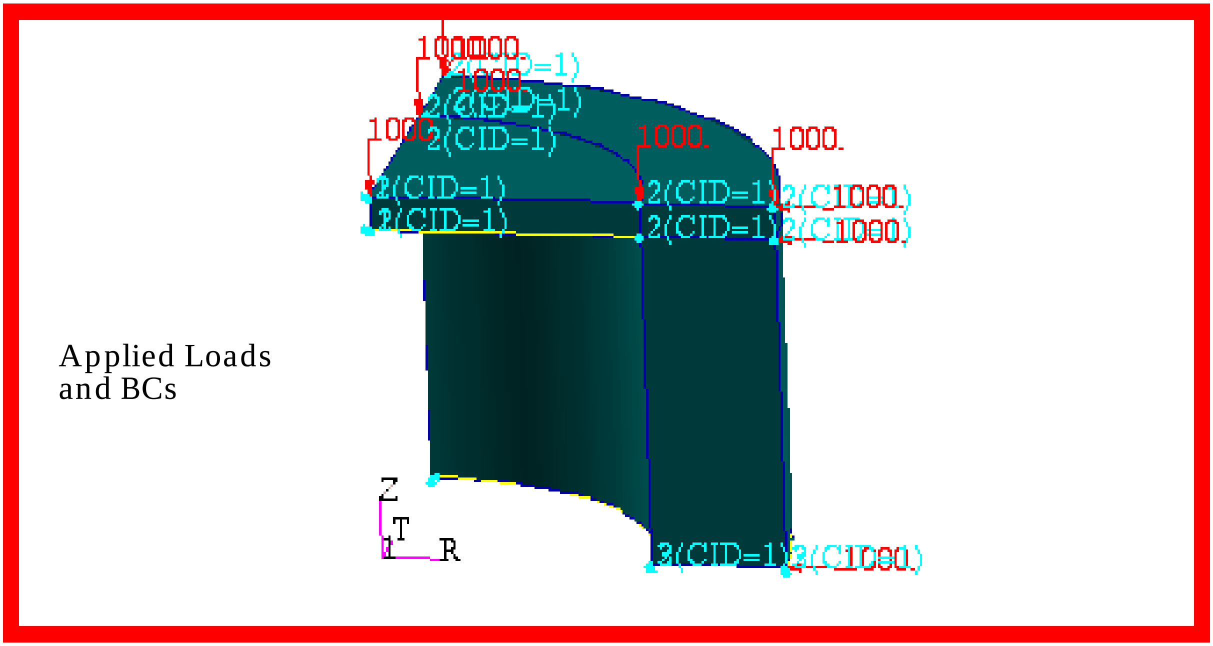


Figure 15-49 Loads and Boundary Conditions of Cylinder.

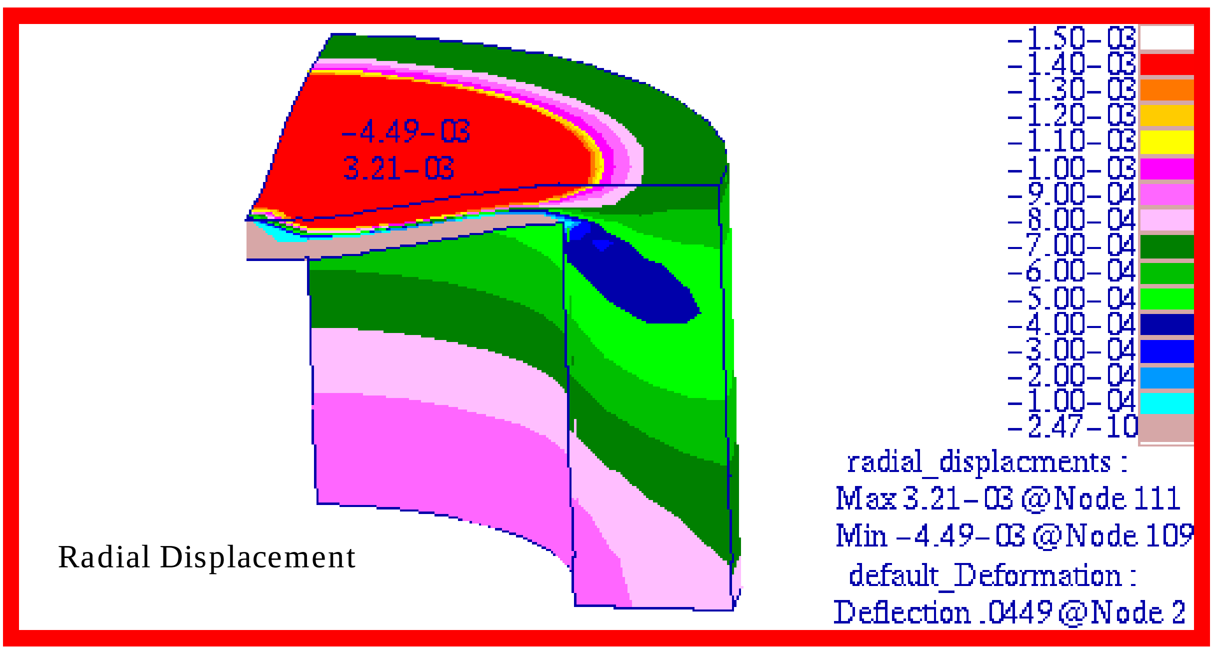


Figure 15-50 Radial Displacements of Thick Walled Cylinder.

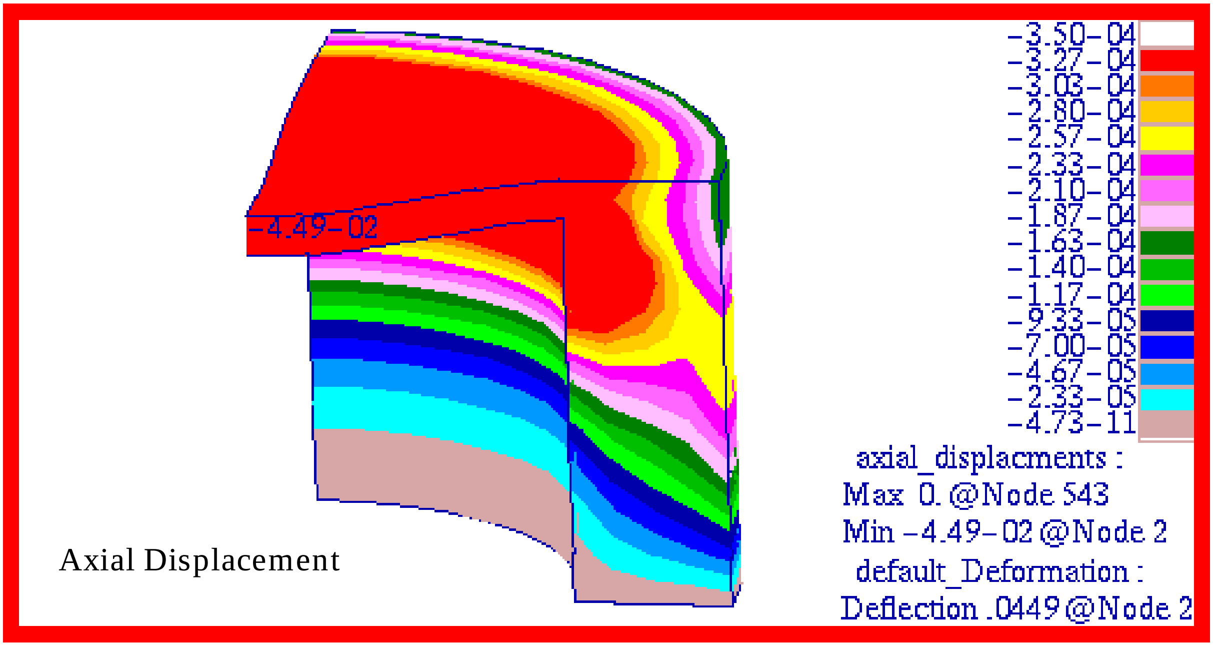


Figure 15-51 Axial Displacement of Thick Walled Cylinder.

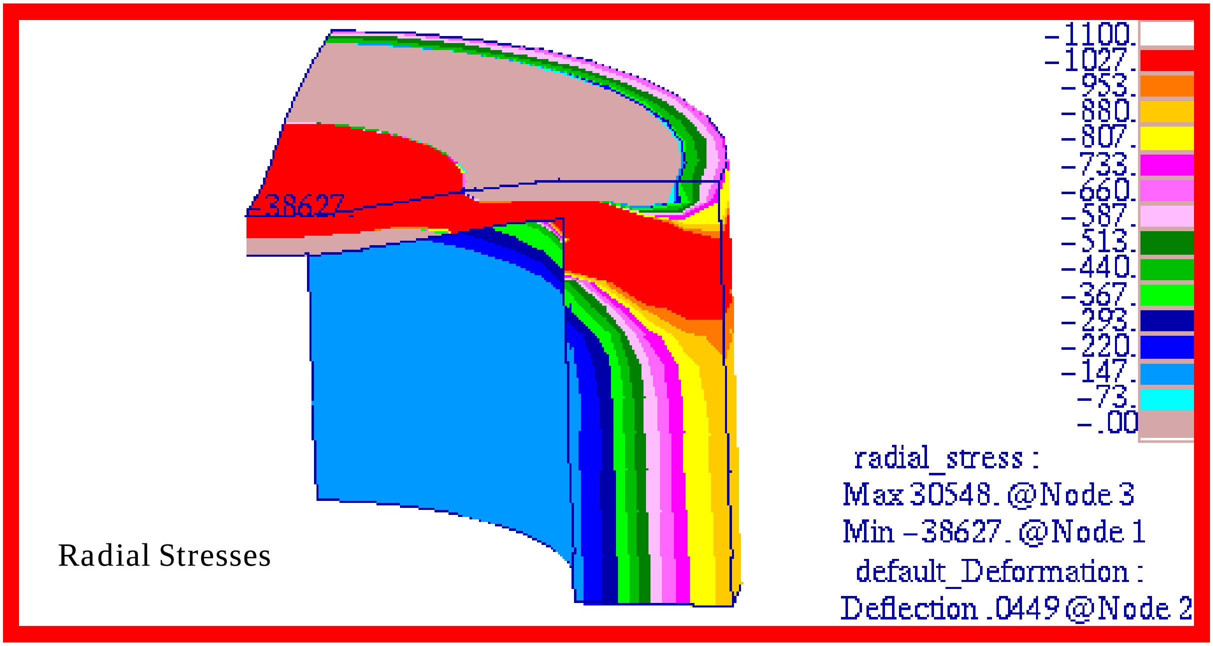


Figure 15-52 Radial, σ_3, Stress of Thick Walled Cylinder.

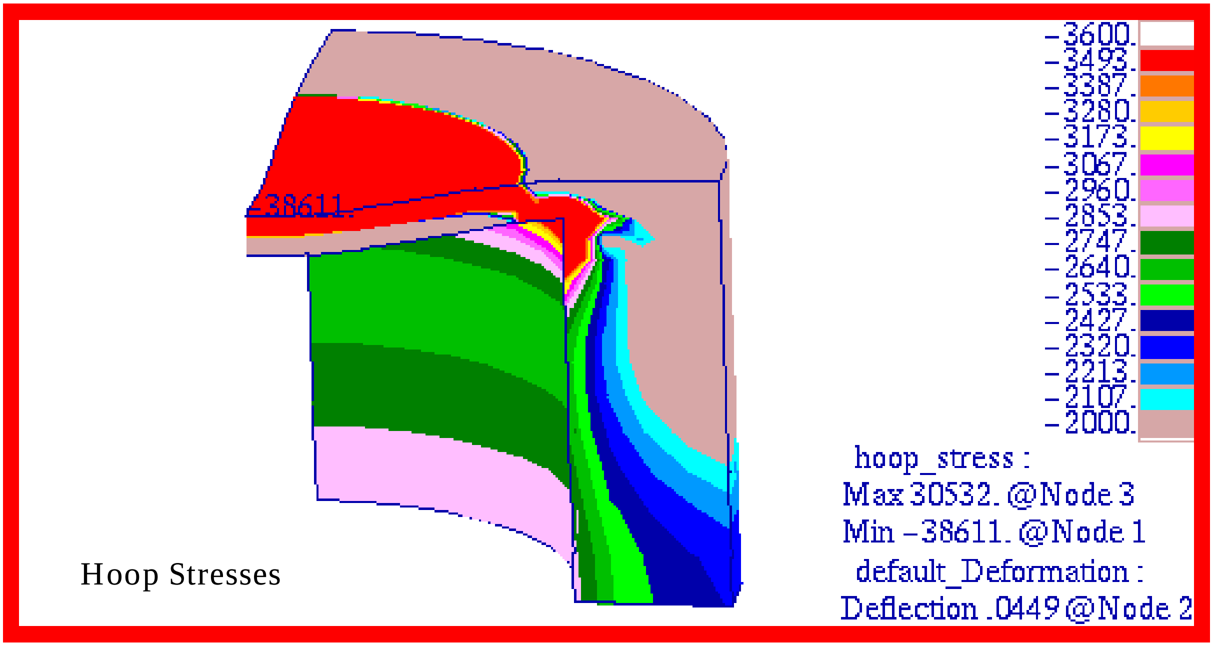


Figure 15-53 Hoop, σ_2, Stress of Thick Walled Cylinder.

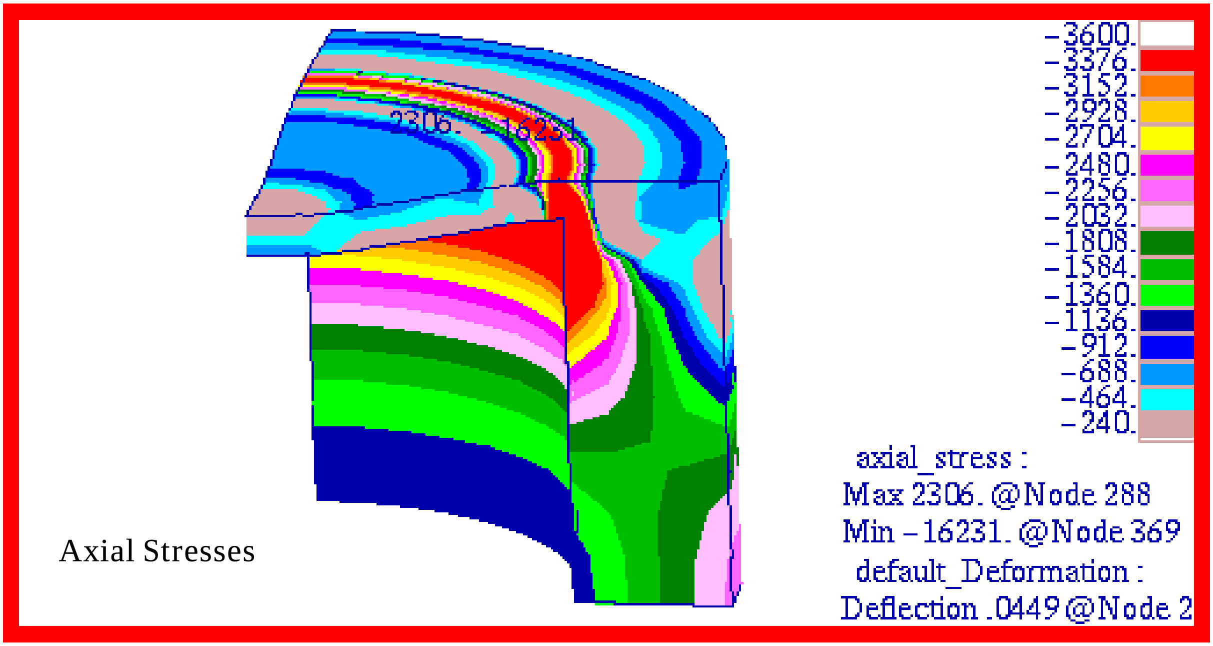


Figure 15-54 Axial, σ_1, Stress of Thick Walled Cylinder.

Problem 8: Linear Statics, Pinned Truss Analysis

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, CROD 1D Elements.

Reference:

Przemieniecki, J.S., *Theory of Matrix Structural Analysis*, McGraw-Hill, Inc., 1968, p. 155.

Problem Description:

A pinned joint truss is loaded with a force at one end and one of the components is heated uniformly to an elevated temperature. Considering thermal effects, find the displacements of the truss joints and the forces and stresses in each of the axial elements.

Engineering Data:

$E = 1.0 \times 10^7$

$\alpha = 1.0 \times 10^{-6} /$ Deg F

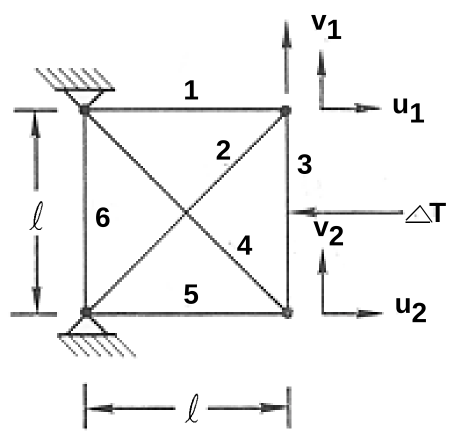
$A_1 = 1.0""\text{in}^2$(Elements 1,3,5 and 6)

$A_2 = 0.7071068""\text{in}^2$(Elements 2 and 4)

$\Delta T = 100.$Deg F (Elements 3 only)

$F = 1000.0$lb.

$\ell = 20.$in



Theoretical Solution:

The theoretical solution was calculated by using the matrix equation on page 159 of the reference using the values listed above for F, α, and ΔT. The results are as follows:

Displacements:

$u_1 = -1.272727 \times 10^{-3}$in $\quad u_2 = +7.272727 \times 10^{-4}$in

$v_1 = +6.363636 \times 10^{-3}$in $\quad v_2 = +3.636364 \times 10^{-3}$in

Element Forces:

$$P_i = \frac{EA}{\ell} \times u_i$$

P1 = -636.36 lb (compression)

P_2 = +899.95 lb (tension)

P_3 = +363.64 lb (tension)

P_4 = -514.25 lb (compression)

P_5 = +363.64 lb (tension)

P_6 = 0. lb

Element Stresses:

$$\sigma_1 = P_1 / A_1 = -636.36 \text{ lb/in}^2$$

$$\sigma_2 = P_2 / A_2 = +1272.73 \text{ lb/in}^2$$

$$\sigma_3 = P_3 / A_1 = +363.64 \text{ lb/in}^2$$

$$\sigma_4 = P_4 / A_2 = -727.27 \text{ lb/in}^2$$

$$\sigma_5 = P_5 / A_5 = +363.64 \text{ lb/in}^2$$

$$\sigma_6 = P_6 / A_6 = 0. \text{ lb/in}^2$$

MSC.Nastran Results:

Due to the pinned construction of the truss, all members are only capable of carrying axial loads. This necessitated that the truss be modeled using MSC.Nastran 1D CROD elements. The model that was created using MSC.Patran is shown in **Figure 15-55**. Note that each truss member was modeled using a single CROD element. The loads and boundary conditions that were applied to the model are also shown in **Figure 15-55**.

The following results were obtained with MSC.Nastran.

Table 15-27 Displacements, (inches)

	u_1	v_1	u_2	v_2
Theory	$-1.272727\text{x}10^{-3}$	$+6.363636\text{x}10^{-3}$	$+7.272727\text{x}10^{-4}$	$+3.636364\text{x}10^{-3}$
MSC.Nastran	$-1.272727\text{x}10^{-3}$	$+6.363636\text{x}10^{-3}$	$+7.272727\text{x}10^{-4}$	$+3.636364\text{x}10^{-3}$
% Difference	0.0%	0.0%	0.0%	0.0%

Table 15-28 Rod Forces (lbs)

	P_1	P_2	P_3	P_4	P_5	P_6
Theory	-636.36	+899.95	+363.64	-514.26	+363.64	0.
MSC.Nastran	-636.36	+899.95	+363.64	-514.26	+363.64	0.
% Difference	0.0%	0.0%	0.0%	0.0%	0.0%	0.0%

Table 15-29 Rod Stresses (psi)

	σ_1	σ_2	σ_3	σ_4	σ_5	σ_6
Theory	-636.36	+1272.73	+363.64	-727.27	+363.64	0.
MSC.Nastran	-636.36	+1272.73	+363.64	-727.27	+363.64	0.
% Difference	0.0%	0.0%	0.0%	0.0%	0.0%	0.0%

The corresponding color fringe plots that were made in MSC.Patran of the x- and y-components of displacement are shown in **Figure 15-56** and **Figure 15-57**, respectively. For the purposes of these plots, the target entity was set to nodes and the averaging domain was specified as all entities. The rod forces and stresses that were predicted by MSC.Nastran are shown in **Figure 15-58** and **Figure 15-59**, respectively. Unlike displacements, it was necessary to set the target entity to elements and the averaging domain to individual. This avoided having any averaging across adjacent elements which otherwise would have given an inaccurate representation of the actual results. A closer inspection of these plots reveals that the results that are being shown are in fact identical to those predicted by MSC.Nastran.

File(s): <install_dir>/results_vv_files/prob008.bdf, prob008.op2

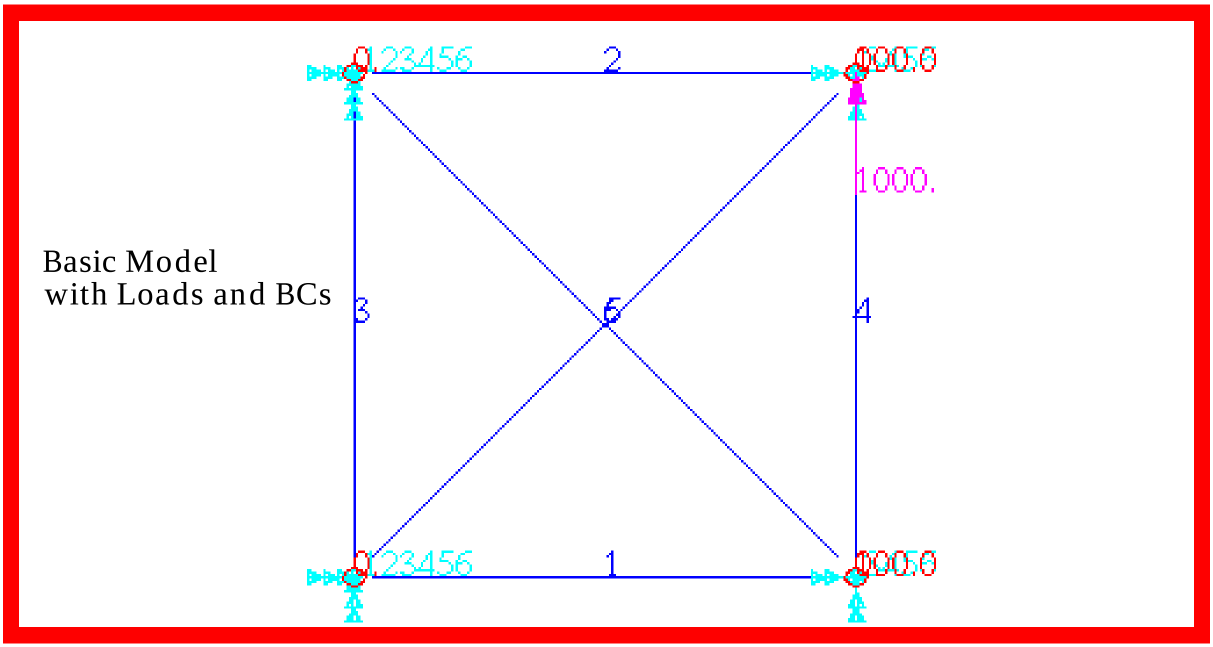


Figure 15-55 Basic Pinned Truss Analysis Model.

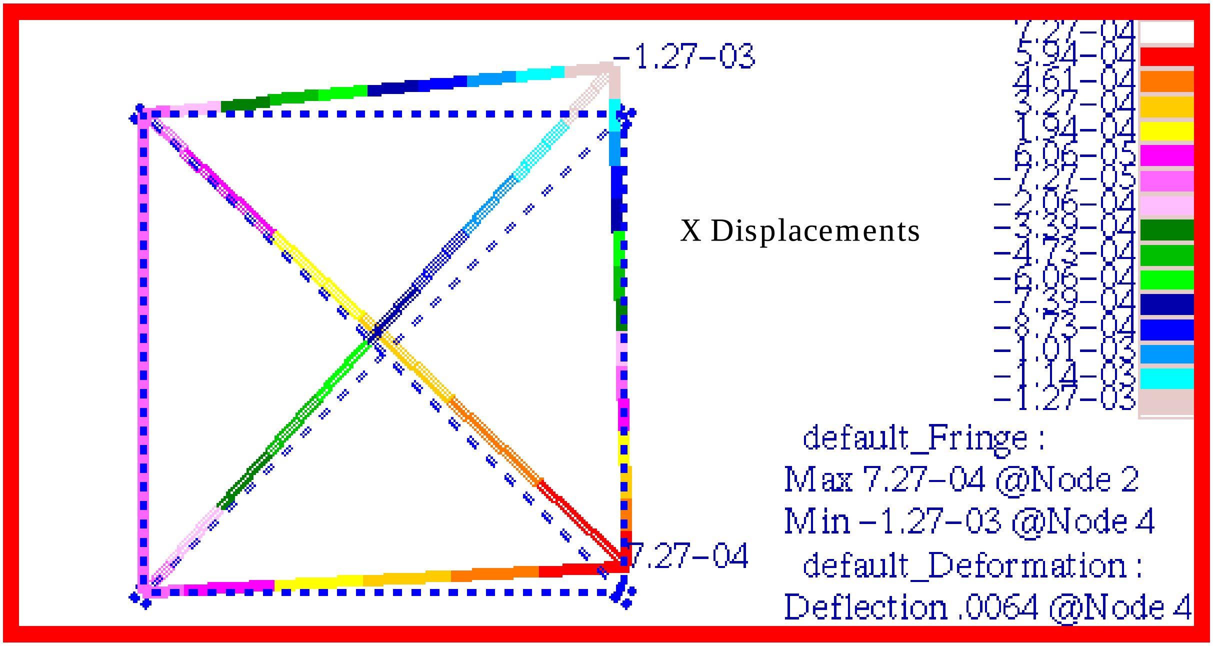


Figure 15-56 X Translational Displacement of Pinned Truss.

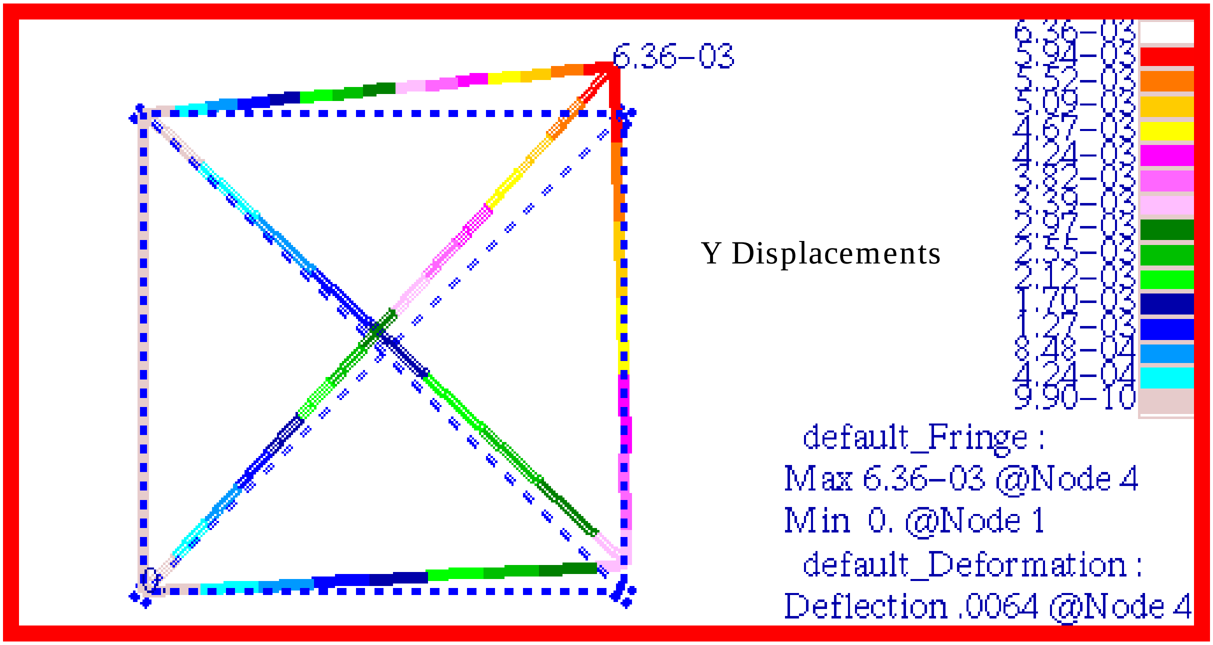


Figure 15-57 Y Translational Displacement of Pinned Truss.

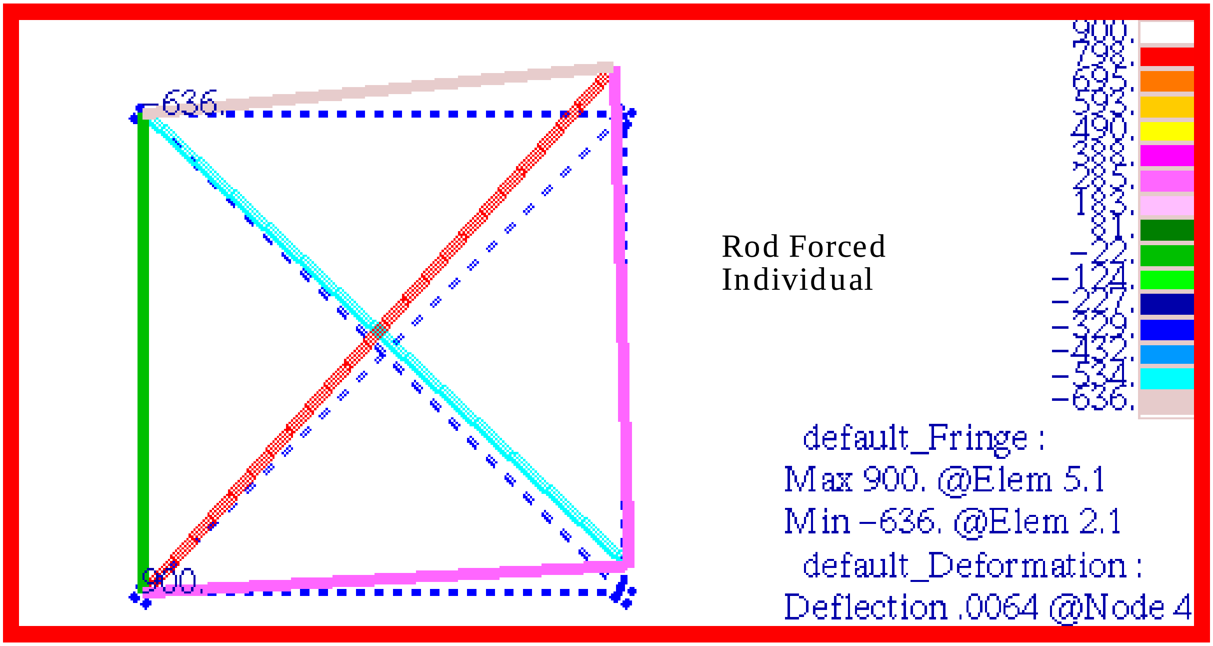


Figure 15-58 Rod Forces from Pinned Truss Analysis.

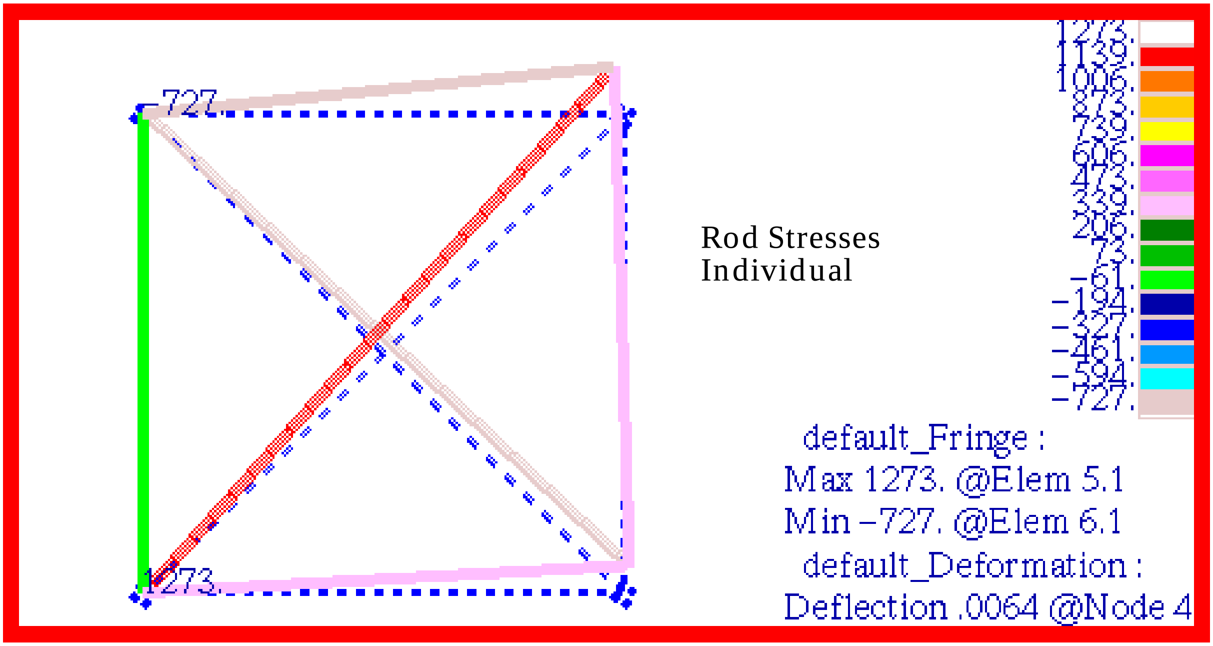


Figure 15-59 Axial Stresses from Pinned Truss Analysis.

Problem 9: Nonlinear Statics, Large Deflection Effects

Solution/Element Type:

MSC.Nastran, Nonlinear Statics, Solution 106, Shell Element with Standard Formulation, CQUAD4 and CQUAD8

Reference:

Timoshenko, S., and Woinowsky-Krieger, S., *Theory of Plates and Shells*, McGraw-Hill, Inc., 1959, p. 422.

Problem Description:

A square plate with clamped edges is loaded uniformly such that the center deflection exceeds the plate thickness. Considering large deflection effects (small strain theory), find the deflection at the center of the plate.

$a = b = 100.0$

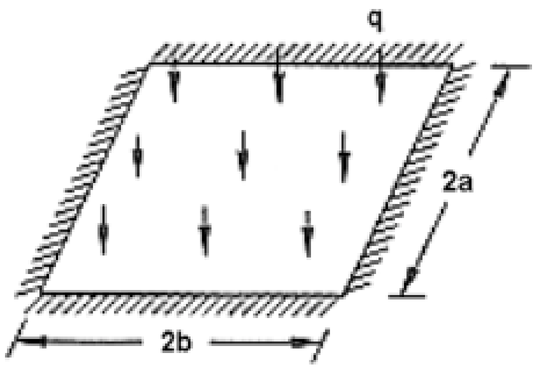
$h = 1.0$

$E = 2.0 \times 10^{11}$

$v = 0.3$

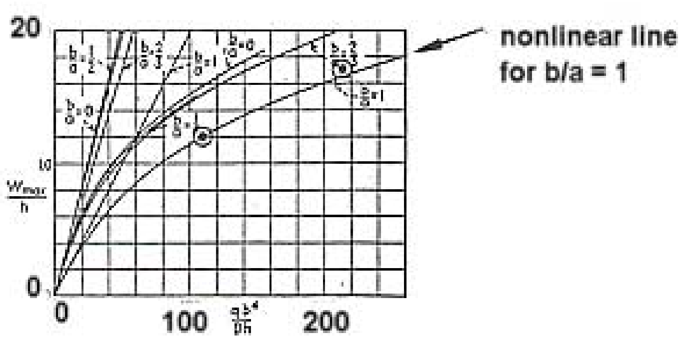
$q = 2.0 \times 10^4$

$$d = \frac{Eh^3}{12(1 - v^2)} = 1.83 \times 10^{10}$$



Theoretical Solution:

The following diagram summarizes the theoretical solution for the case when large deflection effects are both considered and ignored.



For $q \cdot b^4 / (Dh) = 109.3$, $w_{max} / h = 1.20$, with nonlinear effects.

For $q \cdot b^4 / (Dh) = 109.3$, $w_{max} / h = 2.208$, without nonlinear effects.

MSC.Nastran Results:

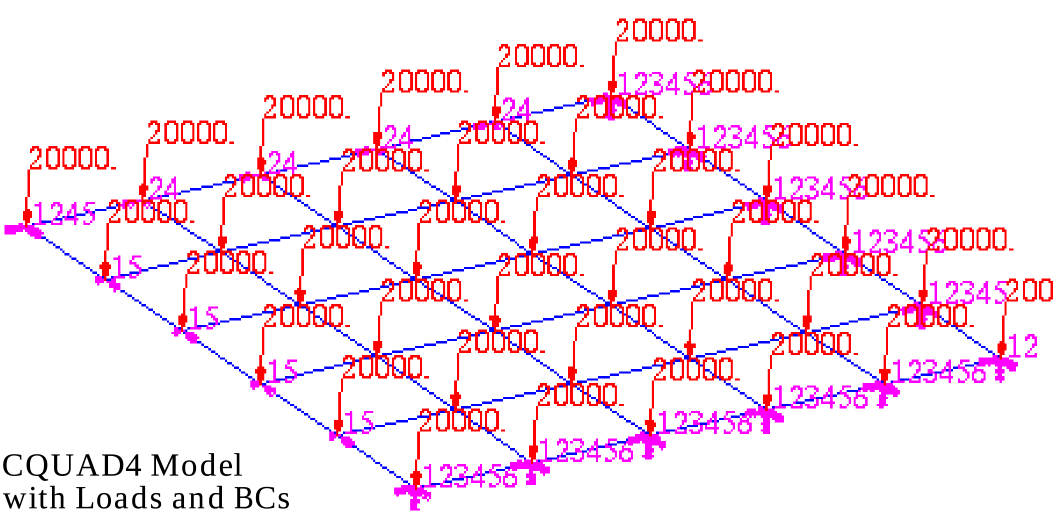
Due to symmetry, only one-fourth of the plate was modeled. The outer edges were fully restrained while the appropriate symmetry boundary conditions were imposed along the remaining free edges and at the center of the plate. Two models were created. The first was meshed with MSC.Nastran CQUAD4 elements which have a finite strain formulation needed to incorporate large deflection effects. The second model was meshed with CQUAD8 elements, which do not have a finite strain formulation. Both models along with the applied loading and imposed boundary conditions are shown in **Figure 15-60** and **Figure 15-61**. The MSC.Nastran results that were obtained are summarized below and plotted in **Figure 15-62** and **Figure 15-63**.

Table 15-30 Large Deflection Results

Element Type	w_{max}/h, (Theory)	w_{max}/h, (MSC.Nastran)	% Difference
QUAD4	1.20	1.26	5.0%
QUAD8	2.208	2.21	.091%

It should be noted that although both analyses were conducted using MSC.Nastran, Solution 106 with large deflection effects included, the analysis performed with CQUAD8 excluded any geometric nonlinearities since this type of element has no finite strain formulation. This illustrates the care that must be exercised when performing a nonlinear statics analysis with mixed element types. The same problem exists for CTRIA3 and CTRIA6 elements.

File(s): `<install_dir>/results_vv_files/prob009_Q4.bdf, prob009_Q4.op2, prob009_Q8.bdf, prob009_Q9.op2, prob009_T3.bdf, prob009_T3.op2, prob009_T6.bdf, prob009_T6.op2`



CQUAD4 Model
with Loads and BCs

Figure 15-60 CQUAD4 Model of Plate with Loads and BCs.

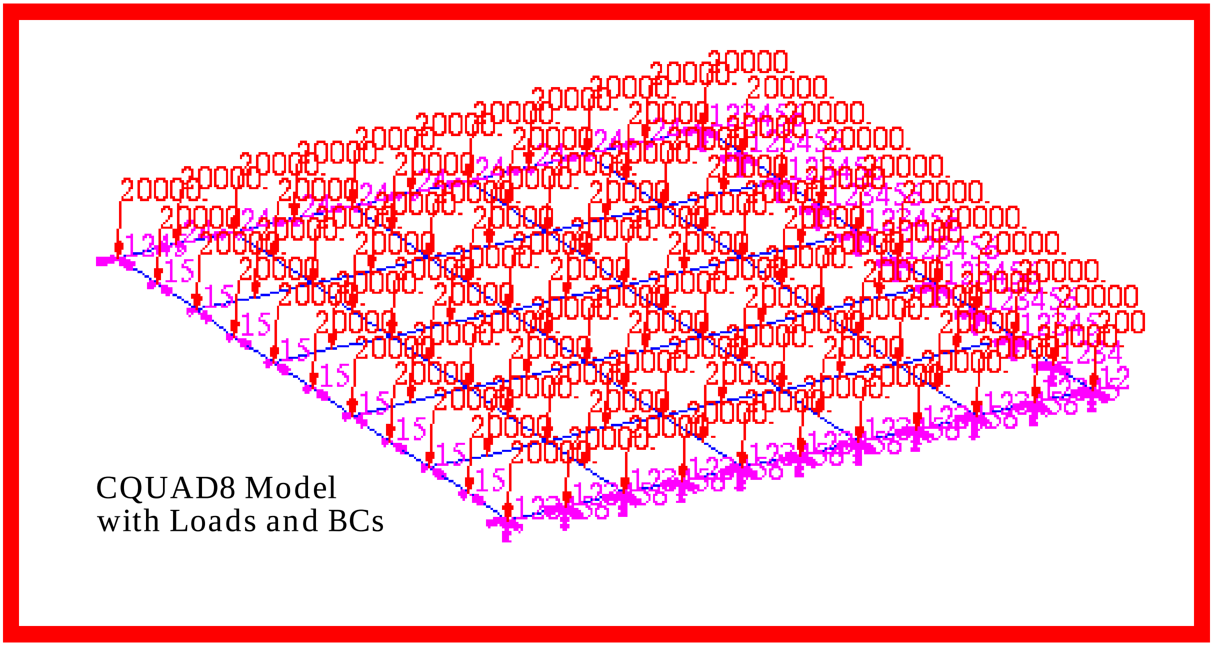


Figure 15-61 CQUAD4 Model of Plate with Loads and BCs.

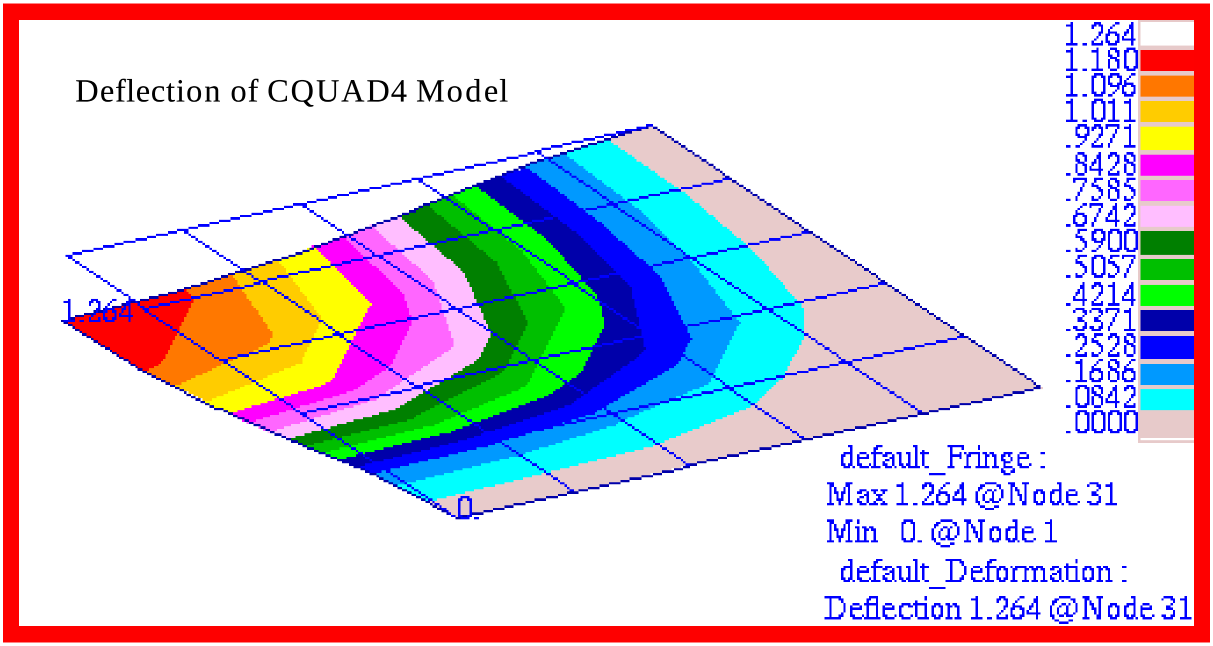


Figure 15-62 Large Deflection Displacements of CQUAD4 Model.

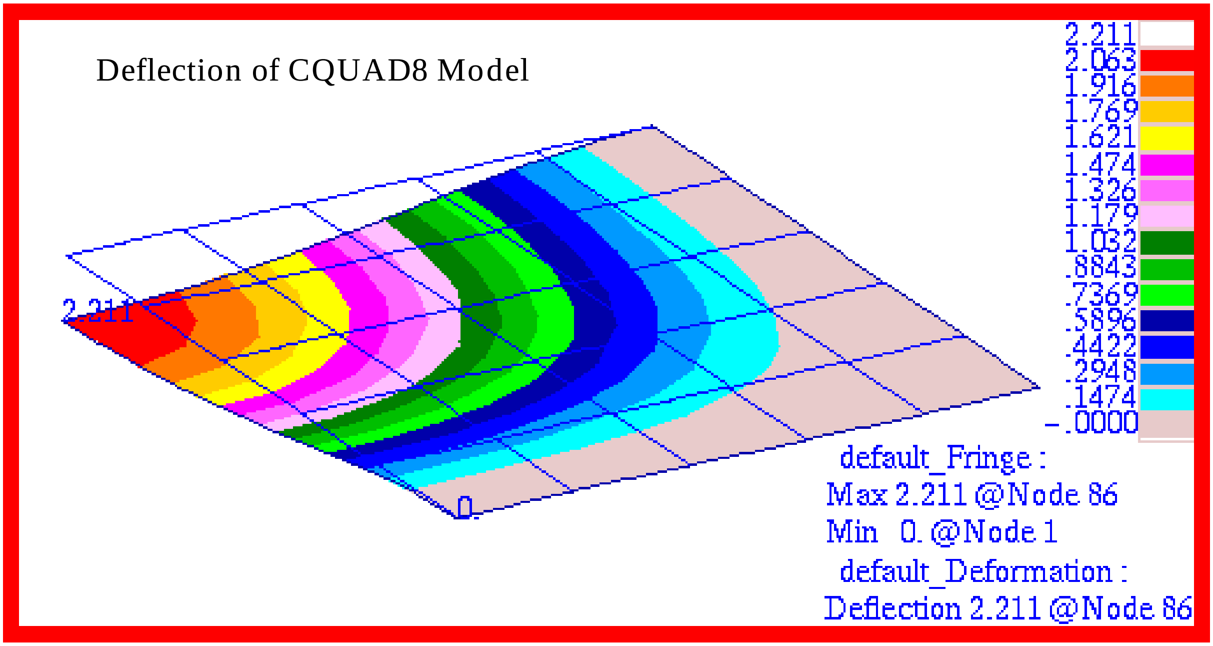


Figure 15-63 Large Deflection Displacements of CQUAD8 Model.

Problem 10: Linear Statics, Thermal Stress with Solids

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, Solid Elements With Standard Formulation, CHEXA (8 and 20 noded), CPENTA (6 and 15 noded), CTETRA (4 and 10 noded).

Reference:

Timoshenko, S., and Goodier, J. N., *Theory of Elasticity*, 2nd ed., McGraw-Hill, Inc., 1951, pp. 401-403.

Problem Description:

Six cubes are subjected to three independent linear temperature gradients in the x-, y- and z- directions. The cubes are meshed with either HEX8, HEX20, WEDGE6, WEDGE15, TET4 or TET10 elements. Each cube is mounted on a weak elastic foundation so that the resultant thermal expansion is unrestrained. Compute the Von Mises stress in each of the cubes.

Theoretical Solution:

In general, when the applied temperature is a linear function of x, y and z, the strain induced by free thermal expansion is:

$$\varepsilon_x = \varepsilon_y = \varepsilon_z = \alpha T \qquad \gamma_x = \gamma_y = \gamma_z = 0 \qquad \text{(a)}$$

The corresponding stress-strain relationships for three dimensional problems are:

$$\varepsilon_x - \alpha T = \frac{1}{E}[\sigma_x - v(\sigma_y + \sigma_z)]$$

$$\varepsilon_y - \alpha T = \frac{1}{E}[\sigma_y - v(\sigma_x + \sigma_z)] \qquad \text{(b)}$$

$$\varepsilon_z - \alpha T = \frac{1}{E}[\sigma_z - v(\sigma_x + \sigma_y)]$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}, \qquad \gamma_{yz} = \frac{\tau_{yz}}{G}, \qquad \gamma_{zx} = \frac{\tau_{zx}}{G}, \qquad \text{(c)}$$

Equations (c) are not affected by temperature since free thermal expansion does not produce angular distortion in an isotropic material. The remaining stress-strain relationships given by equations (b) can only be satisfied after substitution of the strains due to free thermal expansion if

$$\sigma_x = \sigma_y = \sigma_z = 0$$

Thus, a linear temperature gradient should produce no apparent stress, provided the expansion is completely unrestrained.

MSC.Nastran Results:

To demonstrate that thermal stress effects can be properly recovered for all of the various solid element topologies supported by MSC.Nastran, a simple model was created that consisted of 6 cubes. Each cube was subjected to three independent linear temperature gradients of 100, 200 and 300 degrees F per inch along the x, y and z axes, respectively. The cubes were meshed with either HEX8, HEX20, WEDGE6, WEDGE15 TET4 or TET10 elements with a mesh density of one element per edge. The model that was created is shown in **Figure 15-64** and the applied temperatures are shown in **Figure 15-65**.

Due to the presence of three simultaneous temperature gradients, the faces of the cubes will not remain planar. This precludes applying uniform constraints along any face of the cubes in order to prevent any rigid body translation from occurring. Instead every vertex of a cube was attached to a triad of three weak grounded springs that were aligned in either the x, y or z-directions. Each spring had a spring rate of 0.1 lbs / inch, which would provide minimal resistance to thermal expansion. This should generate only a minimal amount of residual stress in each of the cubes.

The maximum von Mises stress that was computed by MSC.Nastran for a given element topology are summarized below.

Table 15-31 Maximum von Mises Stress

Element Type	Max Von Mises Stress (psi)
HEX8	.001512
HEX20	.007403
WEDGE6	49811
WEDGE15	.007900
TET4	7610
TET10	.005904

The corresponding deformed shapes and von Mises stress contours that were generated using MSC.Patran are shown in **Figure 15-66** and **Figure 15-67**, clearly showing the high residual stresses that were generated in both the degenerate WEDGE6 and TET4 elements. The peak values for the von Mises stress shown are somewhat less than those shown in due to the nodal averaging of results at adjacent elements.

Ideally, the presence of a weak elastic support should have provided minimal restraint to thermal expansion, resulting in a near zero stress state. This behavior was observed in both the HEX8 and HEX20 elements as well as the higher order TET10 and WEDGE15 elements. The extremely high stresses observed with TET4 and WEDGE6 elements would not be unexpected since these elements tend to be excessively stiff resulting in a loss of accuracy. Consequently, the results illustrate the care that must be exercised when modeling with the solids and the need to avoid excessive use of degenerate elements, especially in areas where high stress gradients are predicted to occur, such as at the vertices of the cubes in this particular problem.

File(s): `<install_dir>/results_vv_files/prob010.bdf, prob010.op2`

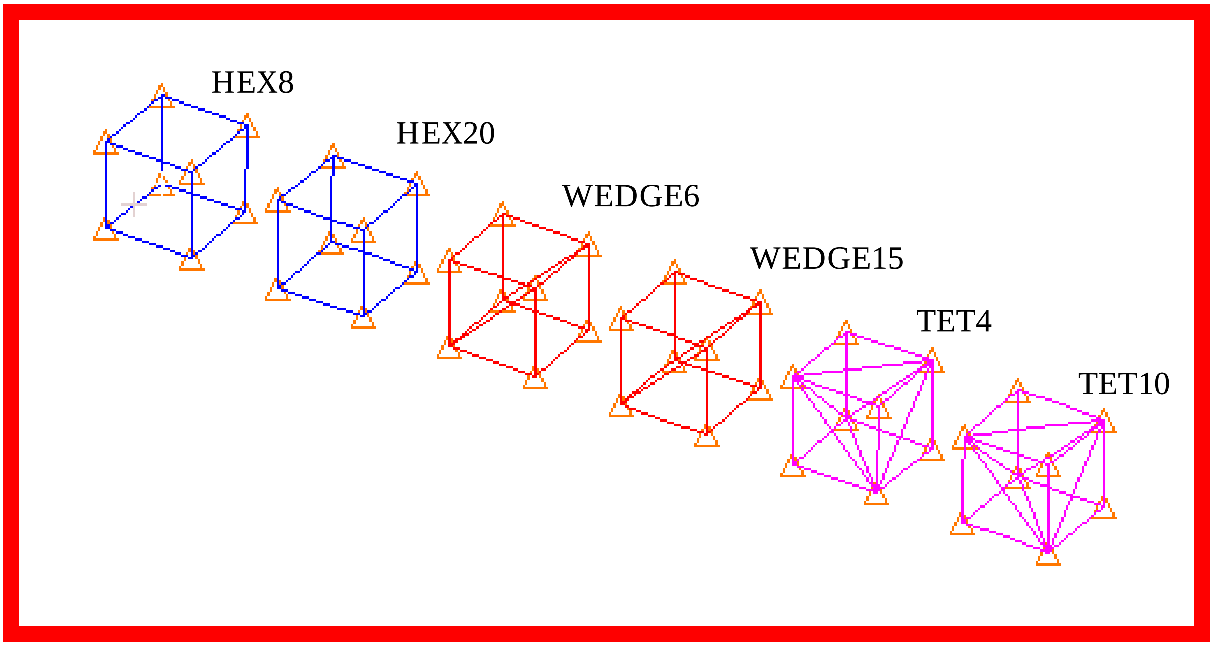


Figure 15-64 Models of Different Solid Element Types.

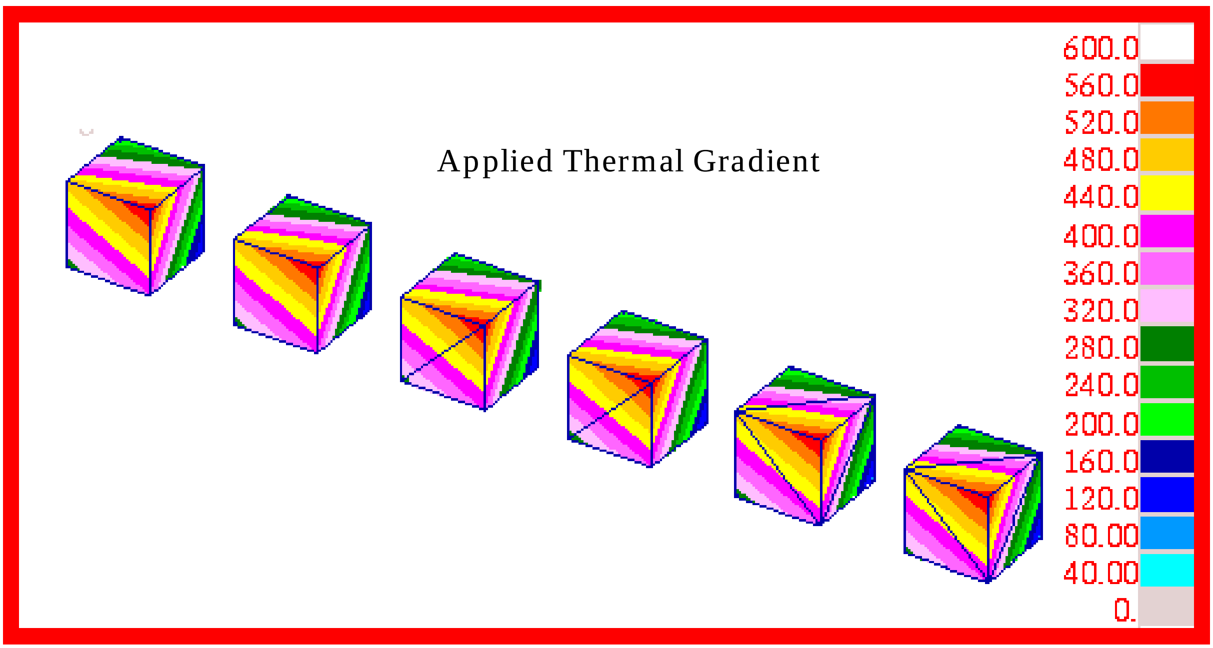


Figure 15-65 Thermal Gradient Applied to Solid Elements.

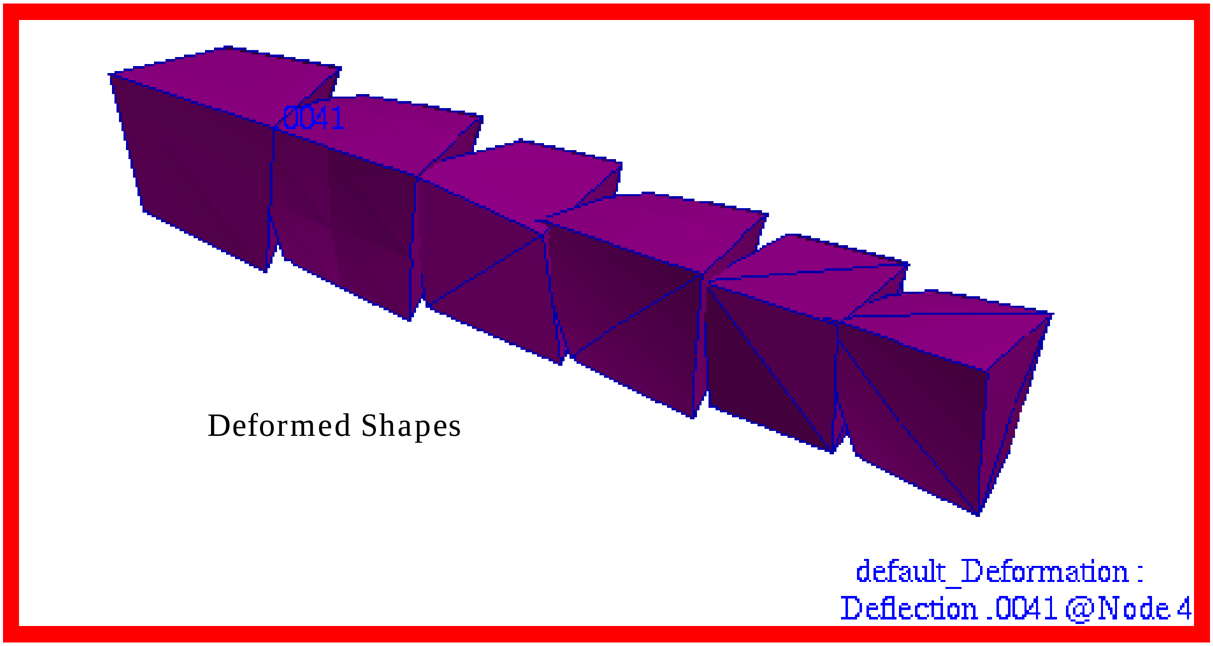


Figure 15-66 Deformed Shape of Elements Due to Thermal Gradient.

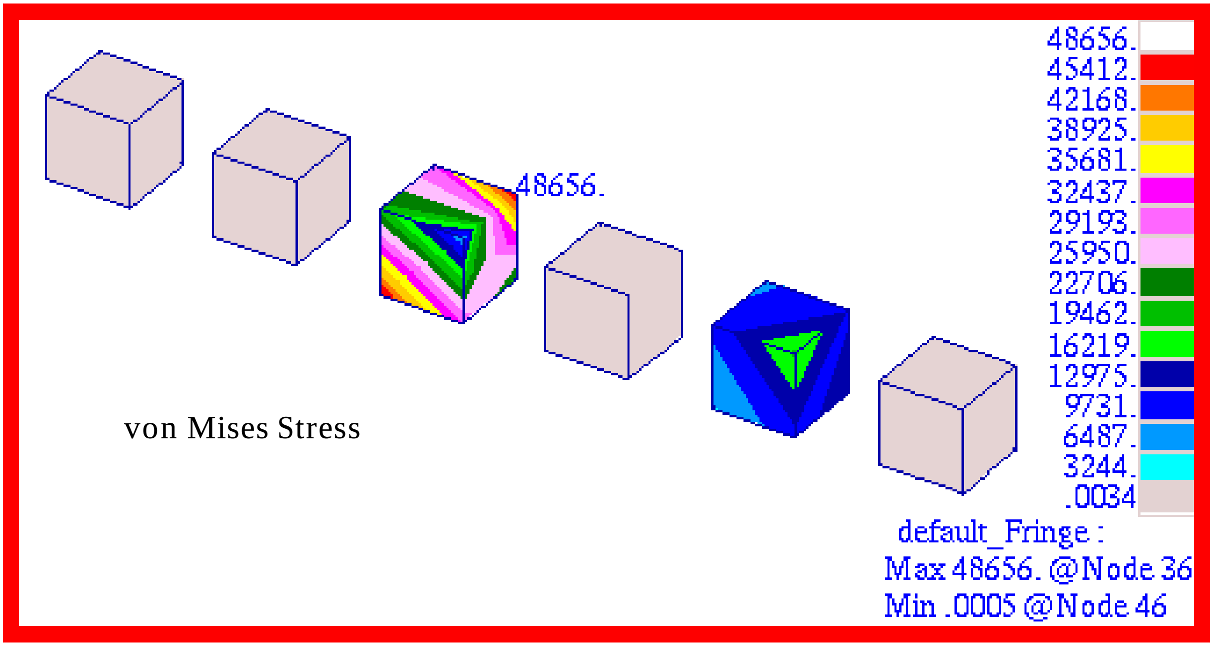


Figure 15-67 von Mises Stress in Elements due to Thermal Gradient.

Problem 11: Superposition of Linear Static Results

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, Simple Shell Elements, CQUAD4 and CTRIA3

Reference:

Ugural, A.C., *Stresses in Plates and Shells,* McGraw-Hill, Inc., 1981, pp. 41-42.

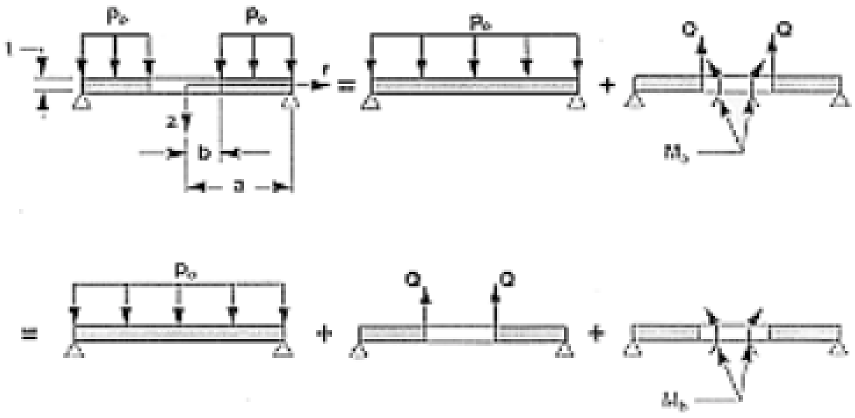
Roark, R.J., and Young, W.C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Inc., 1975, p. 338.

Problem Description:

A flat annular plate is subjected to a uniform pressure. The inner edge is free and the outer edge simply supported. Determine the maximum deflection of the plate.

Theoretical Solution:

The resultant deformation for a simply supported annular plate can be derived by summing the deformations obtained for a flat circular plate subjected to the same pressure load, an annular plate carrying along its inner edge a shear force per unit circumferential length of $p_o b/2$ and an annular plate with a distributed radial bending moment of $p_o(3+v)(a^2-b^2)/16$. All of the plates have the same outer radius and the both annular plates have the same inner radius.



Subcase A Subcase B Subcase C

According to reference (2), the maximum deflection occurs at the inner radius and is given by the expression:

$$w_{max} = \frac{-p_o a^4}{D}\left(\frac{C_1 L_{17}}{C_7} - L_{11}\right) \qquad \text{Eq. 15-1}$$

where

$$C_1 = \frac{1+v}{2}\ln\frac{b}{a} + \frac{1-v}{4}\left(\frac{a}{b} - \frac{b}{a}\right) \qquad \text{Eq. 15-2}$$

$$L_{17} = \frac{1}{4}\left\{1 - \frac{1-v}{4}\left[1 - \left(\frac{b}{a}\right)^4\right] - \left(\frac{b}{a}\right)^2\left[1 + (1+v)\ln\frac{a}{b}\right]\right\} \qquad \text{Eq. 15-3}$$

$$C_7 = \frac{1}{2}(1-v^2)\left(\frac{a}{b} - \frac{b}{a}\right) \qquad \text{Eq. 15-4}$$

$$L_{11} = \frac{1}{64}\left\{1 + 4\left(\frac{b}{a}\right)^2 - 5\left(\frac{b}{a}\right)^4 - 4\left(\frac{b}{a}\right)^2\left[2 + \left(\frac{b}{a}\right)^2\right]\ln\frac{a}{b}\right\} \qquad \text{Eq. 15-5}$$

$$D = \frac{Et^3}{12(1-v^2)} \qquad \text{Eq. 15-6}$$

For the purposes of this problem, models of a flat circular and annular plates where created in MSC.Patran using the following dimensions, applied loading and material properties:

$a = 8.0$ in	$E = 1.0 \times 10^7$ psi
$b = 4.0$ in	$v = 0.33$
$t = 0.25$ in	$p_o = 50.0$ psi

MSC.Nastran Results:

The model that was generated for a uniformly pressurized circular plate is shown in **Figure 15-68**. Due to symmetry, only a 90 degree sector of the plate was modeled with axisymmetric boundary conditions being applied along the lateral edges of the sector. The center of the plate was constrained to move only in the vertical direction. Between the inner and outer radius of the annular plate, a mesh consisting of CQUAD4 elements was used to maximize accuracy. Conversely, inboard of the inner radius, a CTRIA3 mesh was used to avoid having badly distorted CQUAD4 elements that might otherwise result in a loss of accuracy. By having the mesh transition coincide exactly with the edges of the annular plate, this permitted using a single model to investigate the behavior of a complete circular plate as well as an annular plate, with the elements corresponding to just the annular and the complete plate being assigned to a different groups within MSC.Patran. The corresponding models that were used to investigate the case of an annular plate subjected to either a distributed edge shear or moment are shown in **Figure 15-69** and **Figure 15-70**.

Due to axisymmetric response of the plate, all deformations are principally confined to the vertical, or z-direction. The resultant vertical displacements as well as the hoop and radial stresses that were computed for each of the three different subcases are shown in **Figure 15-71** through **Figure 15-79**. The displacement and stress contours have been superimposed upon an exaggerated deformed shape. The resultant displacements and stresses that were derived by combining the results from each of the separate subcases are shown in **Figure 15-80** though **Figure 15-82**. An examination of **Figure 15-80** indicates that a maximum deformation of -0.8799 inches occurs at the inner radius of a flat annular plate subjected to a uniform pressure of 50 psi. Note that for the combined stress plots, the averaging was set to Target Entities to remove the influence of the stresses from the CTRIA3 elements.

Substitution of the model parameters into equations Eq. 15-1 through Eq. 15-6 yields a maximum deflection at the inner radius of

$$w_{max} = -0.8839 \text{ in}$$

The value derived within MSC.Patran was -.8799 in, or a deviation of approximately 0.69% from the theoretical value

File(s):`<install_dir>/results_vv_files/prob011_A.bdf, prob011_A.op2, prob011_B.bdf, prob011_B.op2, prob011_C.bdf, prob011_C.op2`

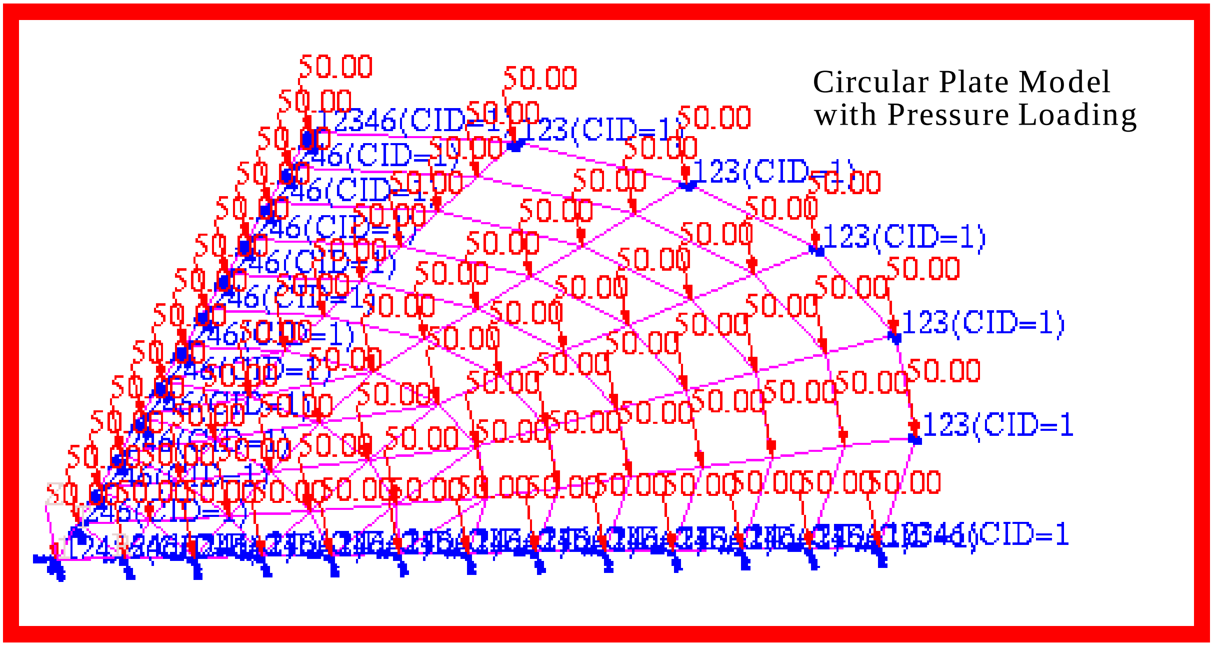


Figure 15-68 Model of Pressurized Circular Plate (Subcase A).

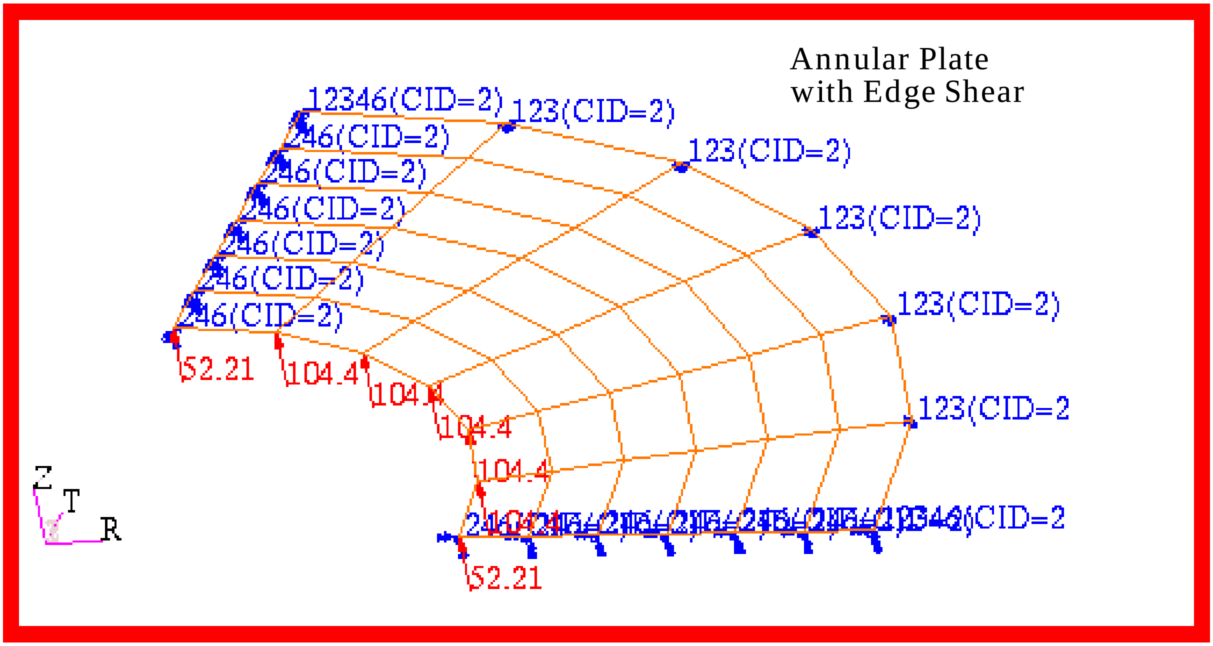


Figure 15-69 Annular Plate with Edge Shear (Subcase B).

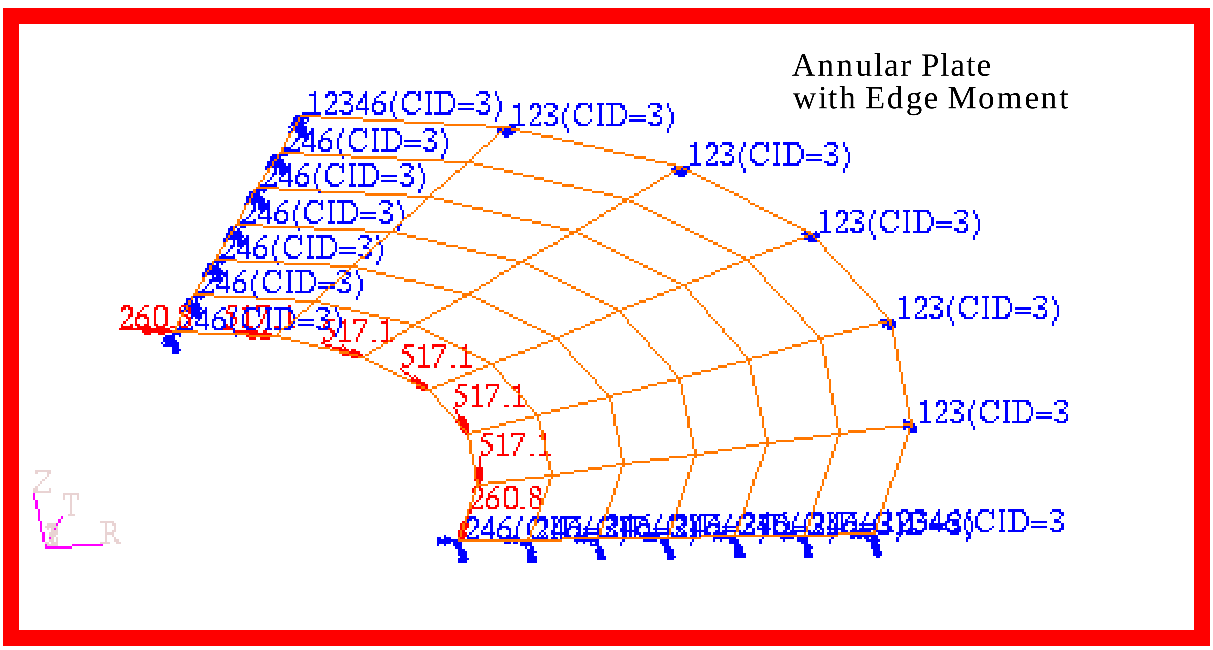


Figure 15-70 Annular Plate with Edge Moment (Subcase C).

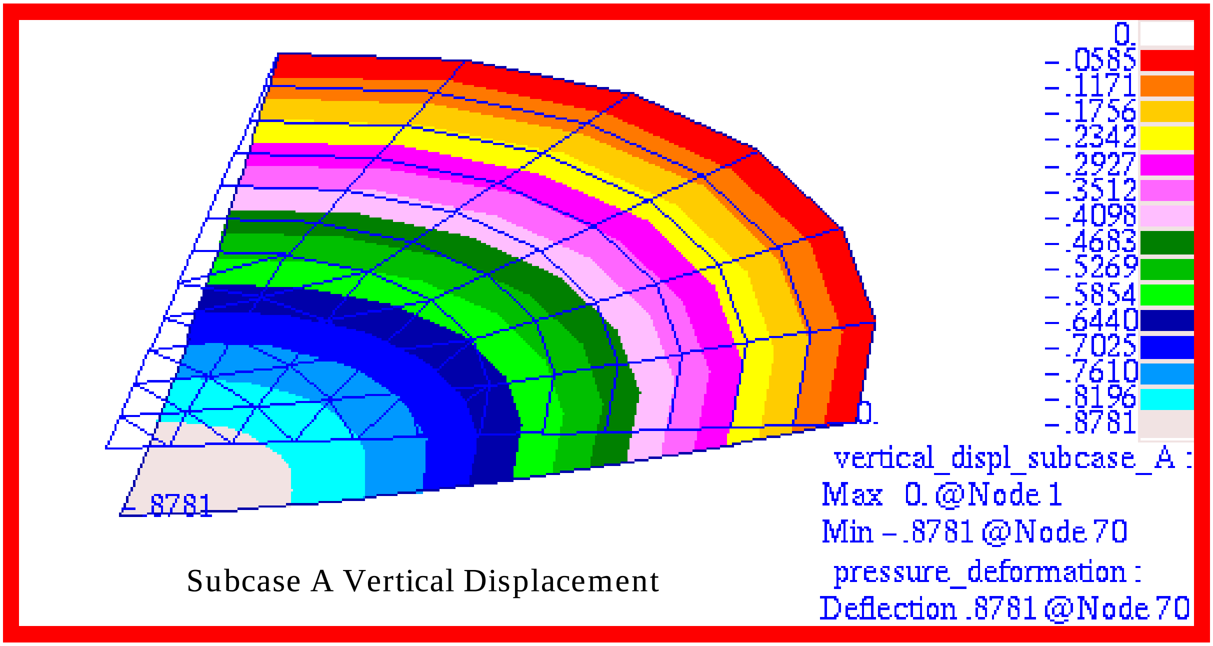


Figure 15-71 Vertical Displacement of Circular Plate (Subcase A).

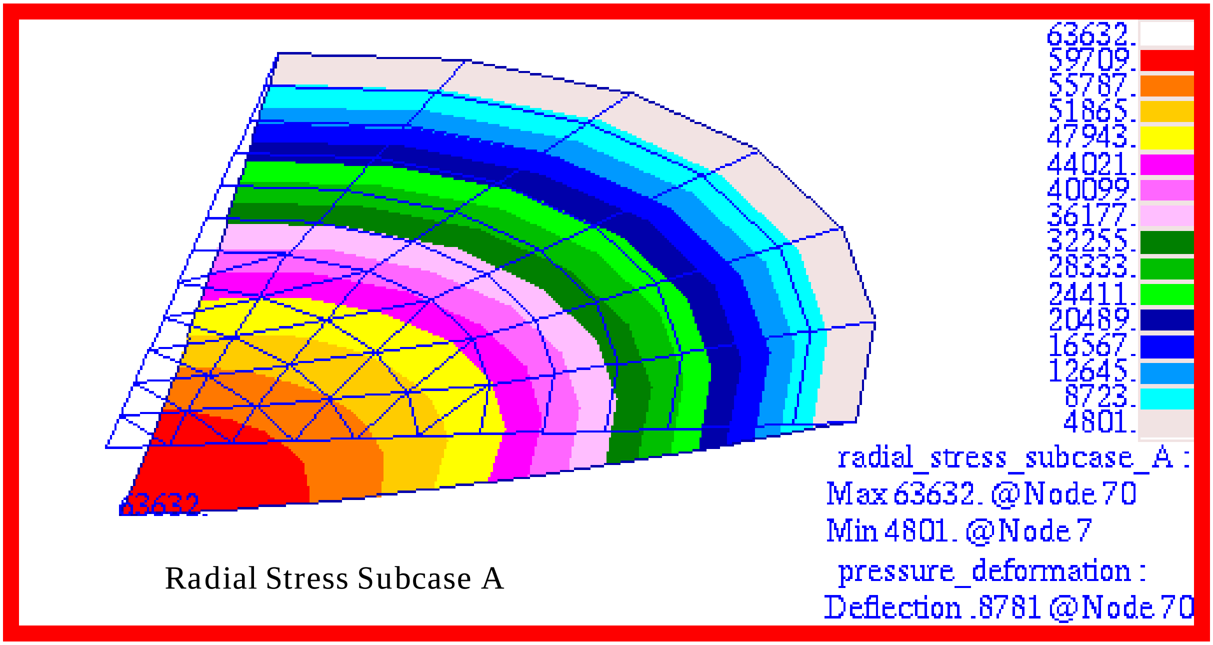


Figure 15-72 Radial Stress of Circular Plate (Subcase A).

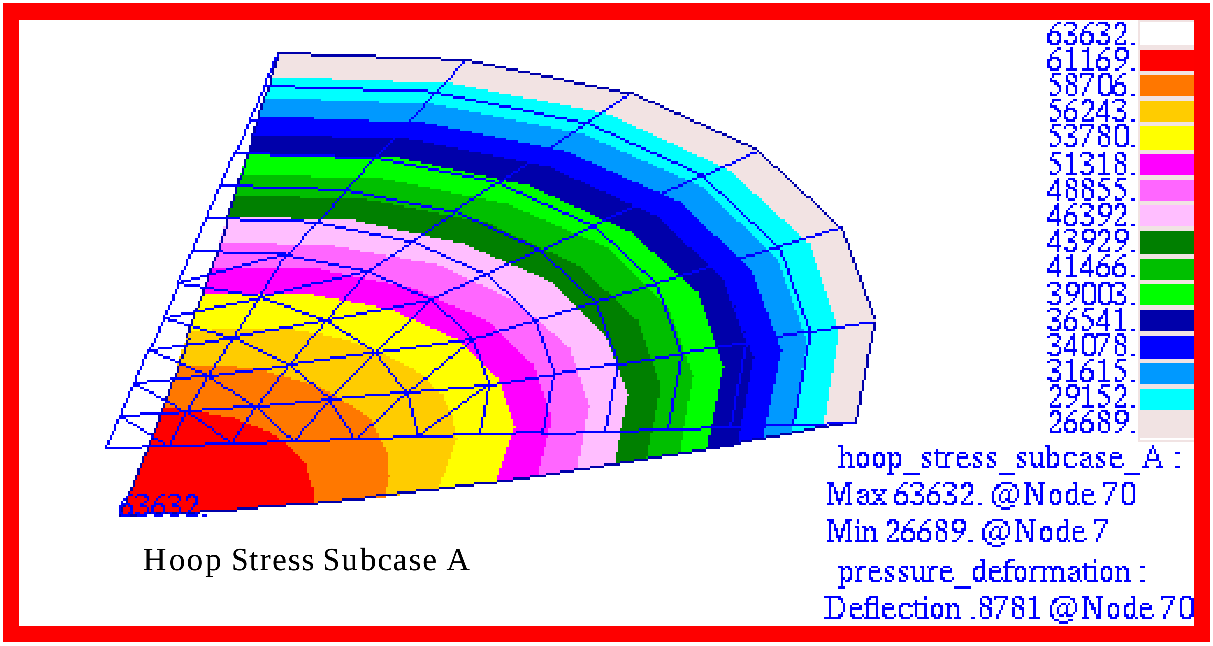


Figure 15-73 Hoop Stress of Circular Plate (Subcase A).

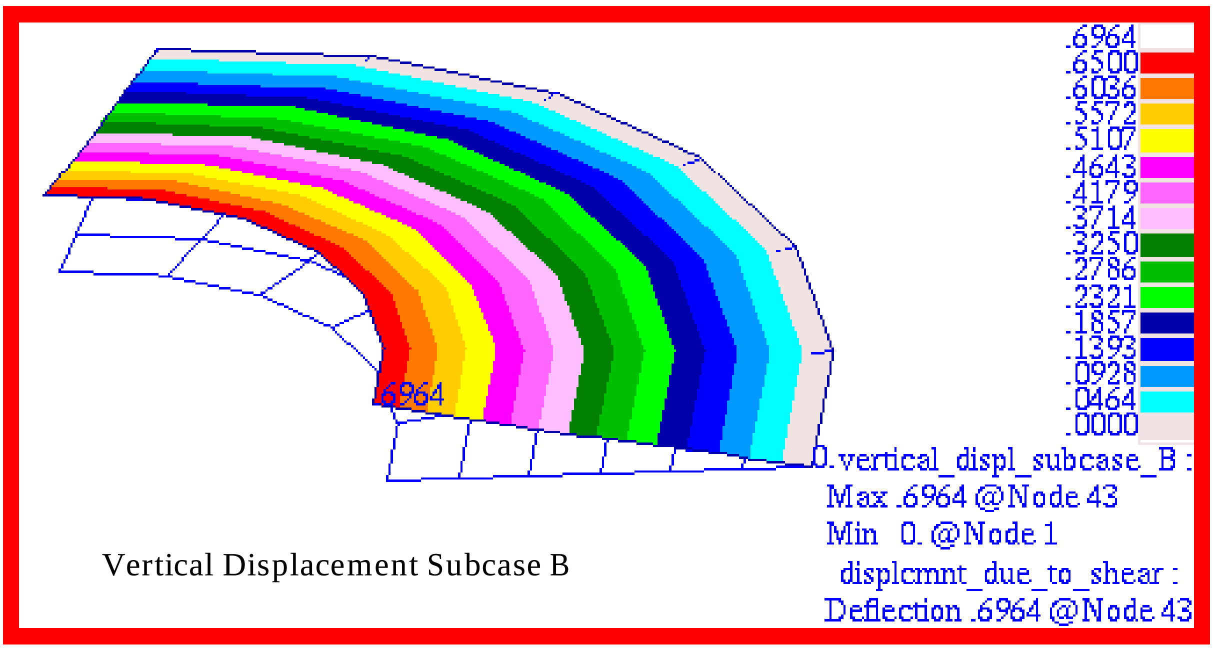


Figure 15-74 Vertical Displacement of Annular Plate (Subcase B).

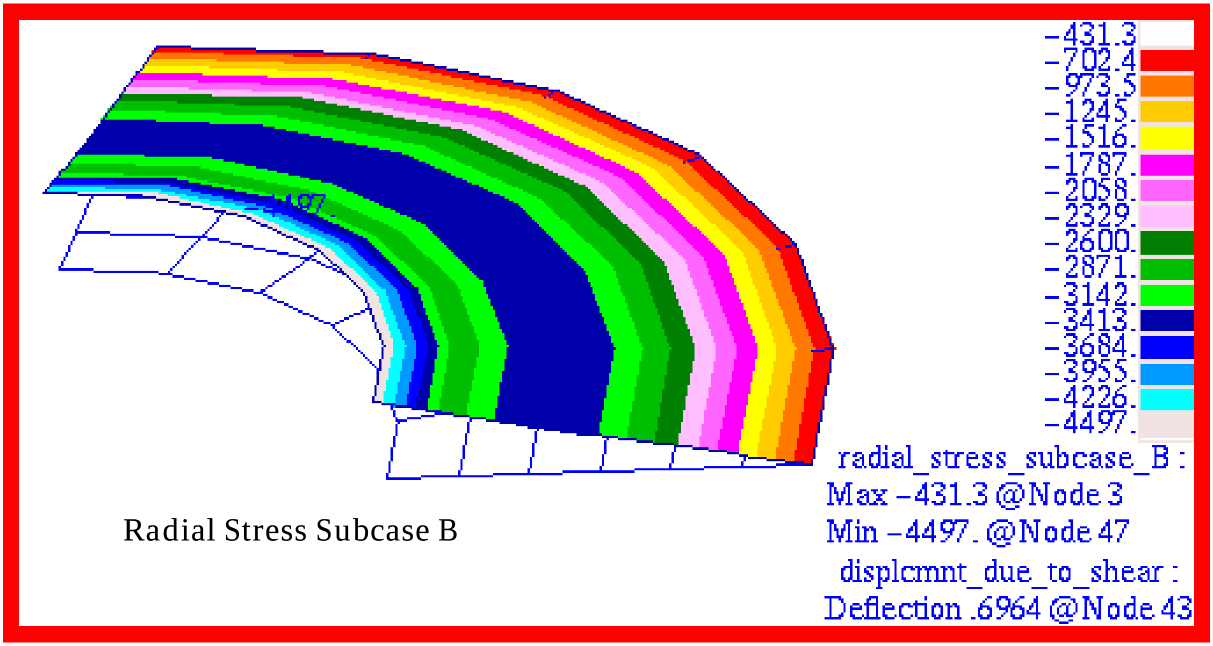


Figure 15-75 Radial Stress of Annular Plate (Subcase B).

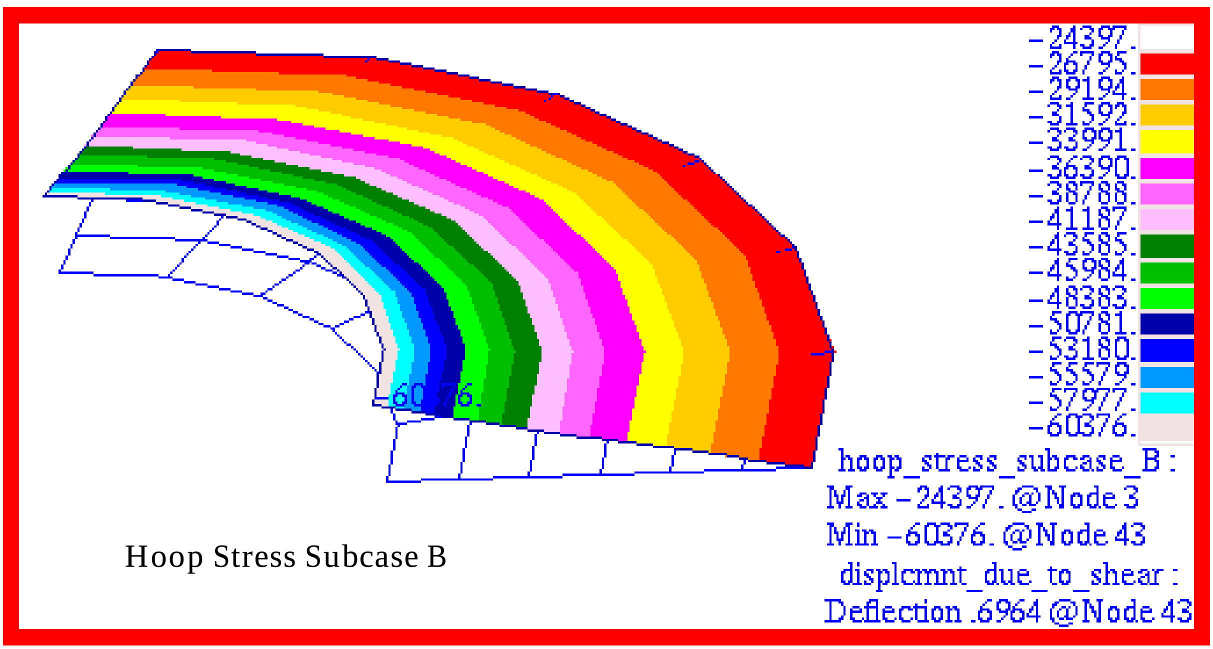


Figure 15-76 Hoop Stress of Annular Plate (Subcase B).

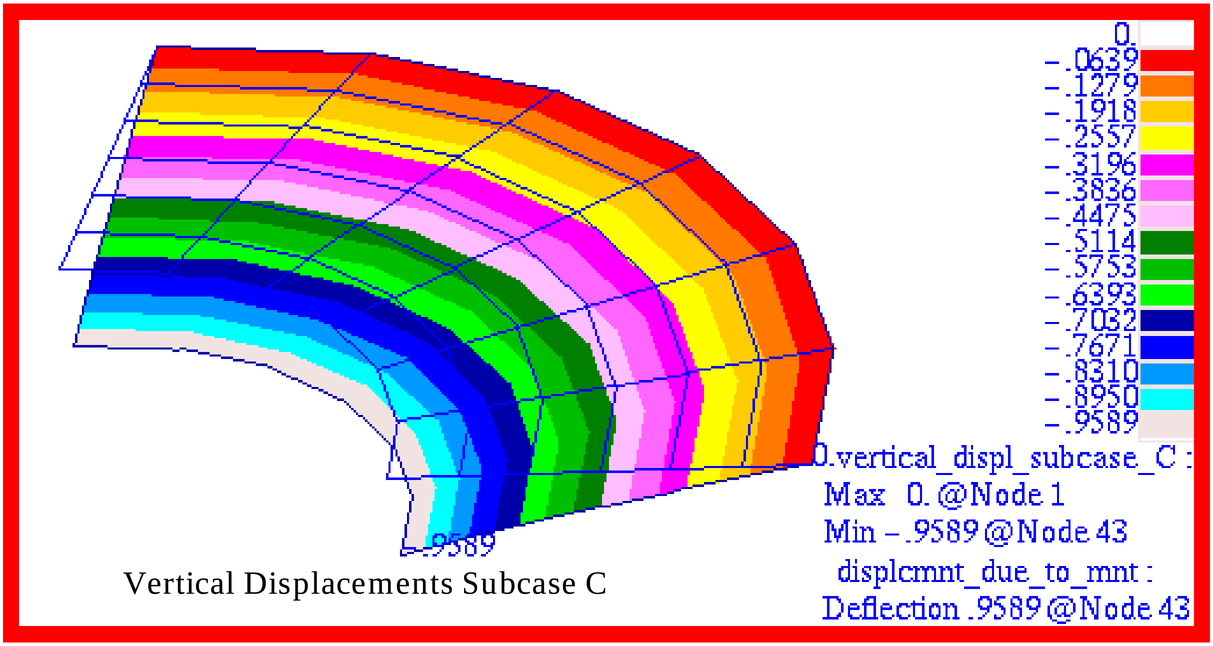


Figure 15-77 Vertical Displacement of Annular Plate (Subcase C).

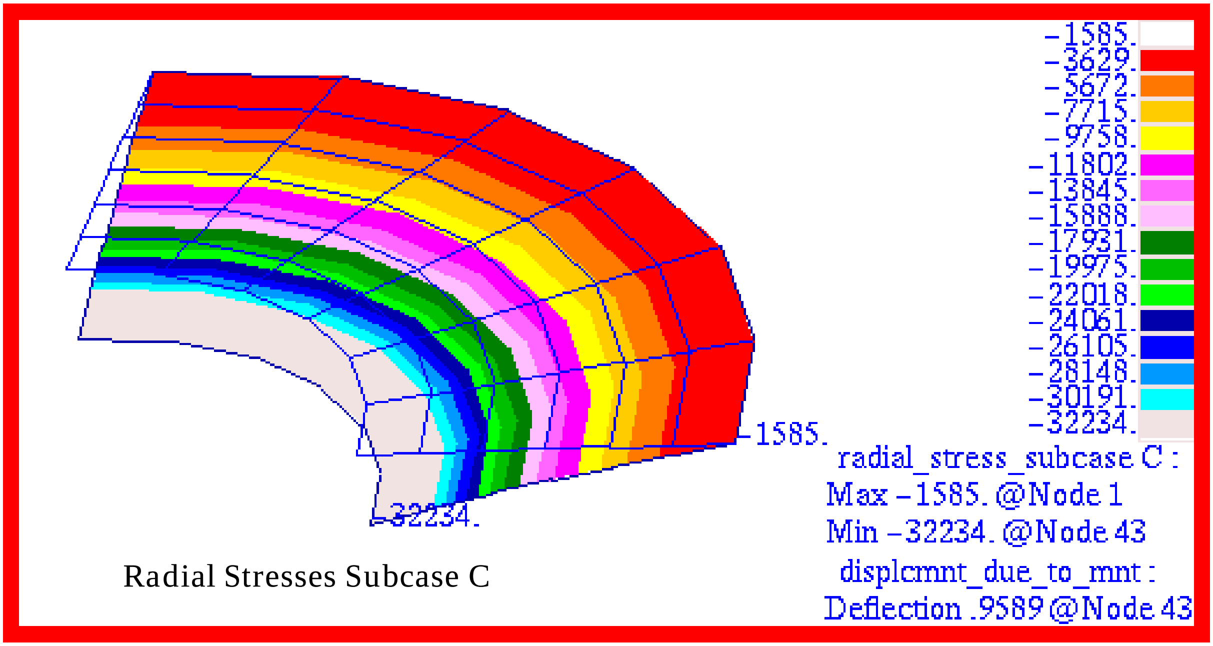


Figure 15-78 Radial Stress of Annular Plate (Subcase C).

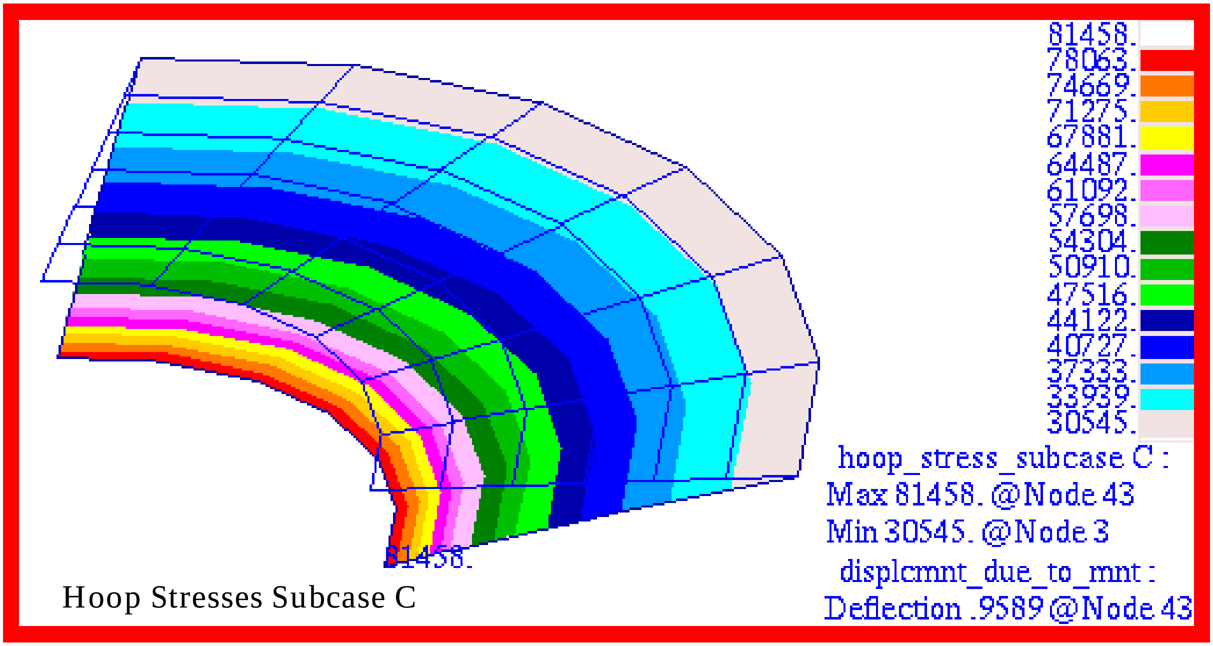


Figure 15-79 Hoop Stress of Annular Plate (Subcase C).

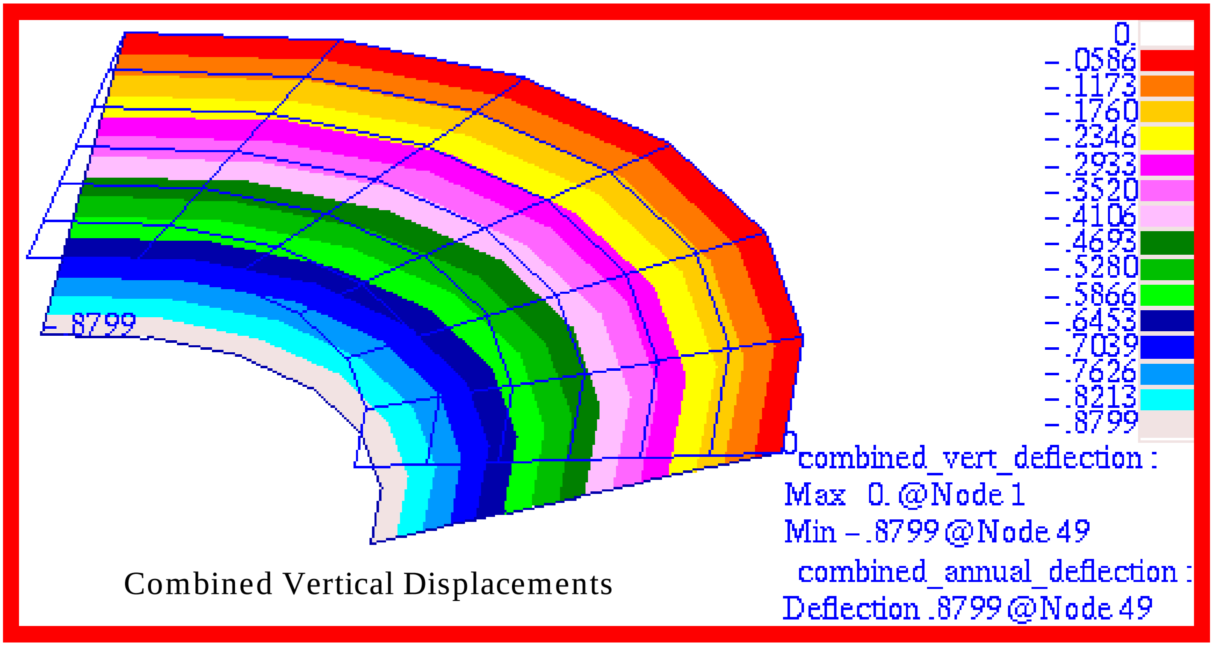


Figure 15-80 Combined Vertical Displacements of Annular Plate.

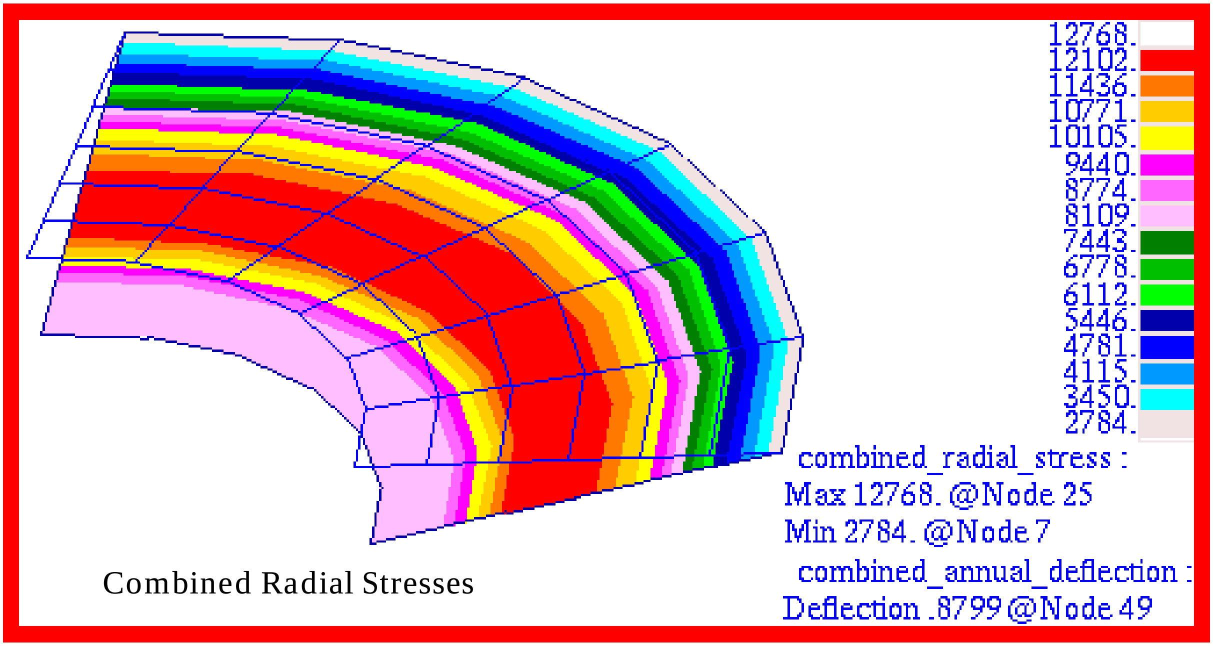


Figure 15-81 Combined Radial Stress of Annular Plate.

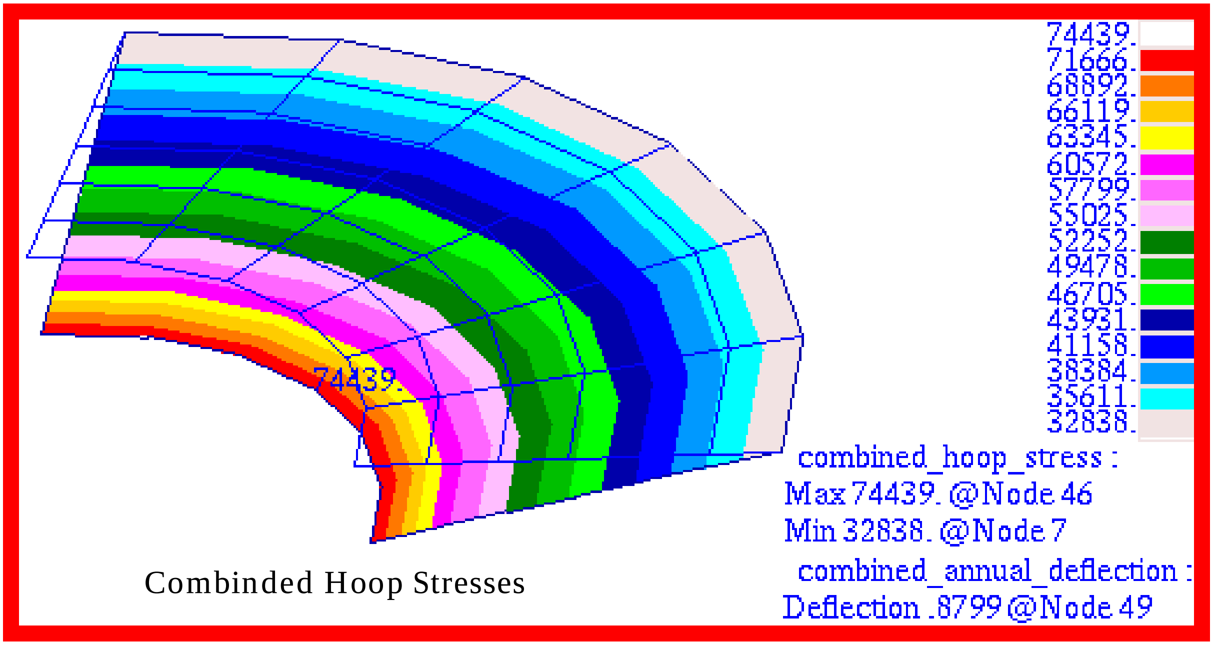


Figure 15-82 Combined Hoop Stress of Annular Plate.

Problem 12: Nonlinear Statics, Post-Buckled Column

Solution/Element Type:

MSC.Nastran, Nonlinear Statics, Solution 106, CBEAM, 1D Beams with Standard Formulation.

Reference:

Timoshenko, S. P., and Gere, J. M., *Theory of Elastic Stability*, McGraw-Hill, Inc., 1961, p. 48.

Problem Description:

Find the displaced position of a post-buckled column, fixed at one end with an applied axial load at the other end.

Engineering Data:

l = 20.0 inches

h = 1.0 inch

d = 0.05 inch

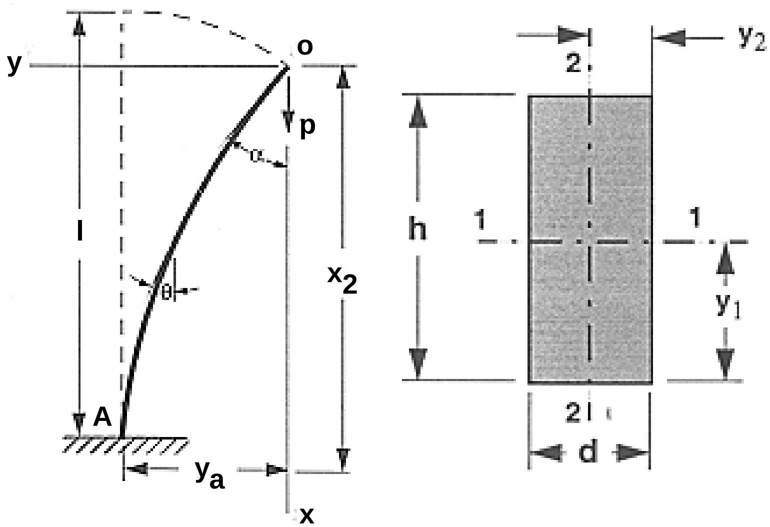
$E = 1 \times 10^8$ psi

v = .333

$\alpha = 60°$

$I_1 = (1/12)dh^3 = 4.20 \times 10^{-3}\ \text{in}^4$

$I_2 = (1/12)hd^3 = 1.04 \times 10^{-5}\ \text{in}^4$



Theoretical Solution:

The post-buckled beam end coordinates and critical buckling load are given by the equations below. The integrals $K(p)$ and $E(p)$ are known as the complex elliptical integral of the first and second kind, respectively. Their values are typically listed as a function of $\sin^{-1}(p)$. For a known deflection angle of $\alpha = 60^0$, the required the load and tip deflection can be derived as shown here

$$:P = k^2EI = (0.0843)^2(1 \times 10^8)(4.2 \times 10^{-3}) = 2984.73 \text{ lb}$$

$$x_a = \frac{2}{k}E(p) - l = \frac{2}{0.0843}(1.4675) - 20 = 14.8161 \text{ in}$$

$$y_a = \frac{2p}{k} = \frac{2(0.5)}{0.0843} = 11.8624 \text{ in}$$

$$P_{cr} = \frac{\pi^2 EI}{4l^2} = \frac{\pi^2(1 \times 10^8)(4.2 \times 10^{-3})}{4(20^2)} = 2590.77 \text{ lb}$$

where:

$$E(p) = \int_0^{\frac{\pi}{2}} \sqrt{1 - p^2(\sin\phi)^2}\, d\phi = E(\sin^{-1}(p)) = E(30^\circ) = 1.4675$$

$$k = \frac{1}{l}K(p) = \frac{1}{20}(1.6858) = 0.0843\ \text{in}^{-1}$$

$$K(p) = \int_0^{\frac{\pi}{2}} \frac{1}{\sqrt{1 - p^2(\sin\phi)^2}}\, d\phi = K(\sin^{-1}(p)) = K(30^\circ) = 1.6858$$

$$p = \sin\left(\frac{\alpha}{2}\right)$$

$$\{\sin^{-1}(p) = \alpha/2 = 30^\circ\}$$

$$\sin(p) = \sin(30^\circ) = 1/2$$

MSC.Nastran Results:

To determine the post-buckled shape for the beam, the model shown in **Figure 15-83** was generated using MSC.Patran. The model consisted of 20 1D CBEAM elements that used a standard generalized formulation. To ensure that the beam will deflect laterally, a slight horizontal load was applied to the beam in addition to an axial load of 2984.73 lbs. In this case a lateral load of 10 lbs was used. In addition, since the structure is post-buckled, it is inherently unstable. This requires that the user set the MSC.Nastran parameter TESTNEG to -2 in the input file.

The following results were obtained with MSC.Nastran.

Table 15-32 Results of Post-Buckled Beam Deflections

Source	x_a	y_a	α
MSC.Nastran	14.4076	12.2275	62.3583 ^{o}
Theory	14.8161	11.8624	60.0 ^{o}
%, Difference	-2.7571%	3.0778%	3.9305%

The corresponding MSC.Patran fringe plots that were made of the horizontal and vertical displacements are shown in **Figure 15-84** through **Figure 15-85**. Deflection plots were set to True Scale as opposed to a percentage of the model. The rotation about the global z-axis given in radians is shown in **Figure 15-86**. For the purposes of evaluating the accuracy of these plots, it is necessary to use the following conversions:

$$\Delta x = y_a$$

$$\Delta y = x_a - 20.$$

where Δx and Δy are the MSC.Nastran horizontal and vertical nodal displacements, respectively. Vector plots of the translational and rotational displacements are shown in **Figure 15-87** and **Figure 15-88**. After making the appropriate conversions, it is apparent that the MSC.Patran results are identical to those obtained with MSC.Nastran.

File(s):<install_dir>/results_vv_files/prob012.bdf, prob012.op2

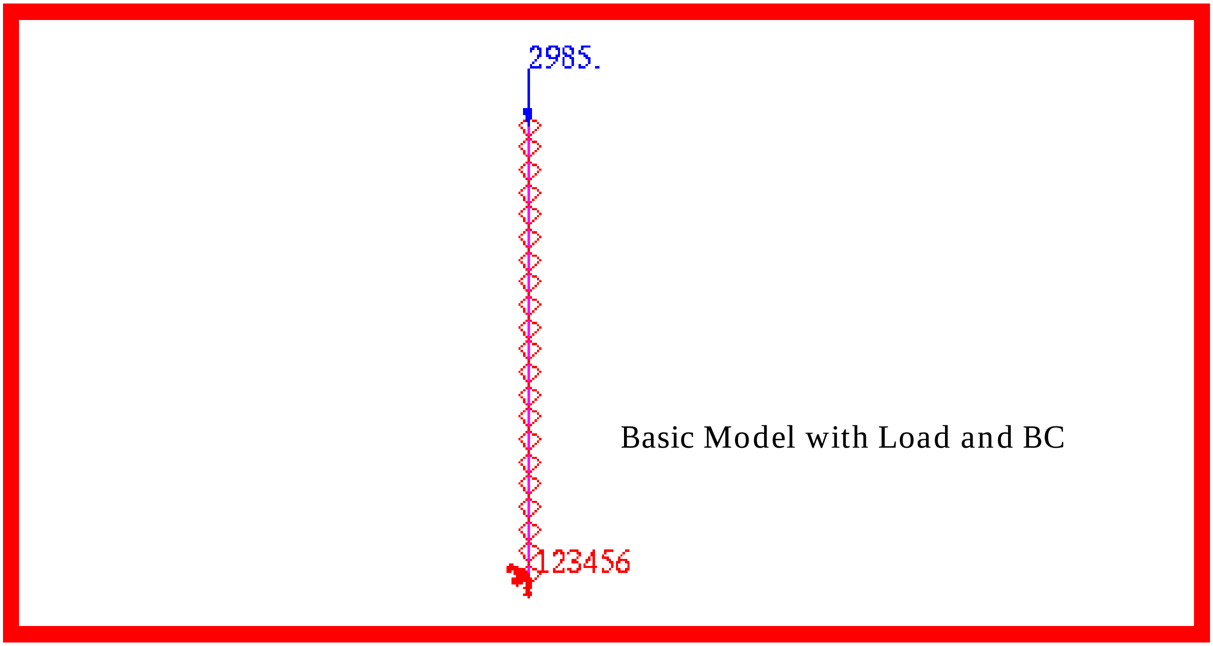


Figure 15-83 Nonlinear Beam Post-Buckling Model.

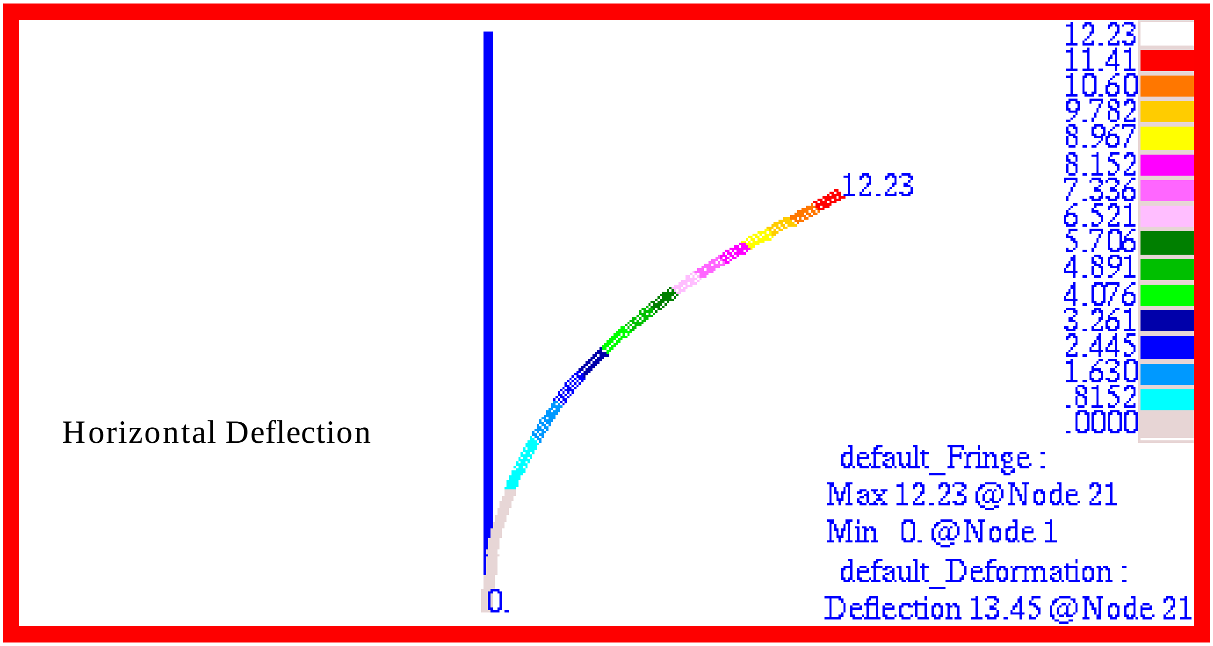


Figure 15-84 Post-Buckled Horizontal Deformation of Beam.

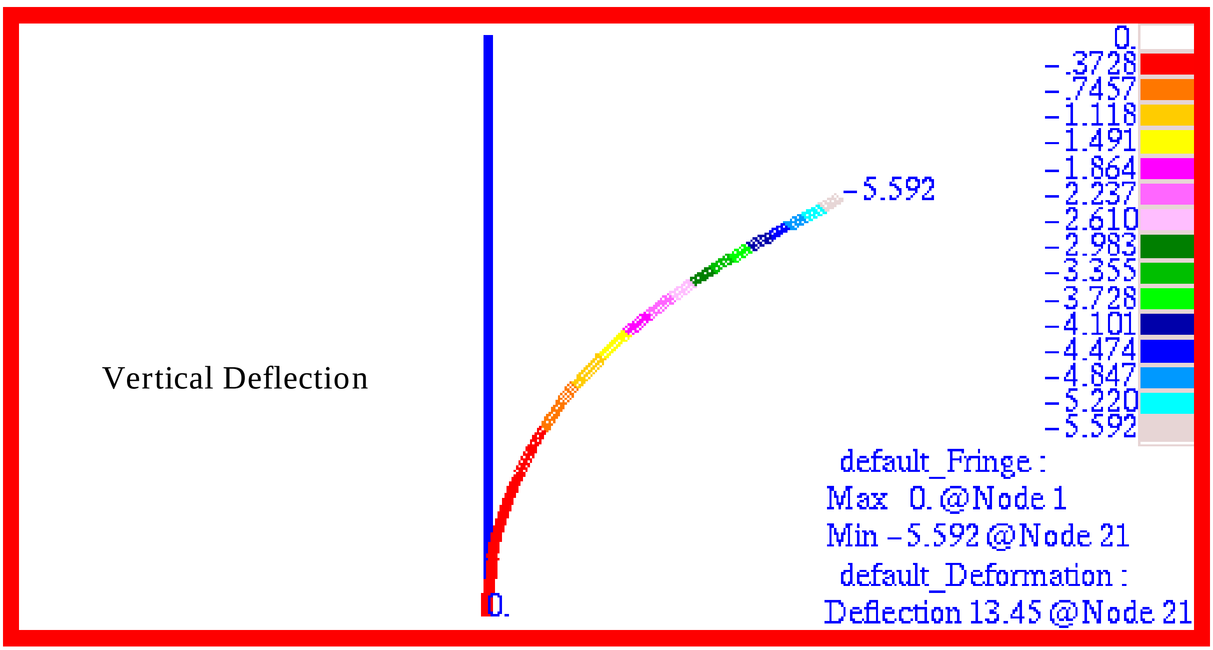


Figure 15-85 Post-Buckled Vertical Deformation of Beam.

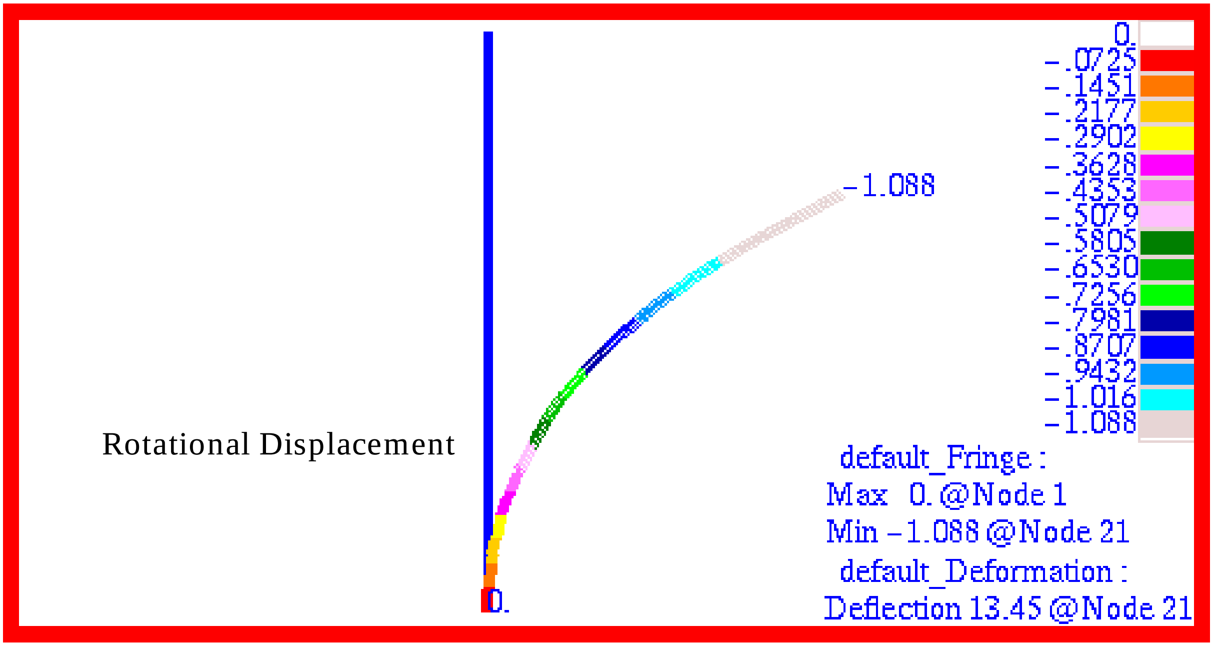


Figure 15-86 Post-Buckled Rotational Deformation about Z of Beam.

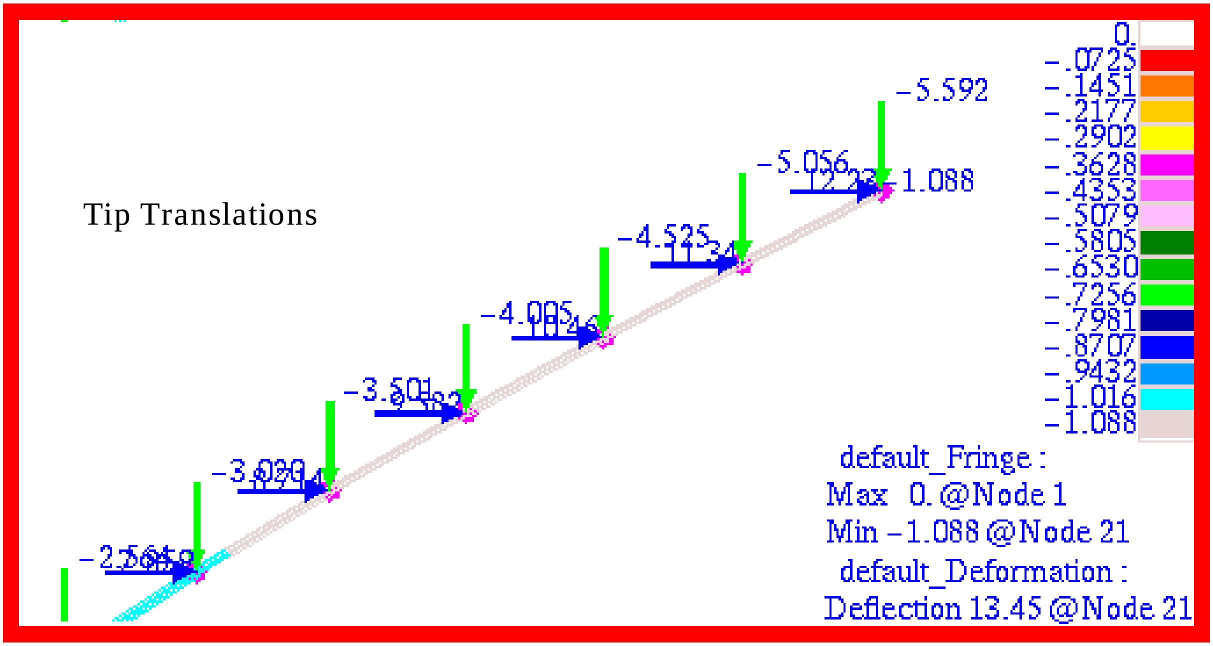


Figure 15-87 Vector Plot of Tip Deflections of Beam.

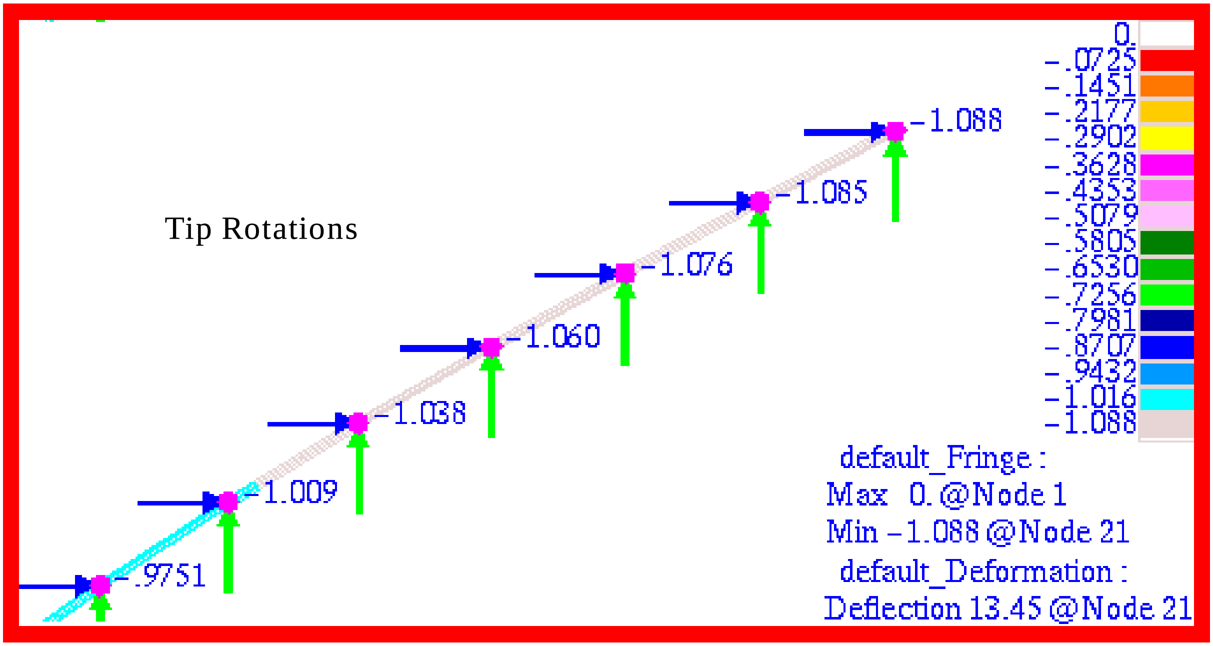


Figure 15-88 Vector Plot of Rotational Deformation of Beam.

Problem 13: Nonlinear Statics, Beams with Gap Elements

Solution Type:

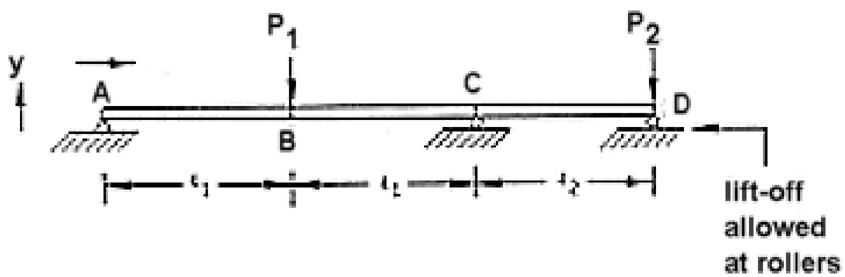
MSC.Nastran, Nonlinear Statics, Solution 106. Normal Modes, CBEAM and CGAP

Reference:

McCormac, J.C., *Structural Analysis*, 3rd ed., New York: Intext Educational Publishers, 1975, p. 323, ex. 17.12.

Problem Description:

A simply supported beam is hinged at one end and supported by lifting rollers at two other locations. Allowing for lift-off to occur, determine the vertical deflections under the load points.



Engineering Data:

$E = 29. \times 10^6$ psi

$A = 83.3$ in^2

$P_1 = 40000$ lb

$\ell_1 = 15$ ft

$I_{zz} = 1000.0$ in^4

$P_2 = 10000$ lb

$\ell_2 = 10$ ft

$I_{yy} = 334.0$ in^4

Theoretical Solution:

(Lift-off occurs at point D only)

Point B: $U_y = -1.01$ in

Point D: $U_y = +0.546$ in

MSC.Nastran Results:

To compute the vertical displacement at the load points, the model shown in **Figure 15-1** was generated using MSC.Patran. This model consisted of 13 CBEAM elements as well as two CGAP elements that were used to model the roller supports at points C and D. Since the CGAP elements had zero length, an additional coordinate frame, shown at the left end of the model, was used to define the orientation for both CGAP elements. In addition, both CGAP elements were provided a high closed stiffness, but no open stiffness. In this way the beam could lift-off the roller supports. Using this model, the following vertical displacements were calculated at both load points:

Table 15-1 Vertical Deflection of Beam with Lift-Off

Source	U_y, Point B	U_y, Point D
MSC.Nastran	-1.01	+0.546
Theory	-1.01	+0.544
%, Difference	0.0%	0.366%

The corresponding fringe plot that was made of the displacements with MSC.Patran is shown in **Figure 15-2**. Here the fringe plot for displacements has been superimposed upon an exaggerated deformation plot. In addition, the maximum and minimum values for the vertical displacements are shown at the nodes where they occur. These values are clearly identical to the MSC.Nastran results.

File(s): `<install_dir>/results_vv_files/prob012.bdf, prob012.op2`

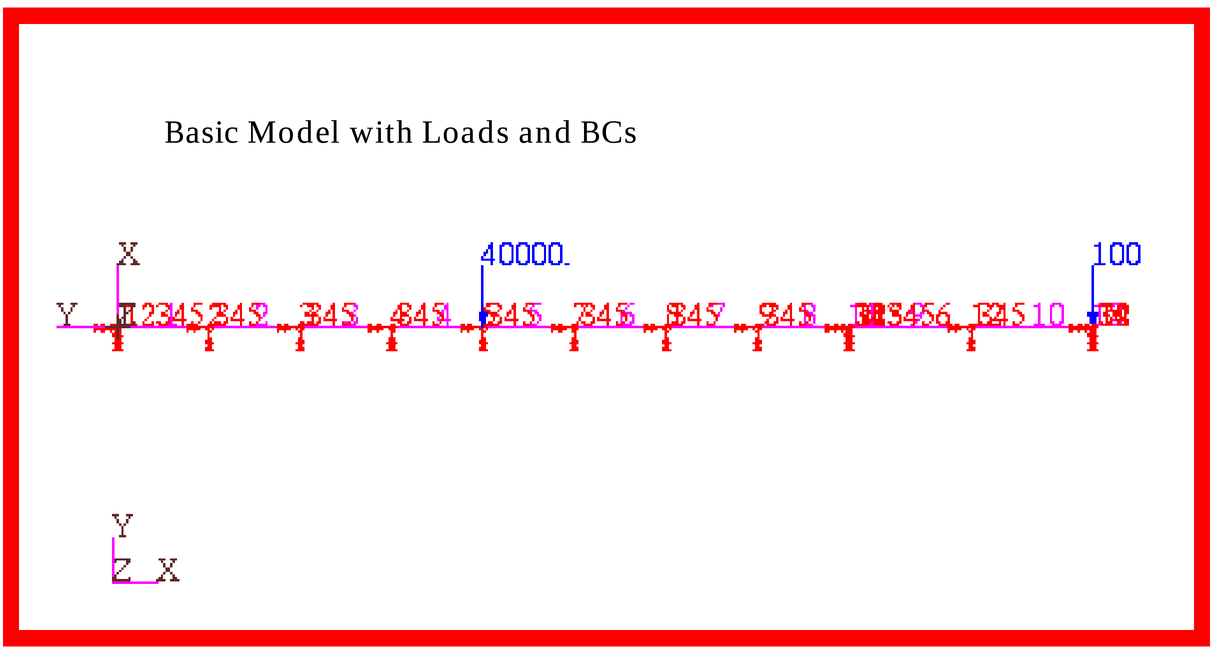


Figure 15-1 Model of Non-Linear Gap Problems with Lift-off.

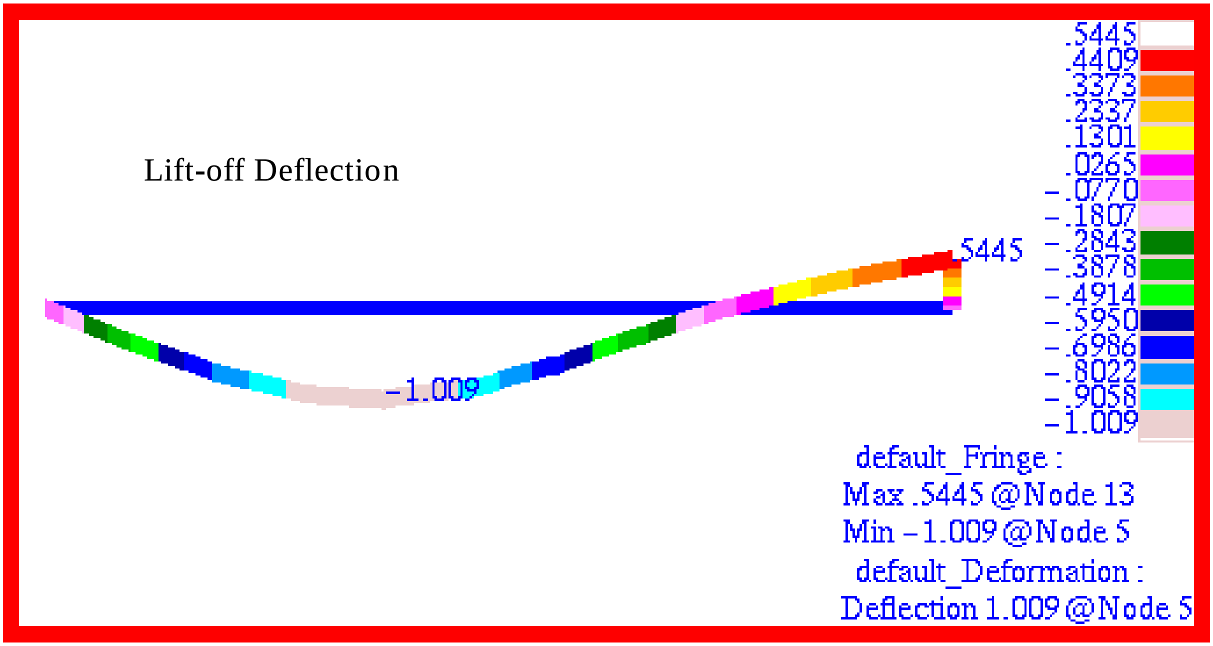


Figure 15-2 Deflection Plot of Lift-off of Beam.

Problem 14: Normal Modes, Point Masses and Linear Springs

Solution/Element Type:

MSC.Nastran, Normal Modes, Solution 103, CONM2 and CELAS1 Elements.

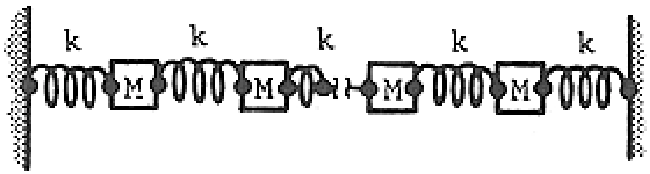
Reference:

Blevins, R.D., *Formulas For Natural Frequency and Mode Shape*, Kreiger Publishing Co., 1984, p. 50.

Problem Description:

Four equal masses are linked by five equal springs. Assuming that all motion is confined to the linear axis of the springs, determine the natural frequencies for the first four modes.

Engineering Data:



$$k = 100 \frac{lb}{in}$$

$$M = 2.0 \frac{lb - \sec^2}{in}$$

Theoretical Solution:

For a system consisting of four equal masses and five springs, the natural frequencies of and mode are given by the equations below. Substituting in the values for the mass and spring constants yields the following natural frequencies for the first four mode shapes:

$$f_i = \frac{\alpha_{4,i}}{2\pi}\left(\frac{k}{M}\right)^{1/2} \ ; \ i = 1, 2, 3, 4$$

$$\alpha_{4,i} = 2 \sin\left[\frac{i}{5}\left(\frac{\pi}{2}\right)\right]$$

$$f_1 = 0.6955 \text{ Hz}$$

$$f_2 = 1.3230 \text{ Hz}$$

$$f_3 = 1.8209 \text{ Hz}$$

$$f_4 = 2.1406 \text{ Hz}$$

MSC.Nastran Results:

To compute the natural frequencies for the spring-mass system, the model shown in **Figure 15-3** was generated using MSC.Patran. The masses were modeled using CONM2 point mass elements while the springs were modeled with linear elastic CELAS1 elements. All motion was restricted to the x-, or axial, direction. The opposing ends of the model were fixed. The following results were computed:

Table 15-2 Natural Frequencies, Hertz

Source	f_1	f_2	f_3	f_4
MSC.Nastran	0.6955	1.3230	1.8209	2.1406
Theory	0.6955	1.3230	1.8209	2.1406
%, Difference	0.0%	0.0%	0.0%	0.0%

Due to the fact that all of the mode shapes merely involve horizontal displacements of the four masses, the mode shapes cannot be readily discerned by making combined fringe and deformation plots of the translational displacements associated with each eigenvector. Instead, vector plots were made of the translational displacement that were superimposed upon the deformed shape of the eigenvectors. In this way, the vectors would be drawn at the displaced position of each node, or mass. The eigenvectors and natural frequencies that were generated by MSC.Patran for the first modes are plotted in **Figure 15-4** through **Figure 15-7**. Note that the natural frequencies recovered through MSC.Patran are identical to those obtained with MSC.Nastran.

File(s): <install_dir>/results_vv_files/prob014.bdf, prob014.op2

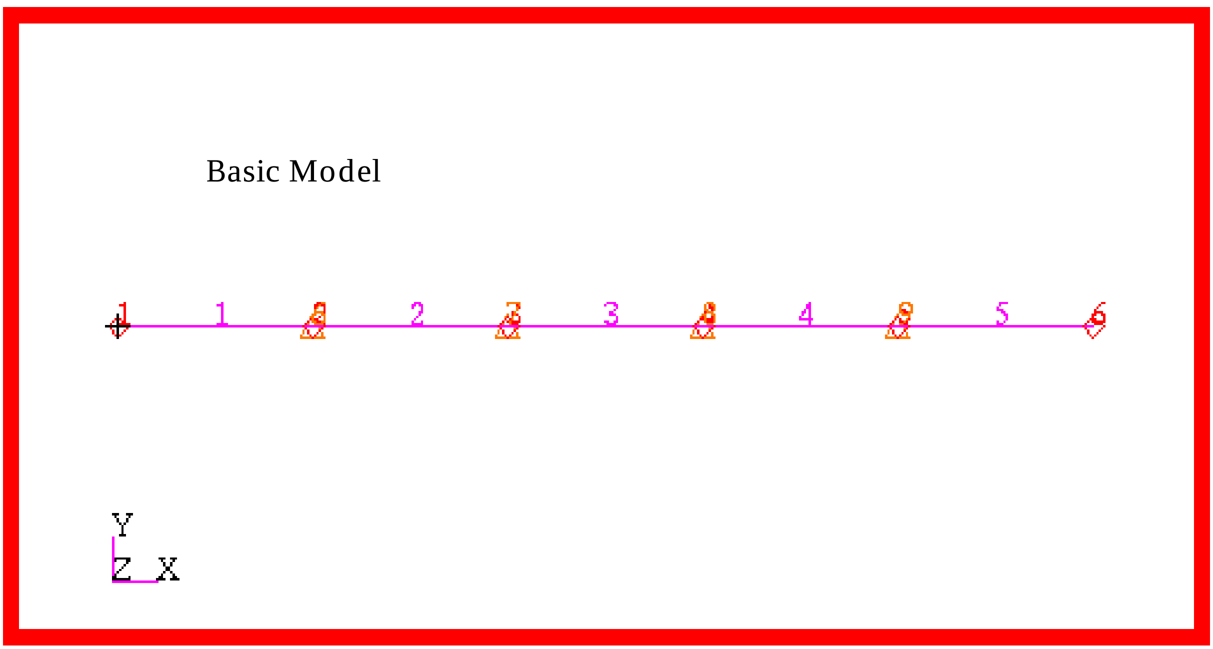


Figure 15-3 Spring and Mass Modal Model.

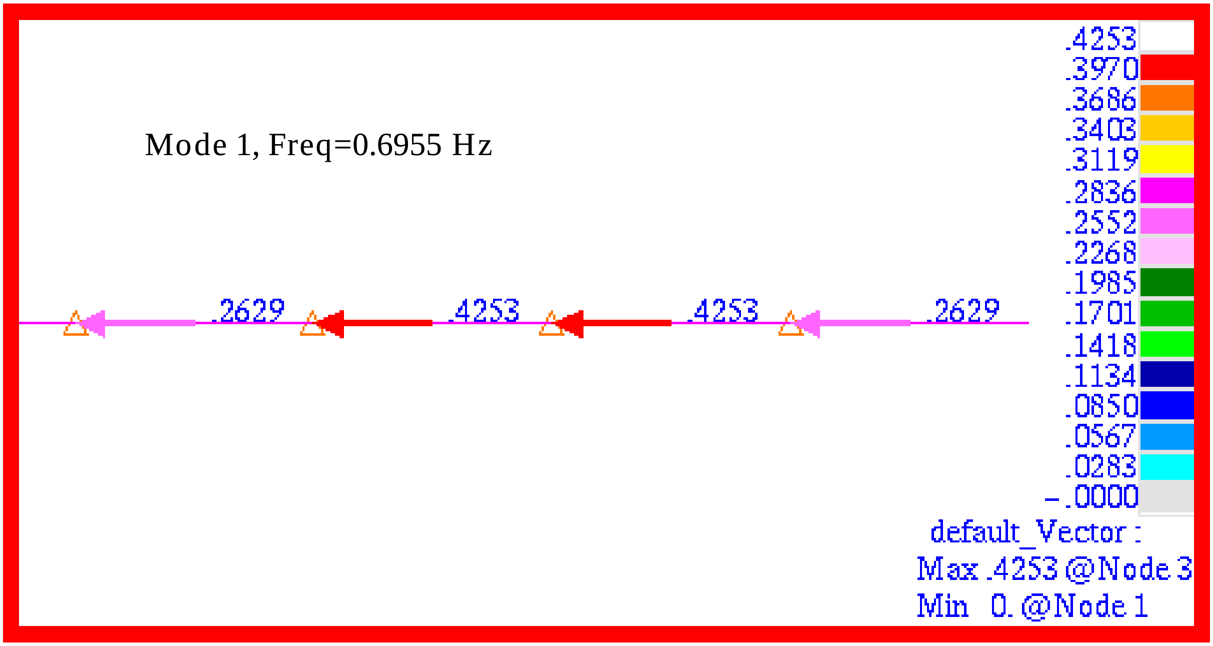


Figure 15-4 Vector Plot of Mode 1.

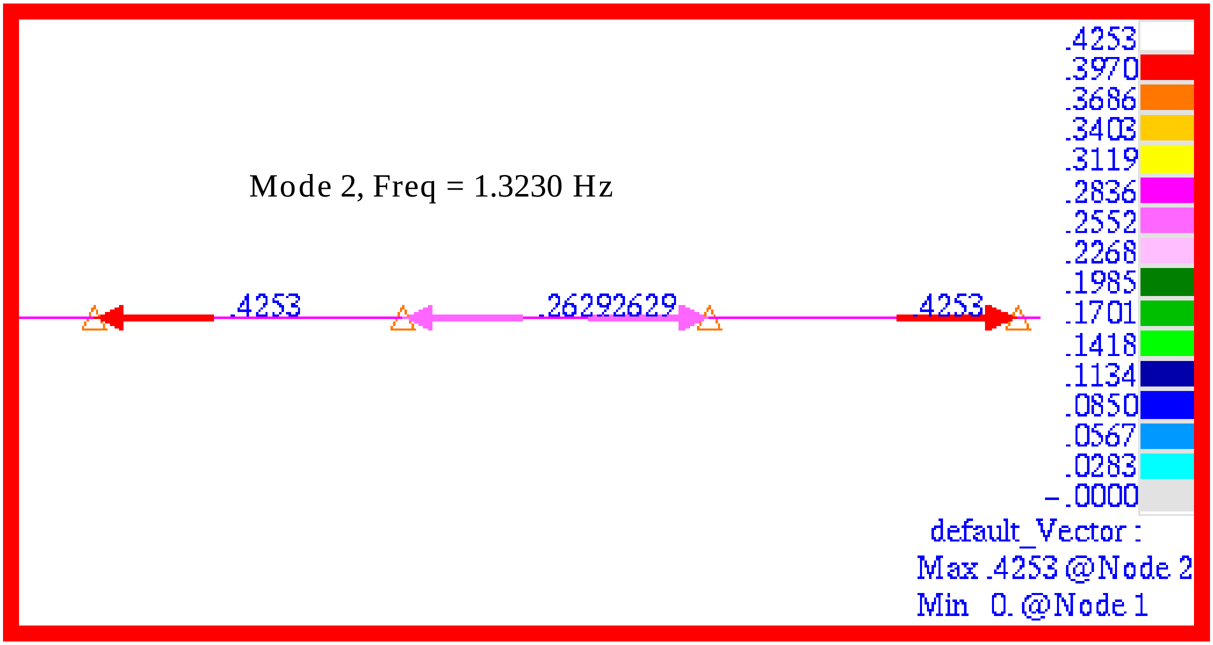


Figure 15-5 Vector Plot of Mode 2.

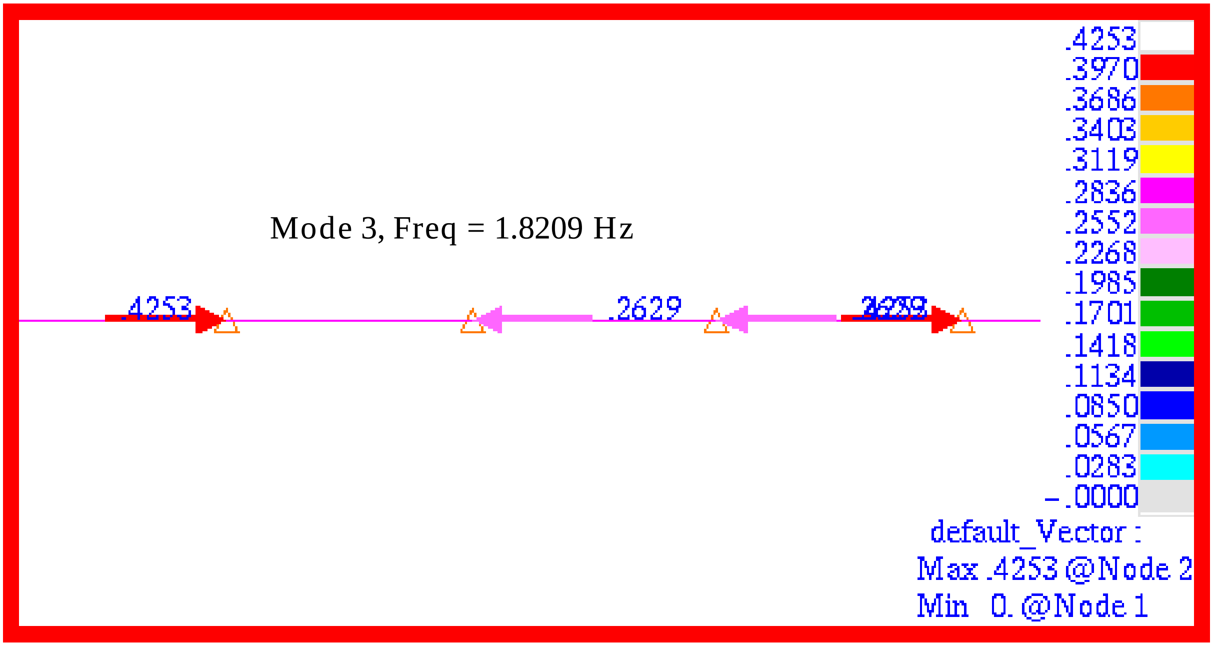


Figure 15-6 Vector Plot of Mode 3.

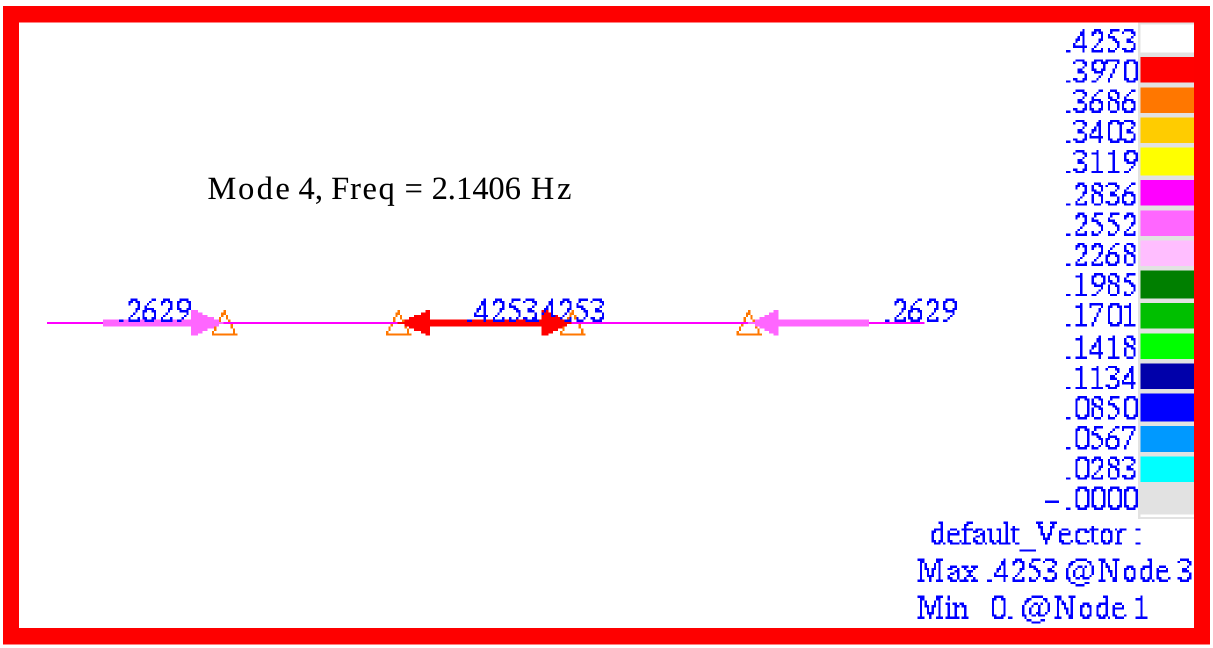


Figure 15-7 Vector Plot of Mode 4.

Problem 15: Normal Modes, Shells and Cylindrical Coordinates

Solution/Element Type:

MSC.Nastran, Normal Modes, Solution 103, CTRIA3 Elements with Standard Formulation.

Reference:

Blevins, R.D., *Formulas For Natural Frequency And Mode Shape*, Kreiger Publishing Co., 1984, p. 240.

Problem Description:

Find the natural frequencies and mode shapes for a simply supported flat circular plate.

Engineering Data:

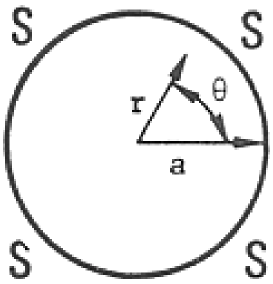
a = radius = 4.0 inches

h = thickness = 0.25 inches

E = elastic modulus = 10^7 psi

ν = poissons ratio = 0.3

γ = weight per unit area = 6.5×10^{-5} lb-sec^2 / in^3



Theoretical Solution:

For simply supported flat circular plate, the natural frequencies are given by the following expression:

$$\text{Natural Frequency (hertz), } f_{ij} = \frac{\lambda_{ij}^2}{2\pi a^2}\left[\frac{Eh^3}{12\gamma(1-\nu^2)}\right]^{1/2} \quad ;i = 0, 1, 2,...; j = 0, 1, 2,...$$

where for various values of i and j, λ_{ij} is given by:

Table 15-3 λ_{ij} **j**

j	i		
	0	**1**	**2**
0	4.977	13.94	25.65
1	29.76	48.51	70.14
2	74.20	102.8	134.3
3	138.3	176.8	218.2

The natural frequencies for various modes are listed and shown below.

f_{00} = 734.53 hz

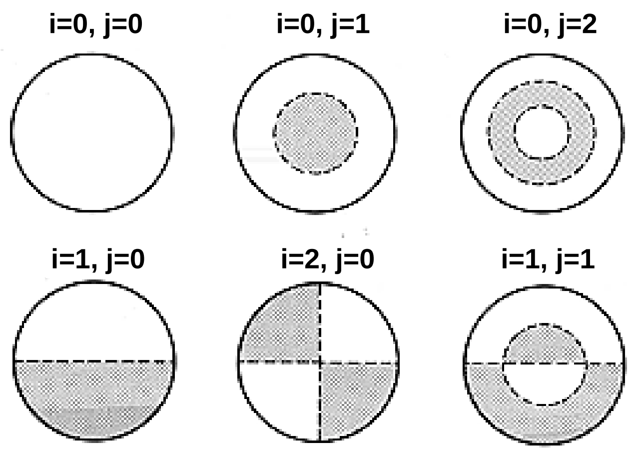
f_{10} = 2057.33 hz

f_{20} = 3785.55 hz

f_{01} = 4392.13 hz

f_{11} = 7159.35 hz

f_{02} = 10950.81 hz



MSC.Nastran Results:

To compute the mode shapes for the plate, the model shown in **Figure 15-8** was generated. Due to the cylindrical geometry, the model was meshed entirely with CTRIA3 elements which should provide accuracy better than a CQUAD4 element that is better suited for rectilinear geometry. In addition, the entire plate was modeled in order to provide better visualization of the modes. The following results were obtained.

Table 15-4 Natural Frequency, f_{ij} (hertz)

Source	f_{00}	f_{10}	f_{20}	f_{01}	f_{11}	f_{02}
Theory	734.53	2057.33	3785.55	4392.13	7159.35	10950.81
MSC.Nastran	721.37	2034.82	3713.55	4238.06	6941.60	10125.20
%, Difference	-1.79%	-1.09%	-1.90%	-3.51%	-3.04%	-7.54%

The corresponding combined fringe and deformation plots that were made for each of these modes are shown in **Figure 15-10** through **Figure 15-13** where the corresponding mode number and frequency have been clearly labeled.

File(s):`<install_dir>/results_vv_files/prob015.bdf, prob015.op2`

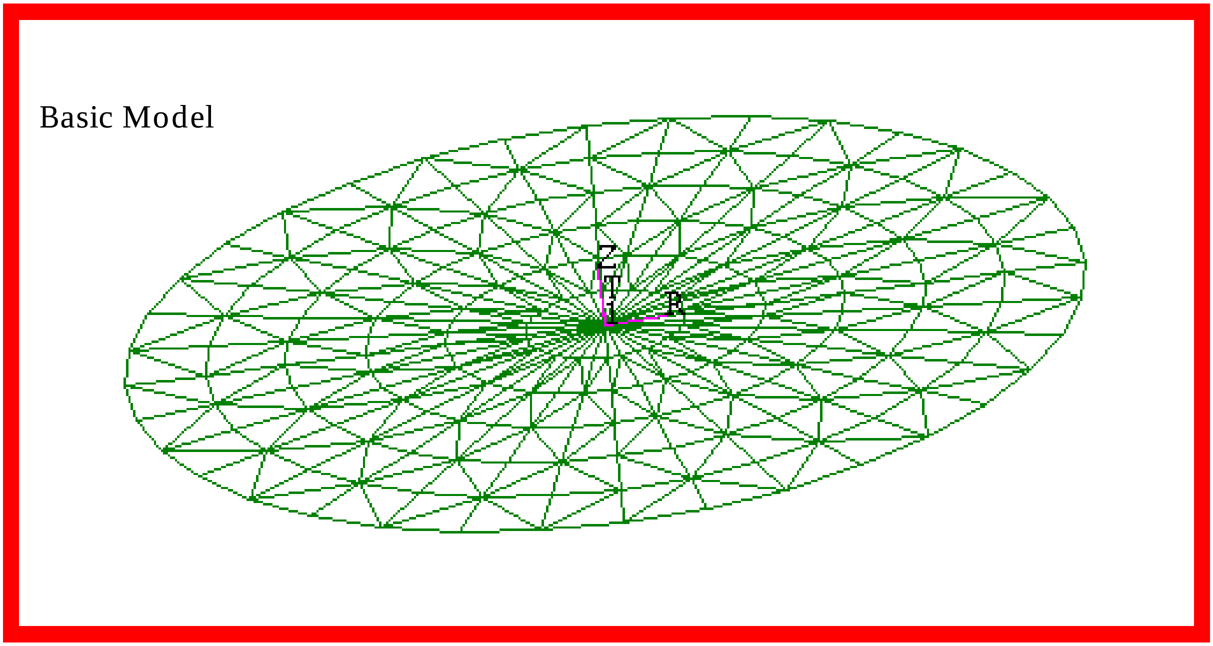


Figure 15-8 Basic Circular Membrane Model with h-Elements.

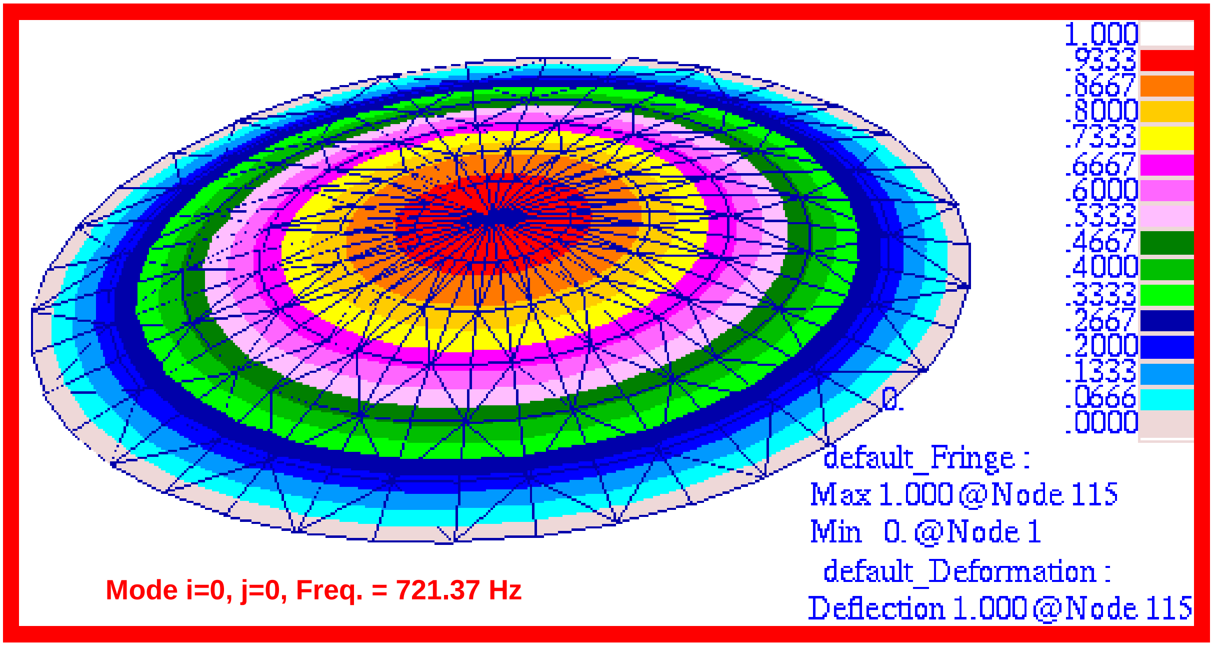


Figure 15-9 Mode 1, f_{00}, for Membrane h-Element Model.

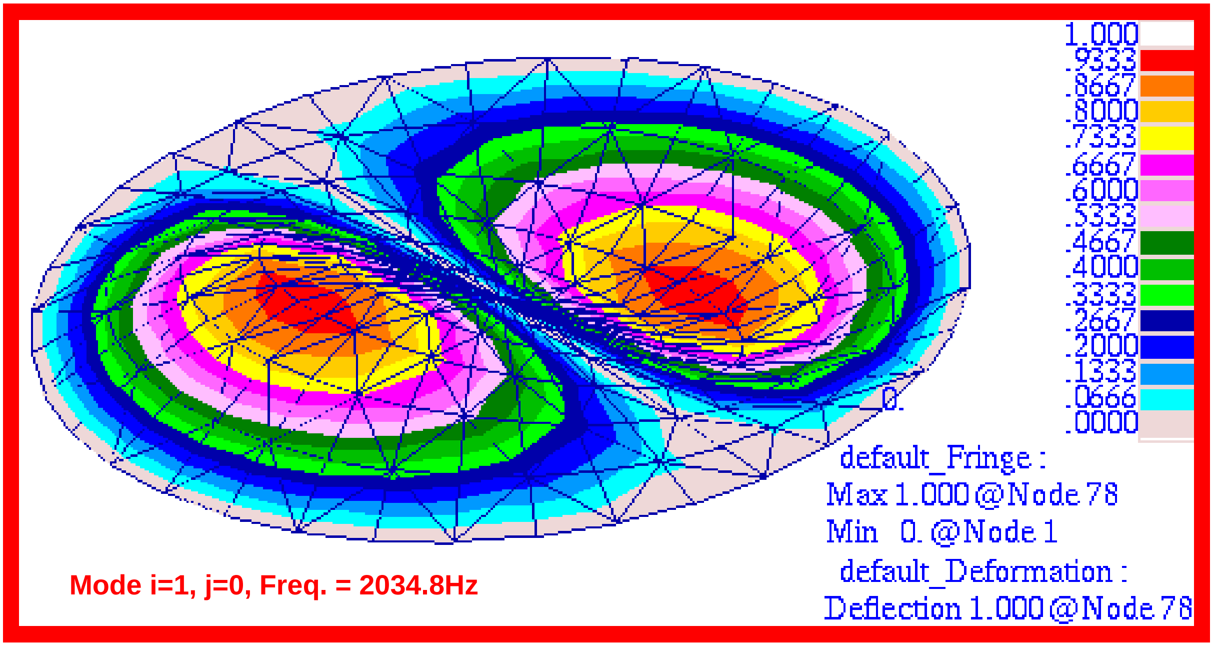


Figure 15-10 Mode 2, f_{10}, for Membrane h-Element Model.

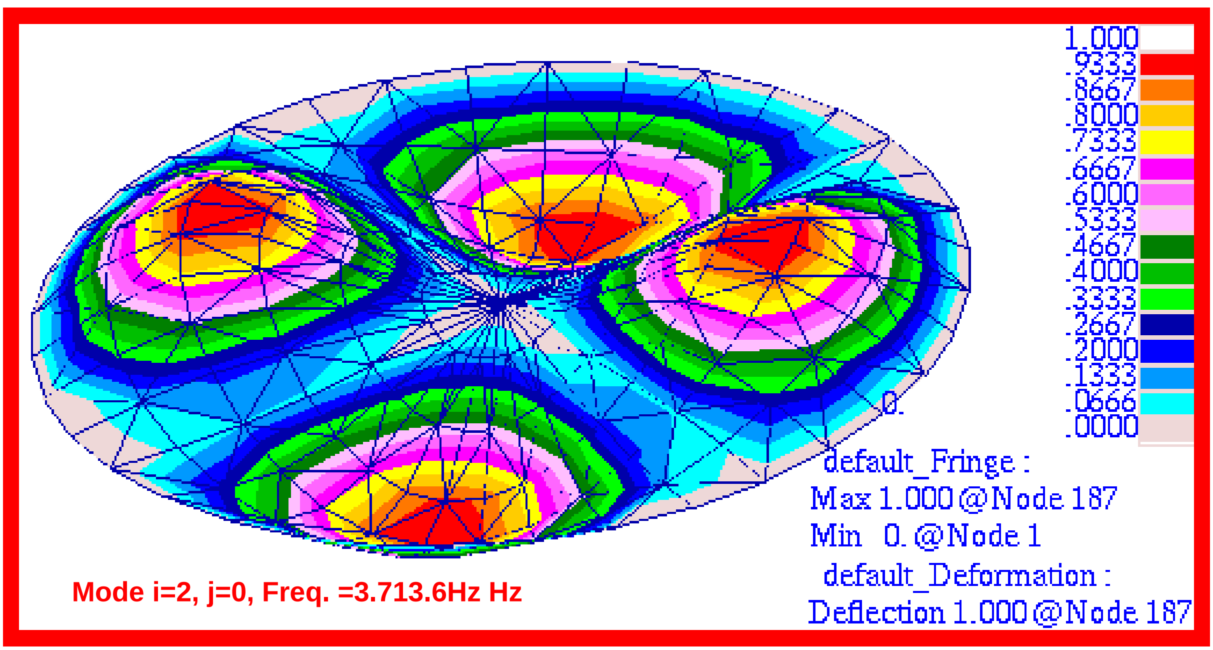


Figure 15-11 Mode 4, f_{20}, for Membrane h-Element Model.

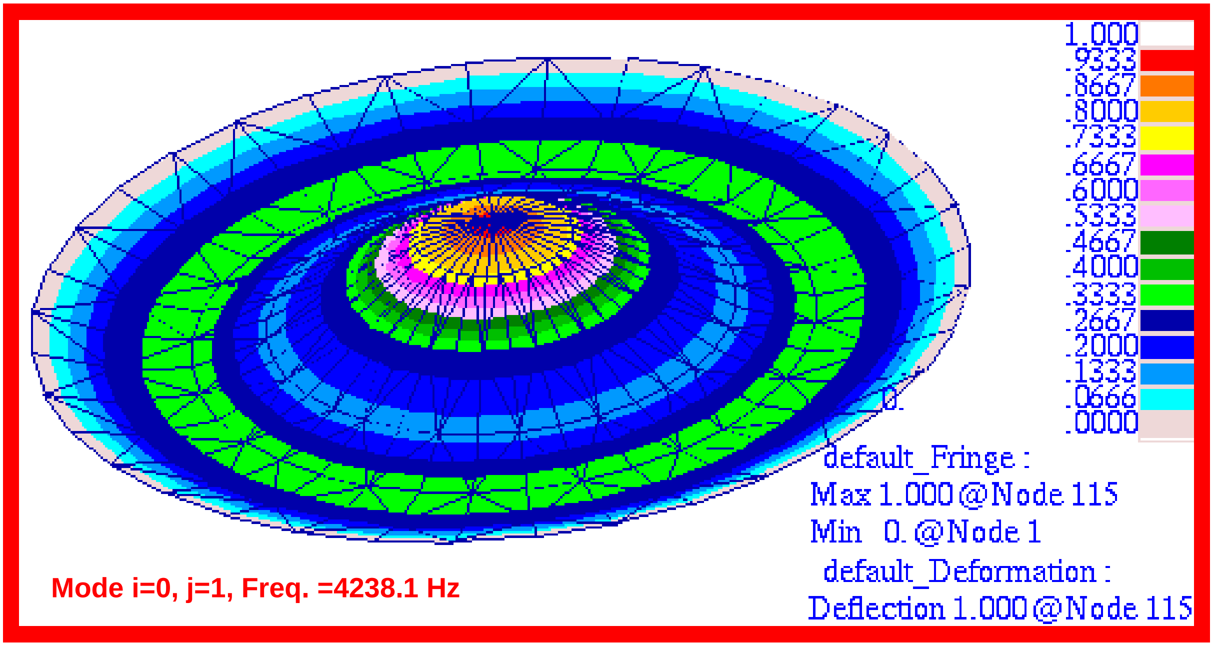


Figure 15-12 Mode 6, f_{01}, for Membrane h-Element Model.

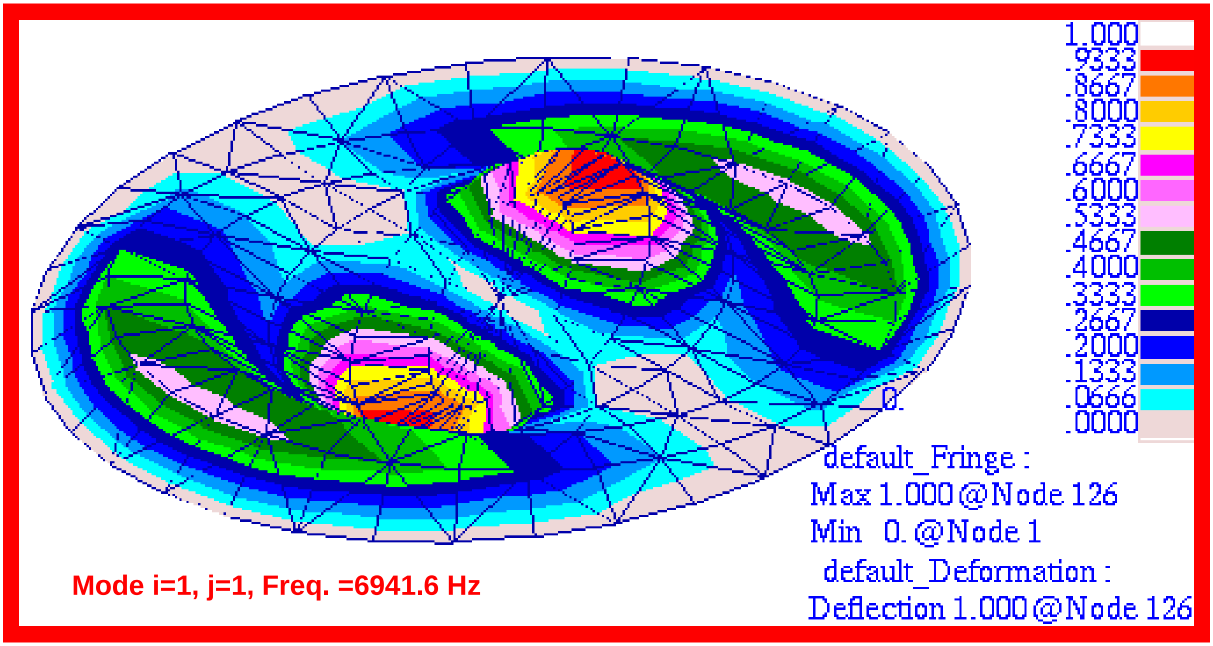


Figure 15-13 Mode 9, f_{11}, for Membrane h-Element Model.

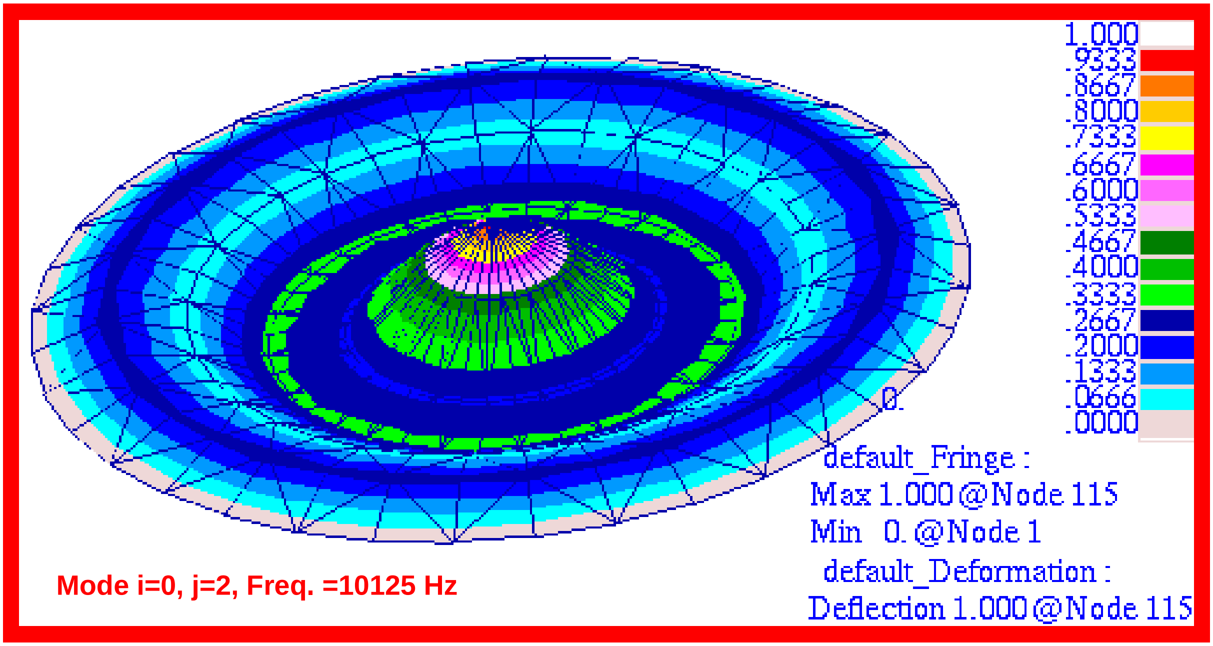


Figure 15-14 Mode 15, f_{02}, for Membrane h-Element Model.

Problem 16: Normal Modes, Pshells and Cylindrical Coordinates

Solution/Element Type:

MSC.Nastran, Normal Modes, Solution 103, CTRIA, P-Formulation

Reference:

Blevins, R.D., *Formulas For Natural Frequency and Mode Shape*, Kreiger Publishing Co,. 1984, p. 240.

Problem Description:

This is a repeat of **Problem 15: Normal Modes, Shells and Cylindrical Coordinates** (p. 403) which is to find the natural frequencies and mode shapes for a simply supported flat circular plate. In this instance, however, perform the calculation using cubic shell p-elements.

Theoretical Solution:

See **Problem 15: Normal Modes, Shells and Cylindrical Coordinates** (p. 403).

MSC.Nastran Results:

To compute the modes for the plate, the model shown in **Figure 15-15** was generated using MSC.Patran. Due to the curvilinear geometry and nature of the modes shapes, the model was meshed entirely with triangular shell elements. In this instance, all of the elements were cubic TRIA13 p-elements. Due to the higher order of these elements, this permitted using a considerably sparser mesh than was used with simple linear TRIA3 elements in order to properly capture the various modes. A sparser mesh than what is shown in **Figure 15-15** was found to only give reasonably accurate results for only the very lowest modes. The lack of an appropriate number of degrees of freedom necessarily precluded capturing some of the higher modes which typically exhibit considerably more complex shapes.

The following natural frequencies were calculated for some of the various modes.

f_{00} = 726.71 hz $\quad$ f_{20} = 3736.80 hz $\quad$ f_{11} = 7259.80 hz

f_{10} = 2037.80 hz $\quad$ f_{01} = 4377.78 hz $\quad$ f_{02} = 11178.31 hz

Table 15-5 Natural Frequency, f_{ij} (hertz)

Source	f_{00}	f_{10}	f_{20}	f_{01}	f_{11}	f_{02}
Theory	734.53	2057.33	3785.55	4392.13	7159.35	10950.81
MSC.Nastran	726.71	2037.80	3736.80	4377.78	7259.80	11178.31
%, Difference	-1.06%	0.95%	-1.29%	-0.33%	1.40%	2.08%

The corresponding combined fringe and deformation plots that were made with MSC.Patran for each of these modes are shown in **Figure 15-16** through **Figure 15-20** where the mode number and frequency have been clearly labeled. A comparison of these figures with the figures of **Problem 15: Normal Modes, Shells and Cylindrical Coordinates** (p. 403) clearly shows that the predicted mode shapes are being accurately predicted by MSC.Nastran and displayed with

MSC.Patran. In addition, it is interesting to note that the use of higher order pshell elements gives comparable accuracy for the first couple of primary modes but gives noticeably more accurate results for the higher order modes, the error being typically about one-fourth or less.

File(s):<install_dir>/results_vv_files/prob016.bdf, prob016.op2

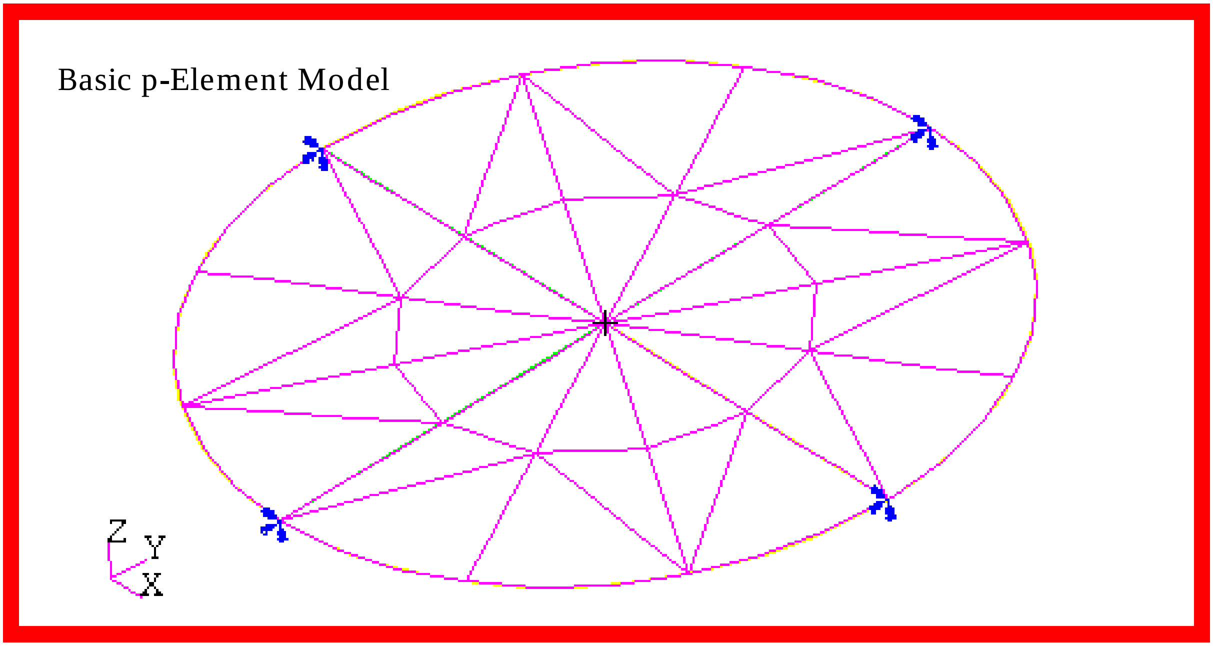


Figure 15-15 Basic p-Element Membrane Model.

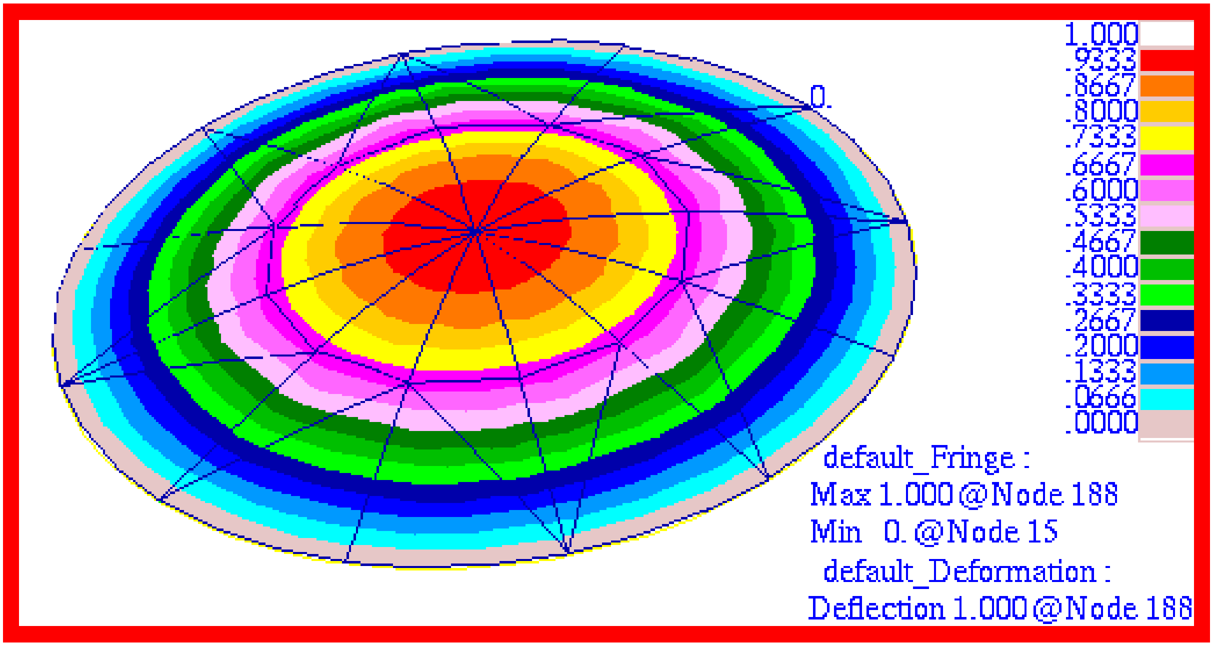


Figure 15-16 Mode 1, f_{00}, of p-Element Membrane Model.

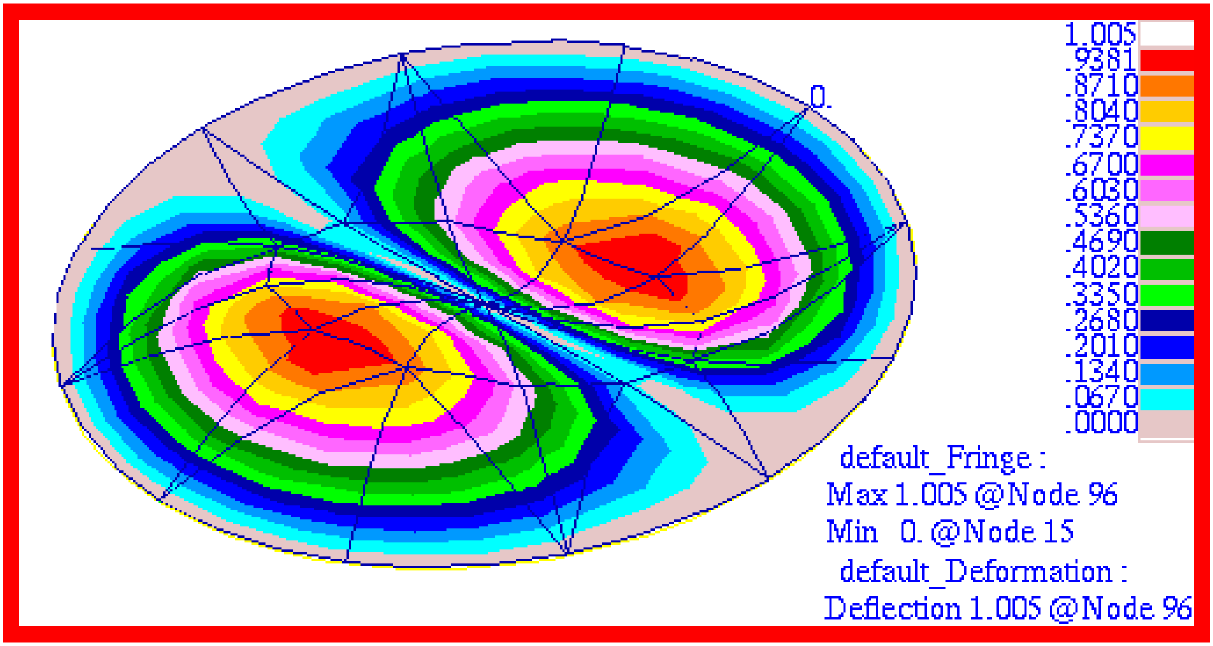


Figure 15-17 Mode 2, f_{10}, of p-Element Membrane Model.

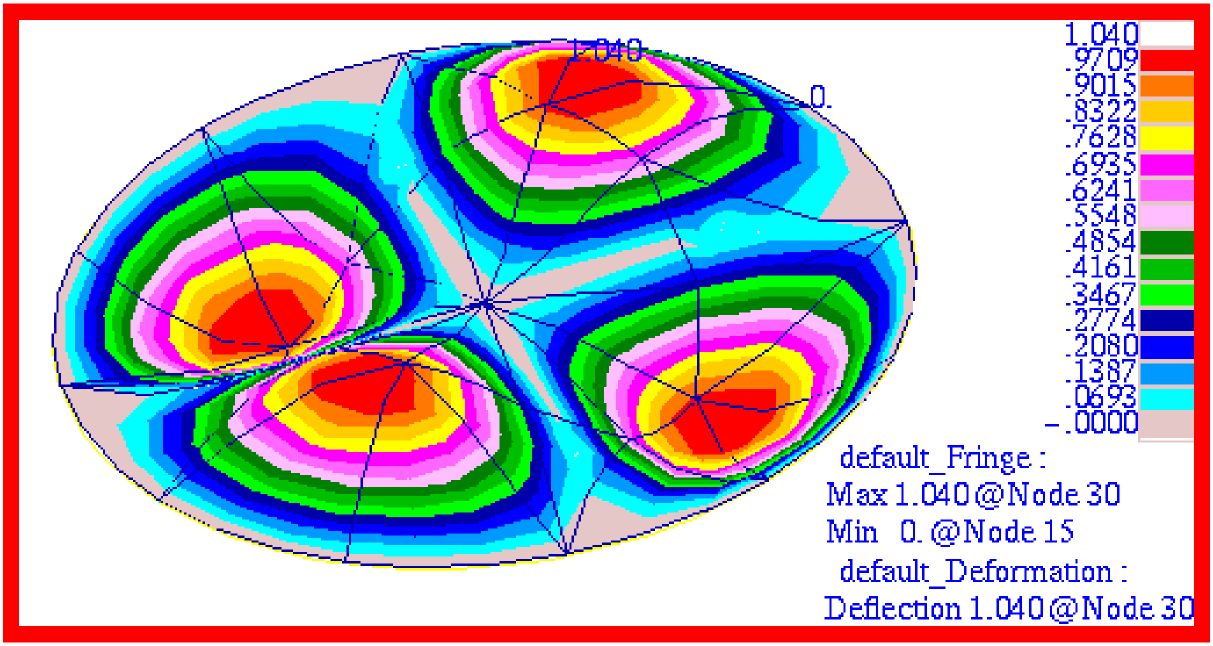


Figure 15-18 Mode 4, f_{20}, of p-Element Membrane Model.

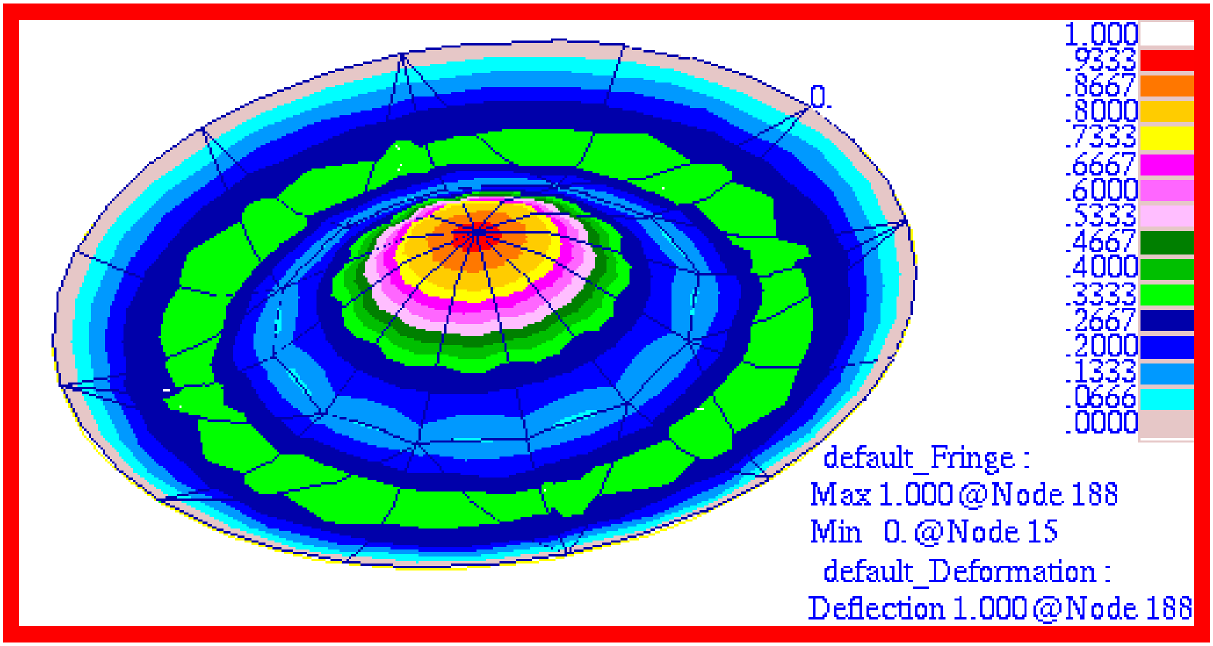


Figure 15-19 Mode 6, f_{01}, of p-Element Membrane Model.

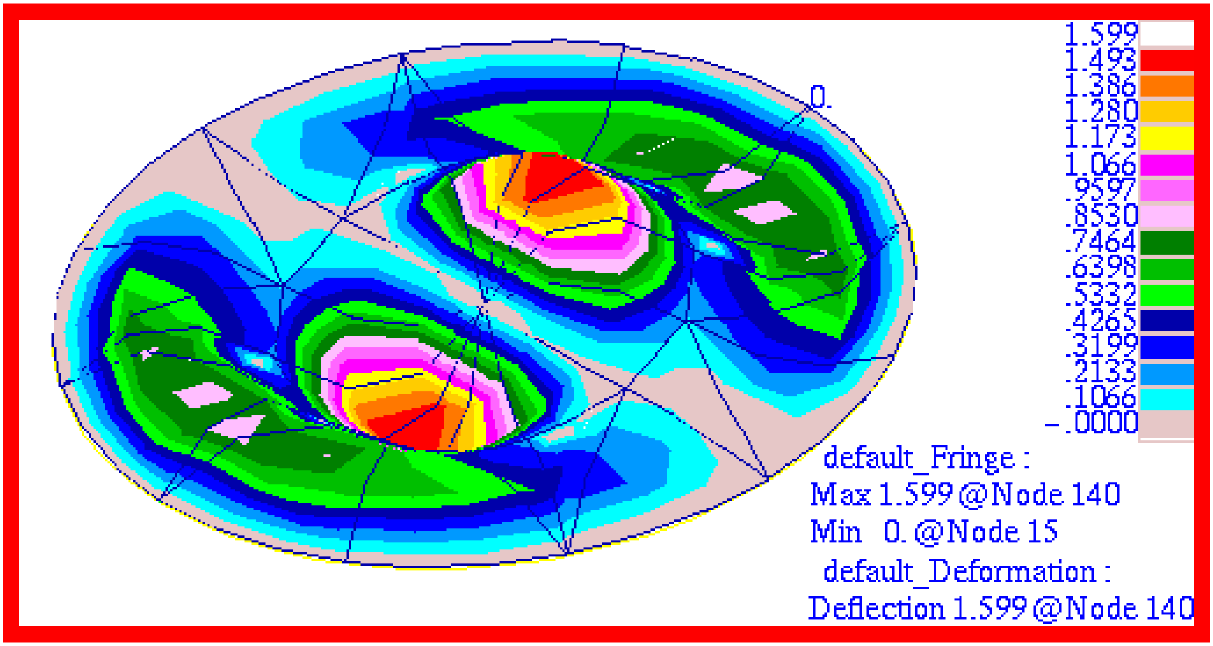


Figure 15-20 Mode 9, f_{11}, for Membrane p-Element Model.

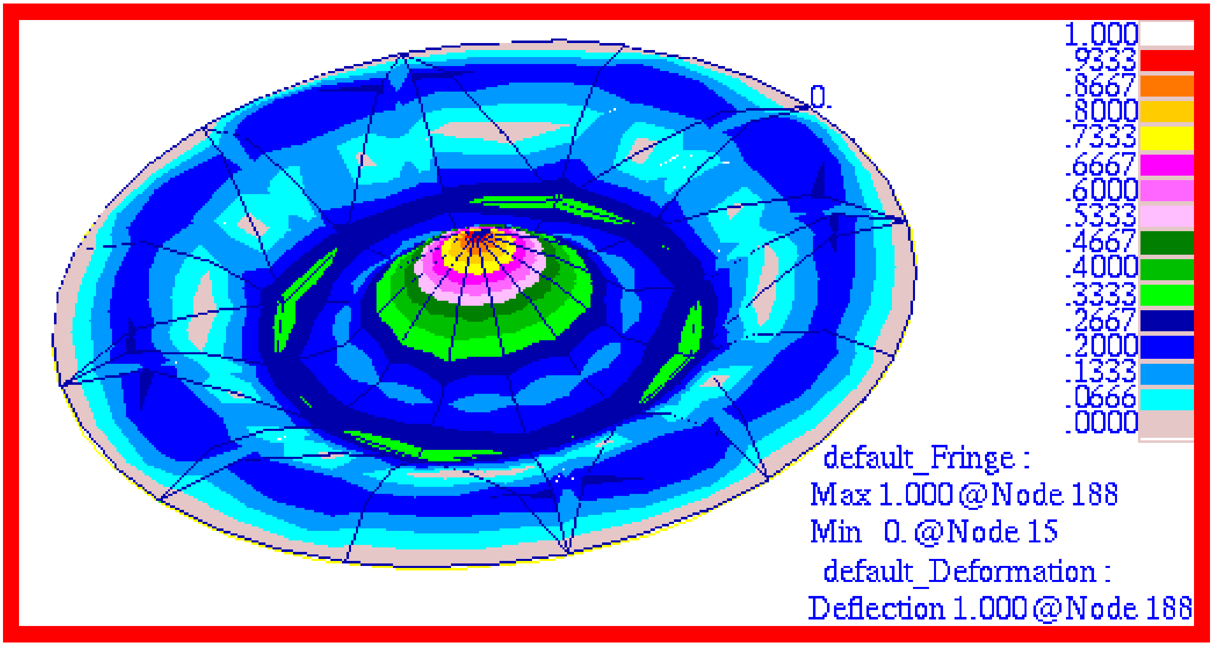


Figure 15-21 Mode 15, f_{02}, of p-Element Membrane Model.

Problem 17: Buckling, shells and Cylindrical Coordinates

Solution/Element Type:

MSC.Nastran, Buckling, Solution 105, CTRIA3, CQUAD4 Elements with Standard Formulation.

Reference:

Roark, R.J., and Young, W.C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill, Book Company, 1975, p. 556.

Problem Description:

Find the critical buckling pressure for a thin wailed cylinder with closed ends subjected to a uniform external pressure both laterally and longitudinally. Assume the endcaps and cylinder have the same thickness and that the ends are held circular.

Engineering Data:

$$E = \text{elastic modulus} = 1 \times 10^7 \text{ psi}$$

$$\nu = \text{Poissons Ratio} = .33$$

$$l = \text{length} = 20 \text{ inches}$$

$$r = \text{radius} = 5.0 \text{ inches}$$

$$t = \text{thickness} = .125 \text{ inches}$$

$$q = \text{pressure}$$

Theoretical Solution:

For a thin walled cylinder with closed ends subjected to a uniform external pressure both axially and longitudinally, the critical buckling pressure is given by the following expression:

$$q_{crit} = \frac{E\frac{t}{r}}{1 + \frac{1}{2}\left(\frac{\pi r}{nl}\right)^2}\left\{\frac{1}{n^2\left[1 + \left(\frac{nl}{\pi r}\right)^2\right]^2} + \frac{n^2 t^2}{12r^2(1-\nu^2)}\left[1 + \left(\frac{\pi r}{nl}\right)^2\right]^2\right\}$$

where n equals the number of lobes formed by the tube when it buckles.

Upon substitution of the engineering design parameters into the preceding equation, it can be shown that the minimum buckling pressure corresponds to a three lobed buckling mode shape. This occurs at a critical pressure equal to:

$$q_{crit} = 256.23 \text{ psi}.$$

MSC.Nastran Results:

To compute the critical buckling pressure for the cylinder, the model shown in Figure 15-22 was generated using MSC.Patran. The lateral sides of the cylinder were meshed with CQUAD4 elements while owing to the curvilinear geometry, the endcaps were meshed with CTRIA3 elements to maximize accuracy. Due to symmetry, only half of the cylinder was modeled with the appropriate symmetry boundary conditions being applied along the free edges. Additional symmetry boundary conditions were applied along the midplane of the cylinder. Also, to prevent recovery of any modes associated with buckling of just the endcaps, the center of each cap was restrained against any motion in either the radial, tangential or axial directions.

A buckling calculation was performed using MSC.Nastran. The critical buckling pressure was predicted to be:

Table 15-6 Critical Buckling Pressure

Theory	256.23psi
MSC.Nastran	270.18psi
%, Difference	5.44%

The corresponding mode shape was plotted with MSC.Patran and is shown in Figure 15-23. The mode shape clearly shows two distinct lobes present on the half of the cylinder that was modeled, with an additional lobe present on the other half that was not included due to symmetry.

File(s):`<install_dir>/results_vv_files/prob017.bdf, prob017.op2`

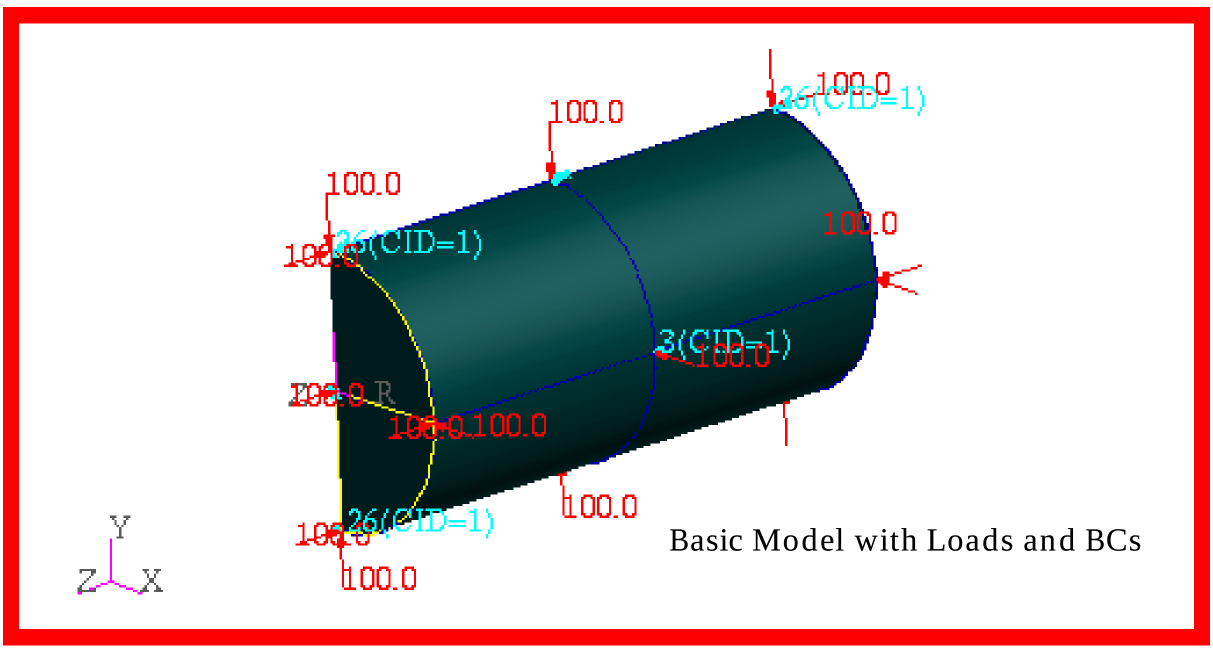


Figure 15-22 Cylinder Buckling Model with Loads and BCs.

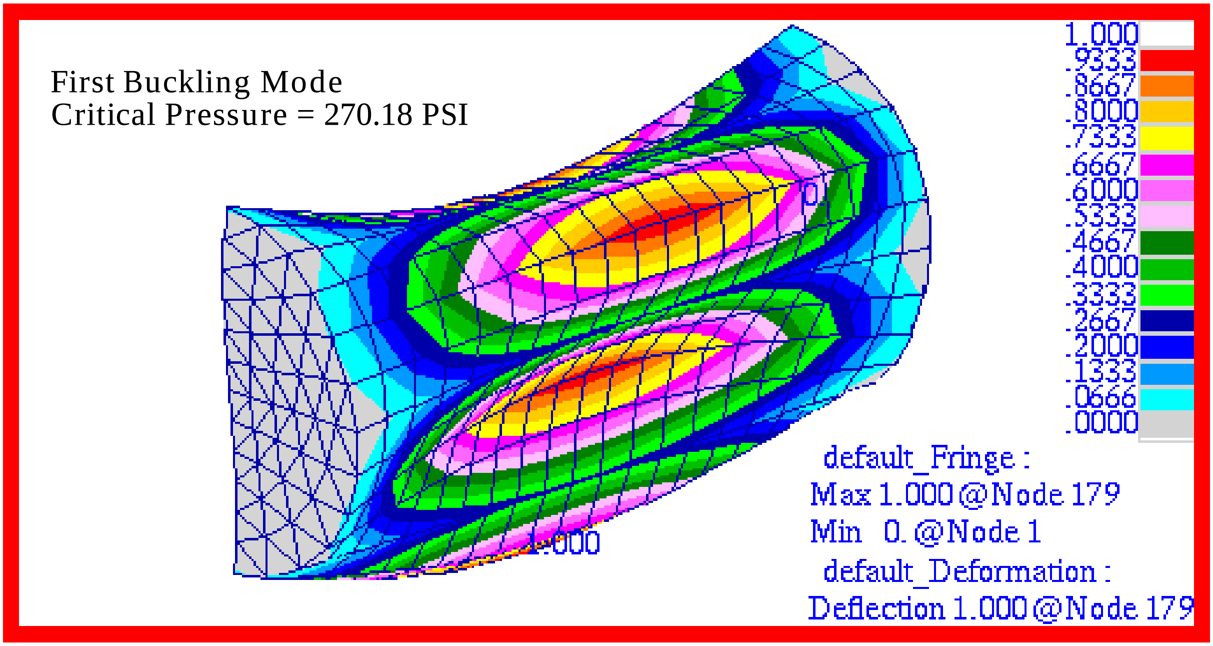


Figure 15-23 Three Lobed Mode Corresponding to Critical Pressure.

Problem 18: Buckling, Flat Plates

Solution/Element Type:

MSC.Nastran, Buckling, Solution 105, CQUAD4 with Standard Formulation.

Reference:

Roark, R. J., and Young, W. C., *Formulas For Stress and Strain*, 5th ed., McGraw-Hill Book Company, 1975, p. 551.

Problem Description:

A simply supported flat rectangular plate is subjected to a uniform compressive edge load. Find the critical buckling edge load and corresponding buckling mode shape.

Engineering Data:

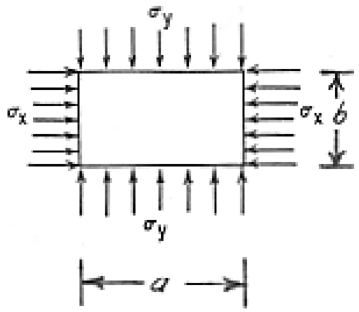
E = elastic modulud = 1×10^7 psi

ν = Poissons Ratio = .33

t = thickness = .1 inches

a = length = 16 inches

b = width = 4 inches



Theoretical Solution:

For a simply supported plate subjected to edge pressures σ_x and σ_y, the critical edge pressure required to induce buckling is given by:

$$\sigma_{x,\,crit}\,\frac{m^2}{a^2} + \sigma_{y,\,crit}\,\frac{n^2}{b^2} = 0.823\frac{E}{1-\nu^2}\,t^2\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2$$

Here m and n signify the number of half-waves in the buckled plate in the x and y directions respectively. Assuming that:

$$\sigma_{x,\,crit} = \sigma_{y,\,crit} = \sigma^*$$

and:

$$m = n = 1$$

yields:

$$\sigma^* = 0.823\,\frac{E}{1-\nu^2}\,t^2\left(\frac{1}{a^2}+\frac{1}{b^2}\right)$$

Substitution of the design parameters for the plate yields a critical edge pressure of:

$$\sigma^* = 6133.13 \text{ psi}$$

which corresponds to a distributed edge load of 613.313 lbs / inch.

MSC.Nastran Results:

To compute the critical buckling load and mode shape for the plate, the model shown in **Figure 15-24** was created using MSC.Patran. Due to symmetry, only one-fourth of the plate was modeled with the appropriate boundary conditions being applied along the planes of symmetry. In addition to further enforce recovery of only the lowest mode, additional boundary conditions were imposed so that there were no surface rotations about axes located in either plane of symmetry. This would ensure that only symmetrical mode shapes were calculated. Lastly, compressive edge pressures were modeled as distributed edge loads equal to the edge pressure multiplied by the plate thickness. Using this model, MSC.Nastran calculated the critical edge pressure to be:

Table 15-7 Critical Edge Pressure

Theory	$\sigma^* = 6133.13$ psi
MSC.Nastran	$\sigma^* = 6133.25$ psi
%, Difference	0.002%

The corresponding buckling mode shape that was plotted using MSC.Patran is shown in **Figure 15-25**. The mode shape plot clearly reveals the presence of a half-sine wave in either the x or y directions, which would correspond to $m = n = 1$.

File(s):`<install_dir>/results_vv_files/prob018.bdf, prob018.op2`

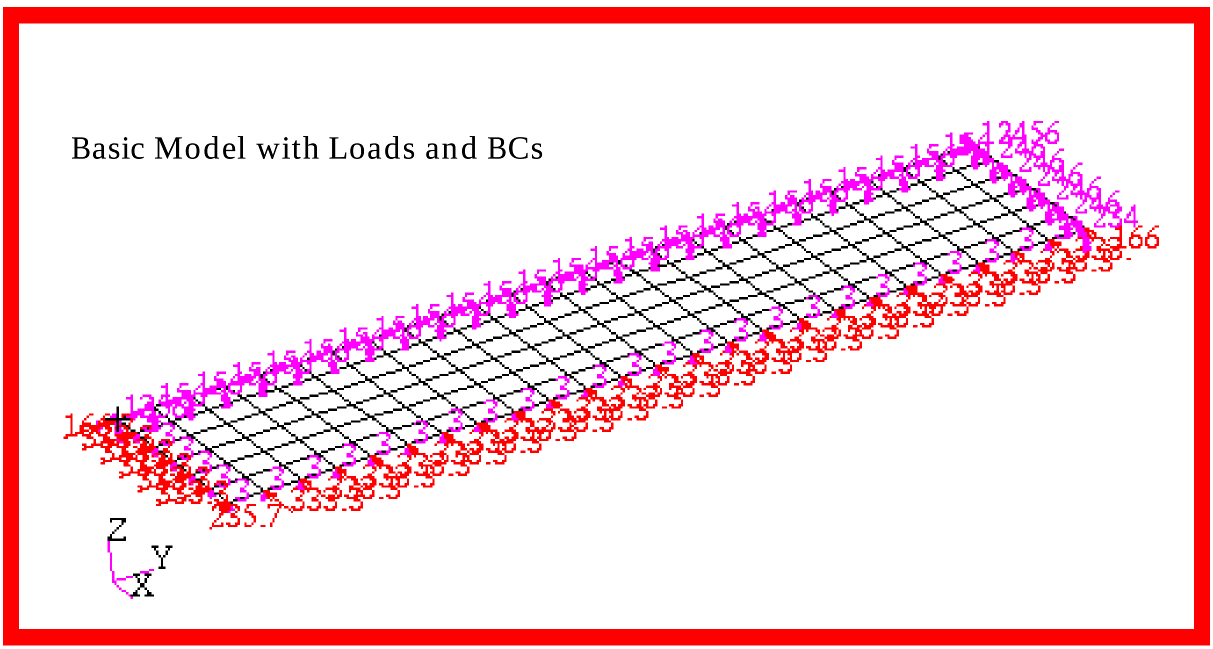


Figure 15-24 Plate Buckling Model with Load and BCs.

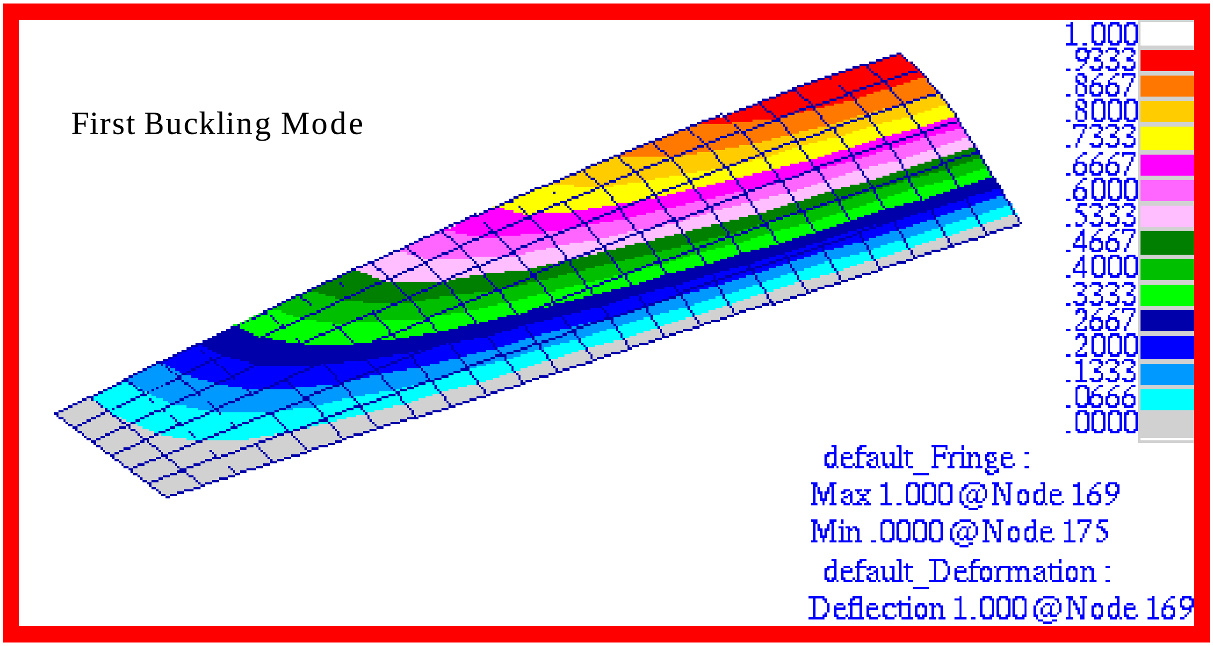


Figure 15-25 Buckling Mode Shape of Plate due to Critical Pressure.

Problem 19: Direct Transient Response, Solids and Cylindrical Coordinates

Solution Type:

Direct Linear Transient Response, Solution 109

Element Type:

CHEX8, Standard Formulation

Reference:

Crandall, S. H., Dahl, N.C., and Lardner, T.J., *An Introduction to the Mechanics of Solids*, 2nd ed., Mc-Graw Hill Book Company, 1972, pp. 293-297.

Problem Description:

A thick walled cylinder is subjected to a sudden internal pressure of 10 psi. Find the peak radial displacement in the cylinder due to the sudden application of the pressure.

Engineering Data:

r_i = 6.0 inches

r_0 = 12.0 inches

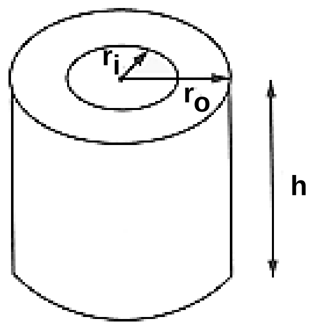
h = 8.0 inches

E = 30.0 x 10^6 psi

$\rho = .28 \text{ lbs} / \text{in}^3$

$\nu = 0.0$

P = 10.0 psi



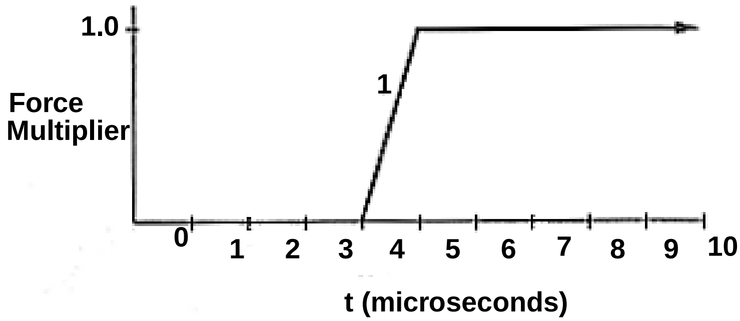
Theoretical Results:

For a single degree-of-freedom system, an instantaneously applied load will give a maximum displacement that is twice the static displacement at the same applied loading.

MSC.Nastran Results:

To determine the transient response of the cylinder, the model shown in **Figure 15-26** was generated using MSC.Patran. Due to symmetry, only a 15 degree sector of the cylinder was modeled with the appropriate axisymmetric boundary conditions applied to the lateral faces of the model. The model was meshed entirely with CHEX8 solid elements using a standard formulation.

For the purposes of this analysis, the pressure was applied in an instantaneous manner as shown below. Several time steps of zero load were input to ensure simulation of a sudden step function in the loading. The integration time step was chosen to be 1.0 microsecond. To prevent numerical instability, the applied loading was ramped over a single time step. Approximately three cycles of the response were simulated and structural damping was neglected with the radial displacement being recovered at three radial positions situated at the inner diameter, midway through and at the outer diameter of the cylinder.



Using the same model, a static analysis was performed with a constant internal pressure of 10.0 psi. The results for both the static and transient analysis are summarized below.

Table 15-8 Transient vs. Static Results

Position	Static Displacement $(x10^{-6})$	Peak Transient Displacement $(x10^{-6})$	Ratio Transient/Static
r_i	3.325737	6.534348	1.965
r_{mean}	2.773202	5.534836	1.996
r_0	2.662867	5.322965	1.999

An XY Plot that was generated by MSC.Patran of the transient radial displacement at inner, mean, and outer radius is shown in **Figure 15-28**. In this figure nodes 120, 123 and 126 correspond to the inner, mean and outer radius, respectively. The results clearly show the expected oscillatory behavior characteristic of an undamped response. The corresponding static results are shown in **Figure 15-29** where the radial displacement has been plotted against radius.

A fringe plot of the static radial displacements is shown in **Figure 15-27**. Both the fringe and xyplots clearly show peak values that agree with he preceding MSC.Nastran results.

File(s):<install_dir>/results_vv_files/prob019_T.bdf, prob019_T.op2, prob019_S.bdf, prob019_S.op2

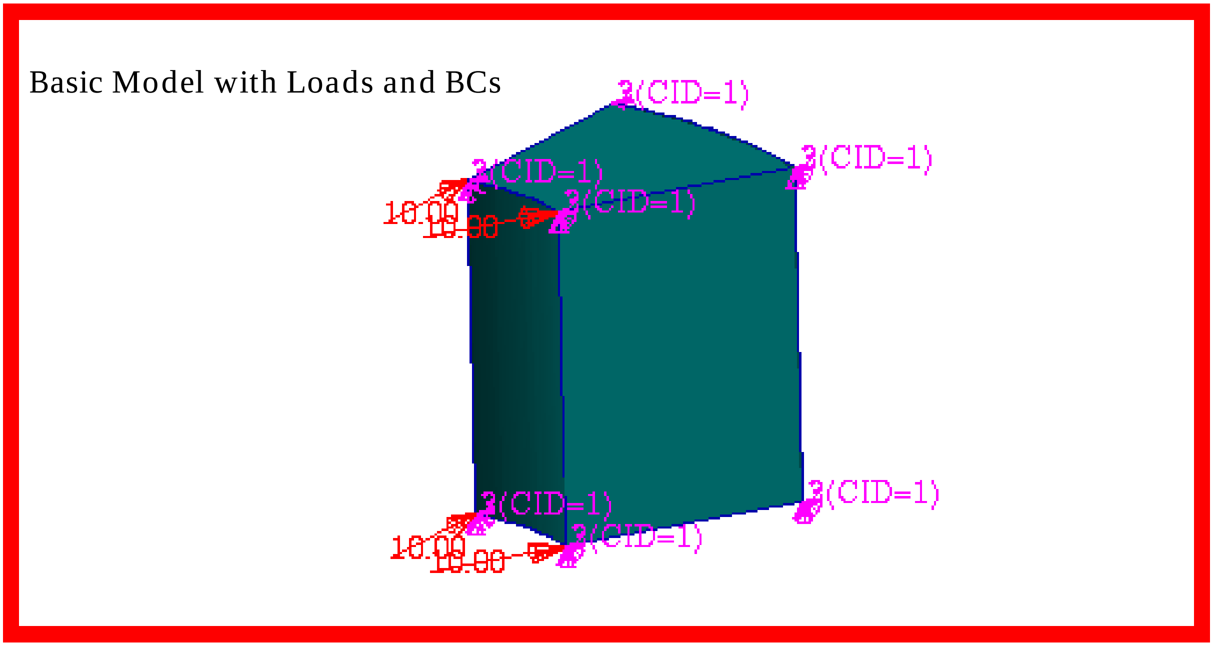


Figure 15-26 Transient Dynamic Model of Cylinder.

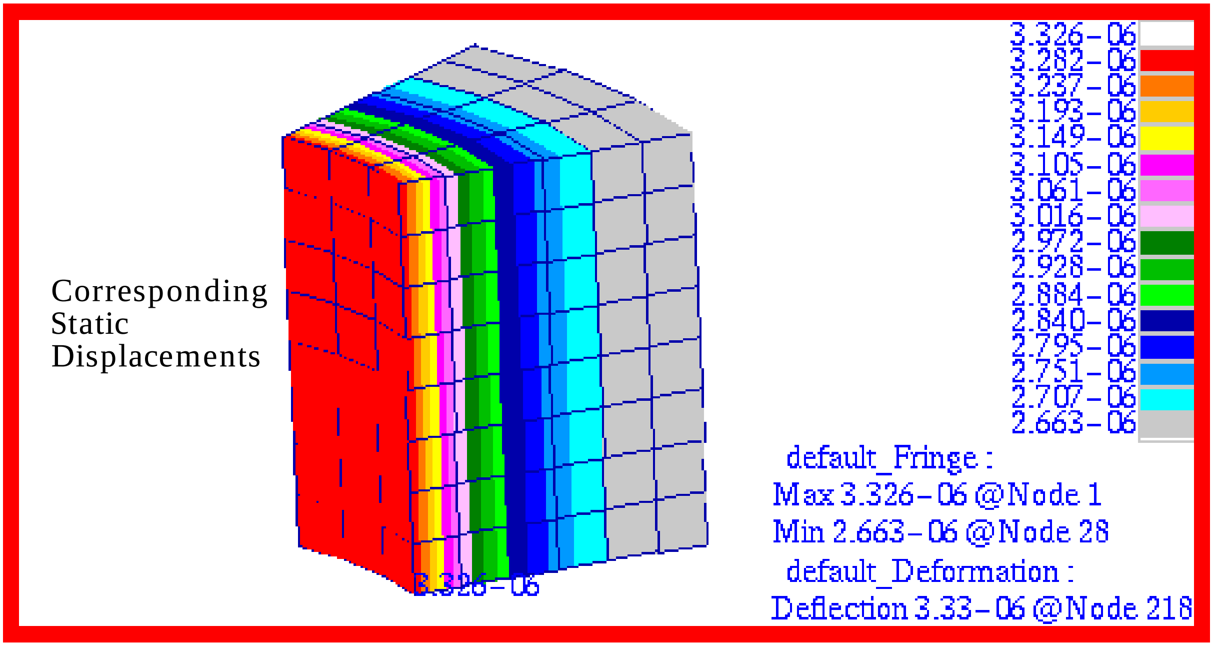


Figure 15-27 Corresponding Static Results of Displacement.

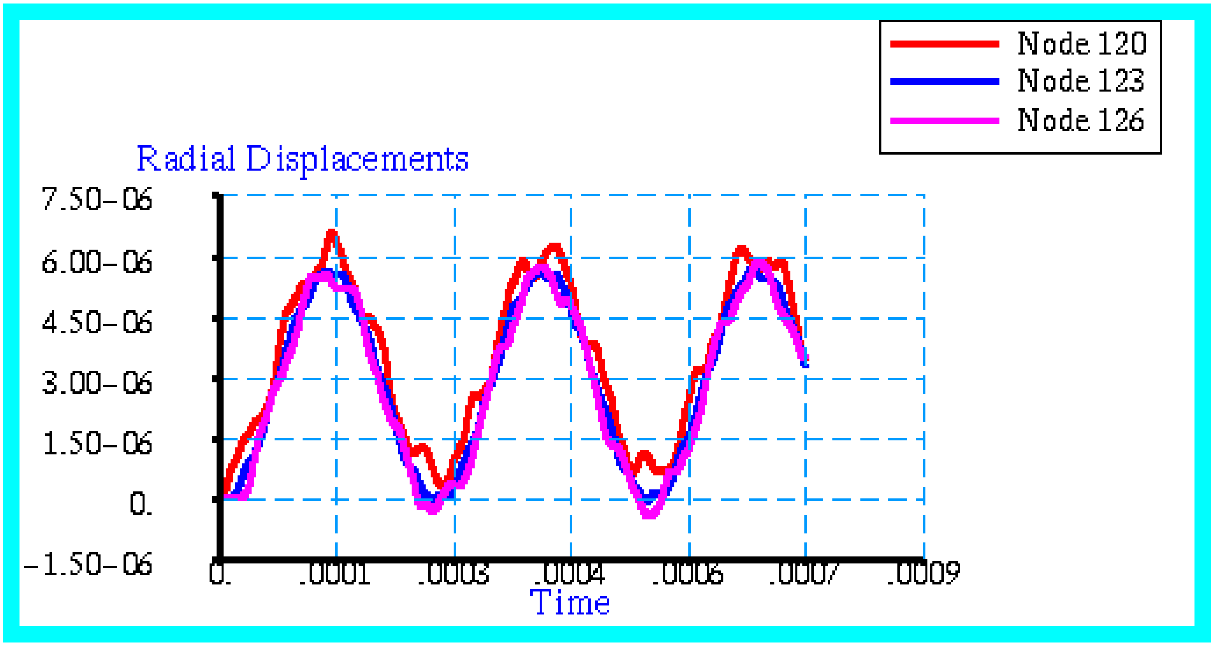


Figure 15-28 Displacement Responses at Various Nodes.

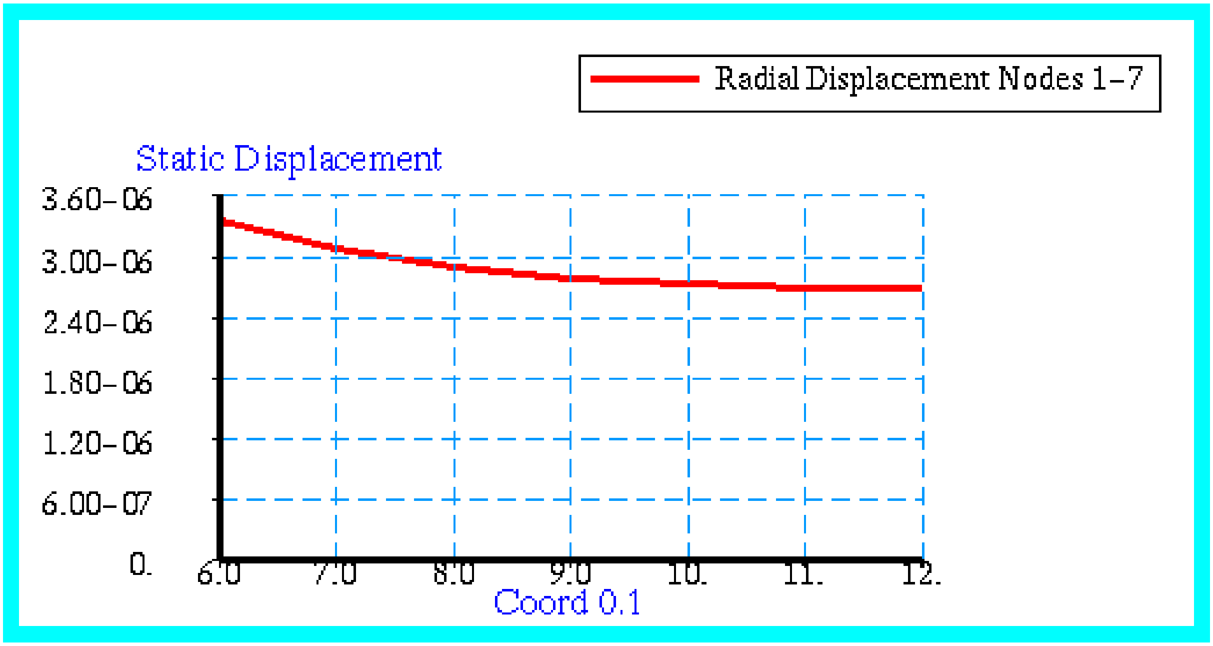


Figure 15-29 Corresponding Static Displacements Across Thickness.

Problem 20:Modal Transient Response with Guyan Reduction and Bars, Springs, Concentrated Masses and Rigid Body Elements

Solution/Element Type:

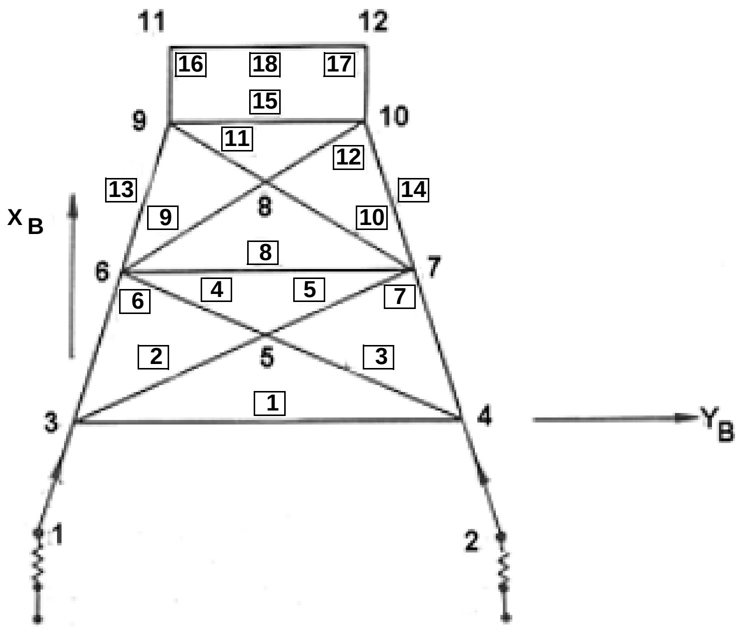
MSC.Nastran, Modal Transient Response, Solution 112, CBAR, CONM2, CMASS1, CELAS1, RBE2.

Reference:

MSC.Nastran *Demonstration Problem Manual*, Version 65

Problem Description:

The transient response is required of a two dimensional truss mounted on an elastic foundation and subjected to a time dependent base excitation. The truss is shown below and consists of bars of varying cross-section with localized concentrated masses. The horizontal acceleration that is applied to the base is shown plotted in **Figure 15-30**.



Engineering Data:

Table 15-9 Bar Properties Foundation Stiffness

Bar ID	Area	I1
1	36.91371	2549.353
2 through 5	30.63052	1456.865
6,7,13,14	266.83900	47701.650
8	30.63052	1456.865
6 through 12	24.34734	731.942
15, 18	147.65490	40789.650
16, 17	162.57740	24234.200
19, 20	537.21123	22965.830

Table 15-10 Foundation Stiffness

Direction	Stiffness
Vertically	23.0×10^{5}
Horizontally	1.0×10^{5}
Rotationally	15.0×10^{9}

Table 15-11 Concentrated Masses

Location (Point ID)	Mass
15, Horizontal	8.0×10^{9}
15, Vertical	8.0×10^{9}
15, Rotational	4.0×10^{15}
3, 4, 6, 7	388.1988
5, 8	25.8799
9, 10	1035.197
11, 12	2070.393

Theoretical Solution:

For this particular problem there exists no simple closed form solution. The only real means of ascertaining the accuracy of a MSC.Nastran analysis model is to ensure that the base acceleration matches the intended profile of the applied excitation. This is a basic check that should always be performed, especially when using the large mass method to simulate an enforced base motion.

MSC.Nastran Results:

To calculate the transient response for the truss, the model shown in **Figure 15-31** was generated using MSC.Patran. The model was comprised of simple CBAR elements and CONM2 elements that were used to model any localized masses, with the exception of the base point (i.e., Node 15). To model the base, three large grounded scalar mass CMASS1 elements were used, one each for the horizontal, vertical and rotational directions. The magnitude of these masses were chosen to be approximately 10^{9} times as great as the mass of the truss. By doing so, this would ensure that the large mass of the base would dominate so that the intended base excitation was properly applied to the entire truss. A single RBE2 rigid body element was used to attach the base node to the ends of the CELAS1 elements that were used to model the elastic foundation.

In performing the analysis, the modal transient method was used along with automated Guyan reduction. The transient response was based upon the first 30 modes which spanned a frequency range that extended to approximately 100 hertz. This was deemed more than acceptable, given the periodicity of the applied base acceleration profile. In addition, it was assumed that throughout this entire frequency range, the truss was 10 percent critically damped.

The resultant acceleration profile that was predicted by MSC.Nastran at the base point is shown plotted in **Figure 15-32**. Here the horizontal acceleration at node 15 has been plotted versus time using the results xyplot utility of MSC.Patran. It can clearly be seen that the acceleration at Node 15 closely matches the intended base excitation which was previously shown in **Figure 15-30**. The acceleration at other selected nodes on the model are shown plotted in **Figure 15-33**. These acceleration profiles appear to show the correct response. Namely, the occurrence of any localized peaks in the base acceleration are matched by localized peaks in the acceleration of the truss. Similarly, the magnitude of the bar forces and stress response plots, shown in **Figure 15-34** and **Figure 15-35**, exhibit the same behavior.

File(s):`<install_dir>/results_vv_files/prob020.bdf, prob020.op2`

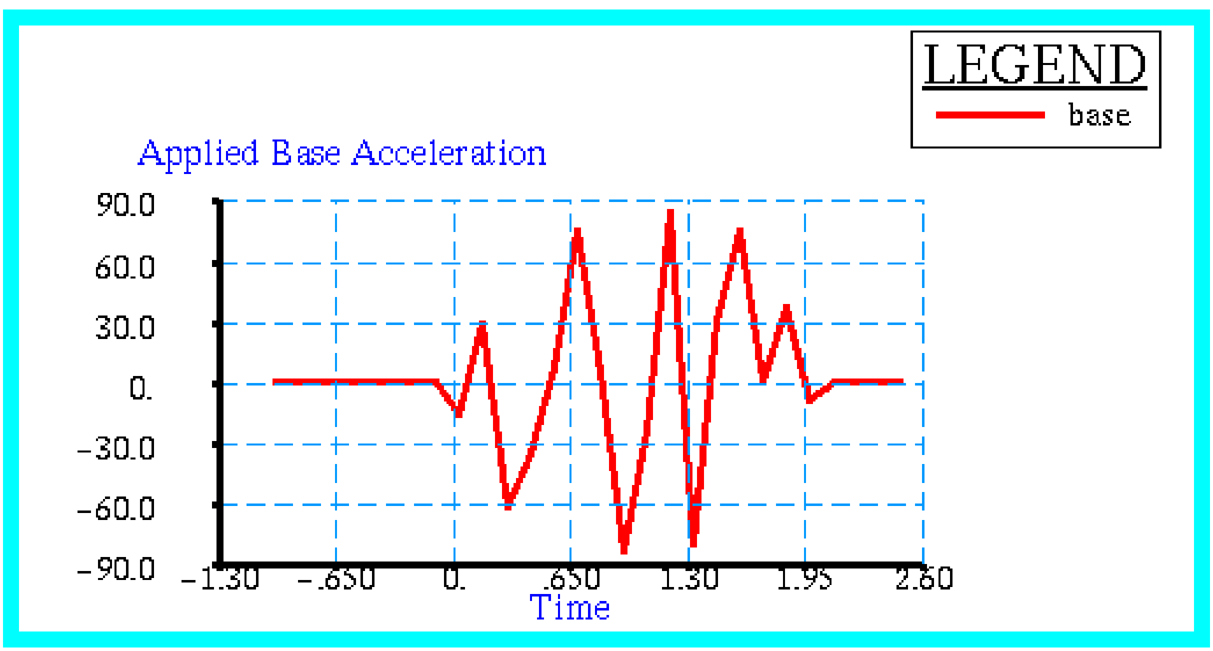


Figure 15-30 Horizontal Acceleration Applied to Truss.

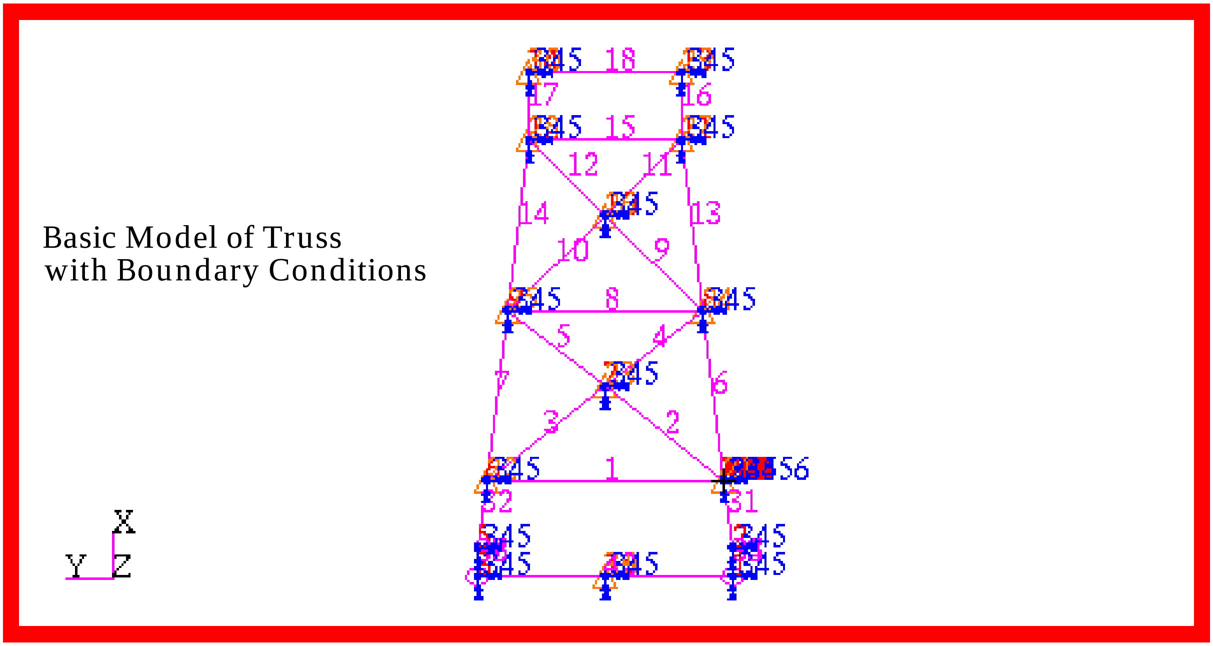


Figure 15-31 Basic Model of 2D-Truss Structure.

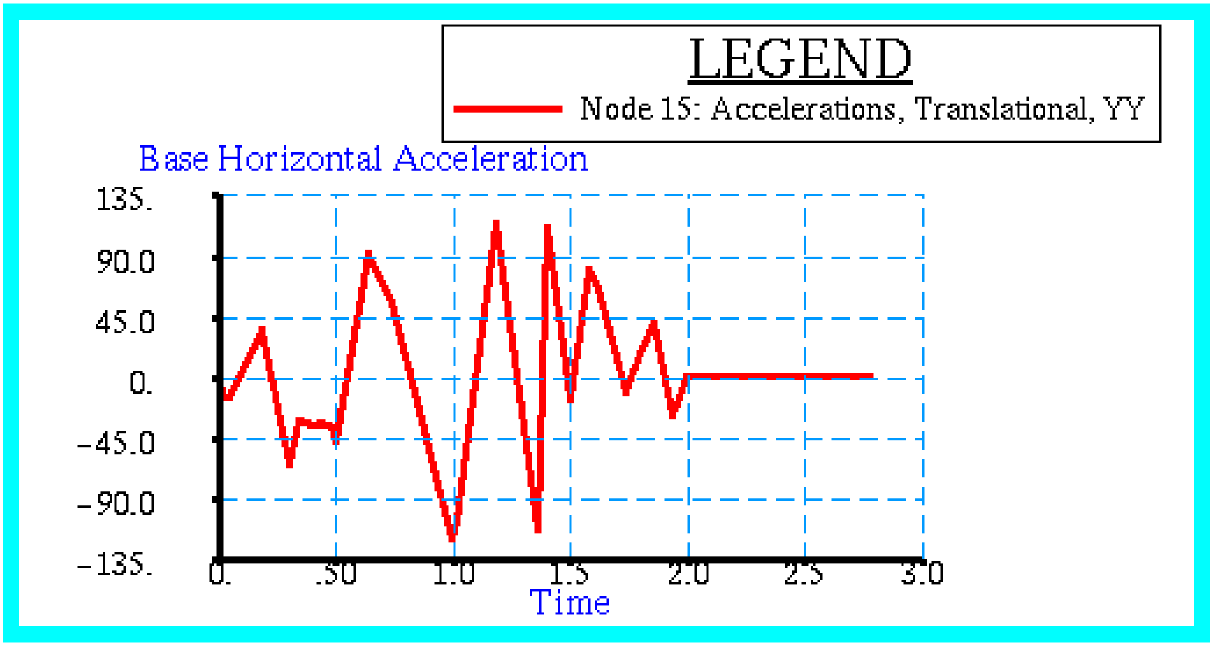


Figure 15-32 Horizontal Acceleration Profile at Base (Node 15).

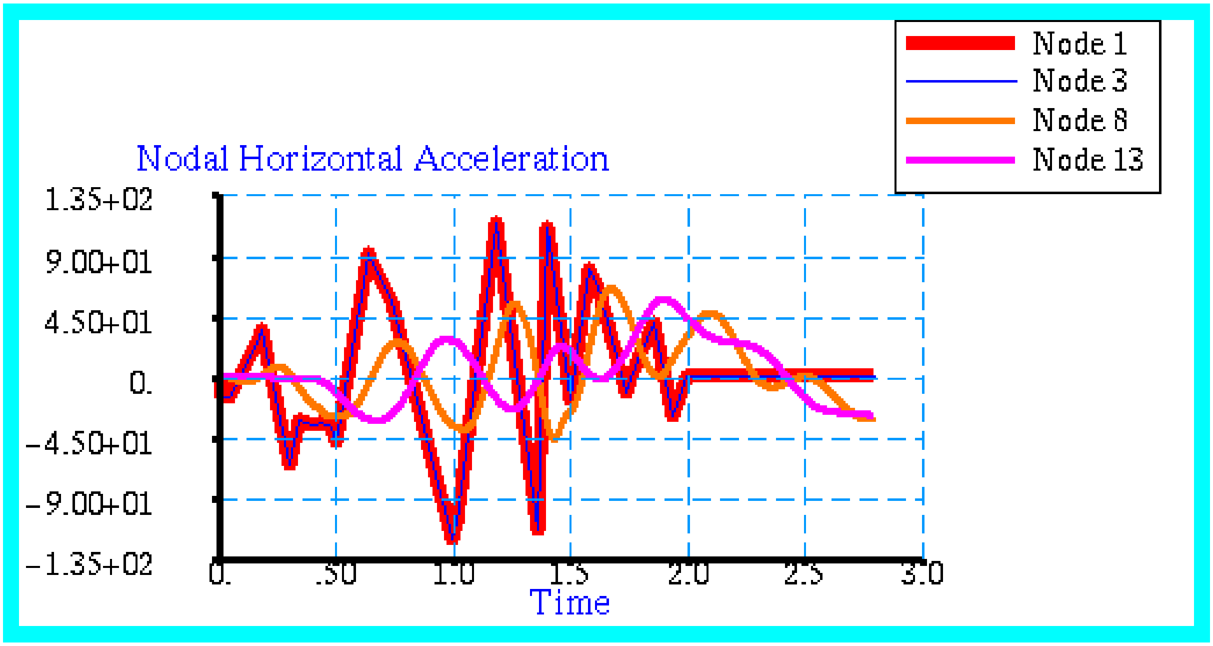


Figure 15-33 Horizontal Acceleration at Selected Points of 2D-Truss.

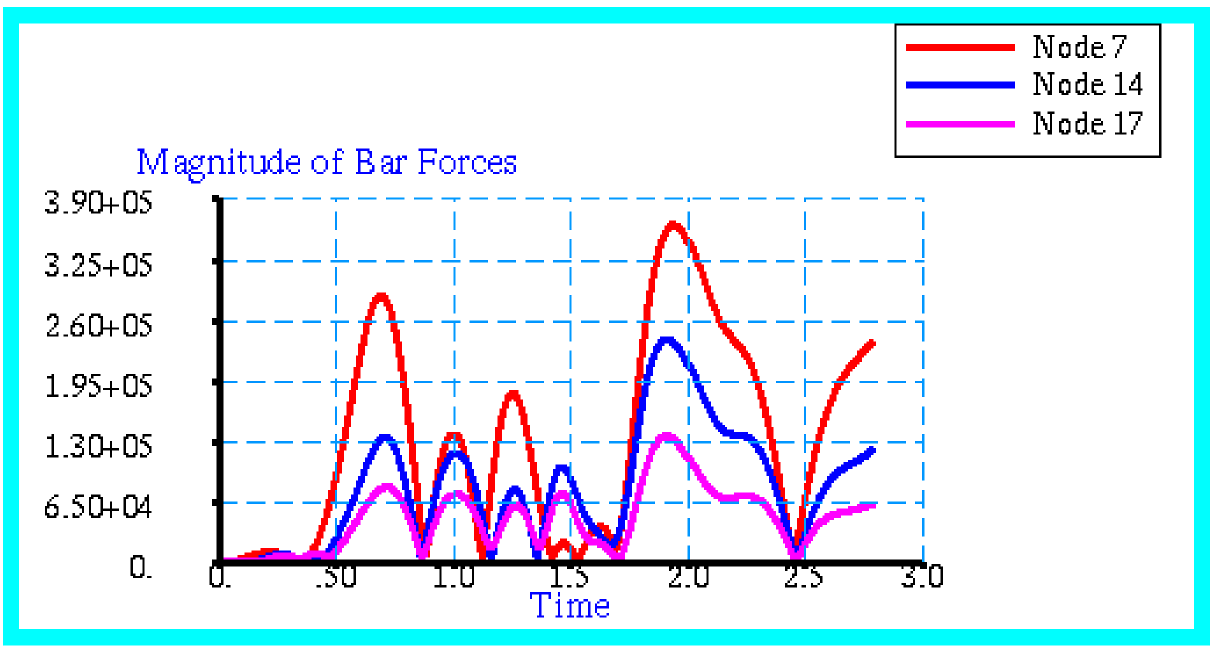


Figure 15-34 Bar Force Responses at Selected Points in 2D-Truss.

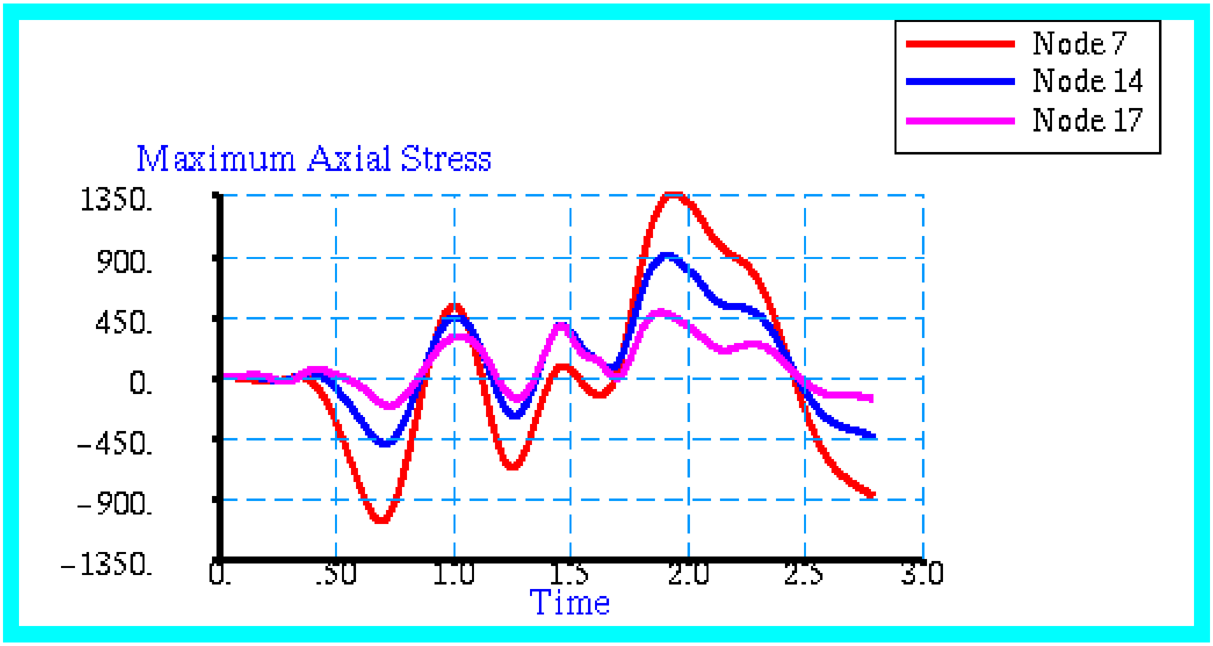


Figure 15-35 Stress Responses at Selected Points in 2D-Truss.

Problem 21: Direct Nonlinear Transient, Stress Wave Propagation with 1D Elements

Solution/Element Type:

MSC.Nastran, Direct Nonlinear Transient, Solution 129, CROD Elements

Reference:

Timoshenko, S., and Goodier, J. N., *Theory of Elasticity*, 2nd ed., McGraw-Hill Book Co., 1951, pp. 492-496.

Juvinall, R.C., *Engineering Consideration of Stress, Strain and Strength*, McGraw-Hill Book Co., 1967, pp. 185-188.

Problem Description:

A rod of uniform cross section is fixed at one end and a constant force is suddenly applied to its free end. A stress wave results that propagates along the length of the rod. Determine the stress history at both the free and fixed ends of the rod as well as the displacement history at the free end.

Engineering Data:

L = rod length = 100.0 m

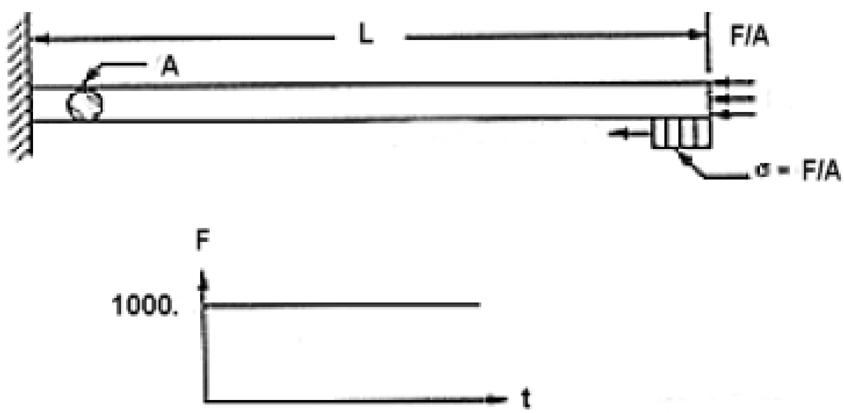
A = cross-sectional area of the rod = 1.0 m^2

E = elastic modulus = 103 N/m^2

ν = Poissons Ratio = 0.3

ρ = mass density = 0.1 kg/m^3

$c = \sqrt{E/\rho} = 100.0$ m/sec = stress wave velocity



Force vs. Time

Theoretical Solution:

Due to the constant force, F, applied at the free end of the rod, the stress at the free end remains constant and is simply equal to

$$\sigma = -\frac{F}{A} = -1000.0 \ N/m^2$$

Before the stress wave first reaches the fixed end, the stress at the fixed end is zero. As soon as the stress wave arrives at the fixed end, it is reflected and produces a stress equal to 2σ at the fixed end. The stress remains unchanged until the wave front returns to this point. The compressive wave leaves the fixed end and upon reaching the free end it is once again reflected, but this time as a tensile wave. When this tensile wave arrives at the fixed end and is reflected, the stress at the fixed end becomes zero. This zero stress state will again remain unchanged for another complete cycle of the stress wave motion.

The displacement time history at the free end can be expressed as

$$u(t) = \int_0^L \frac{\sigma(x, t)}{E} dx$$

Or, expressing u(t) separately for each time interval

$$, u(t) = \frac{-FL}{AE} t \text{ for } 0 \le t \le 2$$

$$u(t) = u(t = 2) + \frac{-FL}{AE}(t - 2) \text{ for } 2 \le t \le 4$$

etc.

Thus, the displacement at the free end is a linear function of time during each interval it takes the stress wave to completely traverse the rod.

MSC.Nastran Results:

To determine the stress and displacement history at the opposing ends of the rod, the model shown in **Figure 15-36** was generated using MSC.Patran. The model consisted of 20 CROD elements. One end was fixed while the remainder of the rod was constrained to motion in the x, or axial, direction only. The upper bound stability limit for the integration time increment was determined by:

$$\Delta t = \frac{1}{4(\omega_n)_{max}}$$

where $(\varpi_n)_{max}$ is the highest natural frequency in cycles per second of all of the modes of interest in the model. In this instance, this would mandate that modal frequencies be at least greater than the frequency associated with one complete cycle for the stress wave to completely traverse the rod, or 0.5 cycles per second. For the purposes of this analysis, an integration time increment of 0.025 seconds was used. In addition, a slight amount of damping was introduced to reduce the overall occurrence of any oscillations in the solution.

Stress history plots that were made with MSC.Patran of the MSC.Nastran results are shown in **Figure 15-37** and **Figure 15-38** for the fixed and free ends of the rod, respectively. The displacement history at the free end of the rod is shown plotted in **Figure 15-39**. Despite the presence of the damping, the results do exhibit some slight oscillations, especially whenever a stress wave is reflected as well as upon initial application of the loading. However, these oscillations are rapidly attenuated. Overall, the results demonstrate the correct periodicity of the stress and displacement at both the fixed and free ends of the rod.

Table 15-12 Maximum Displacement, t=2 Seconds

Theory	200.00 m
MSC.Nastran	200.98 m
%, Difference	0.49%

Table 15-13 Maximum Stress at Fixed End, 1<t<3 Seconds

Theory	$-2000.\ N / m^2$
MSC.Nastran	$-2264.6\ N / m^2$
%, Difference	13.23%

File(s): `<install_dir>/results_vv_files/prob021.bdf, prob021.op2`

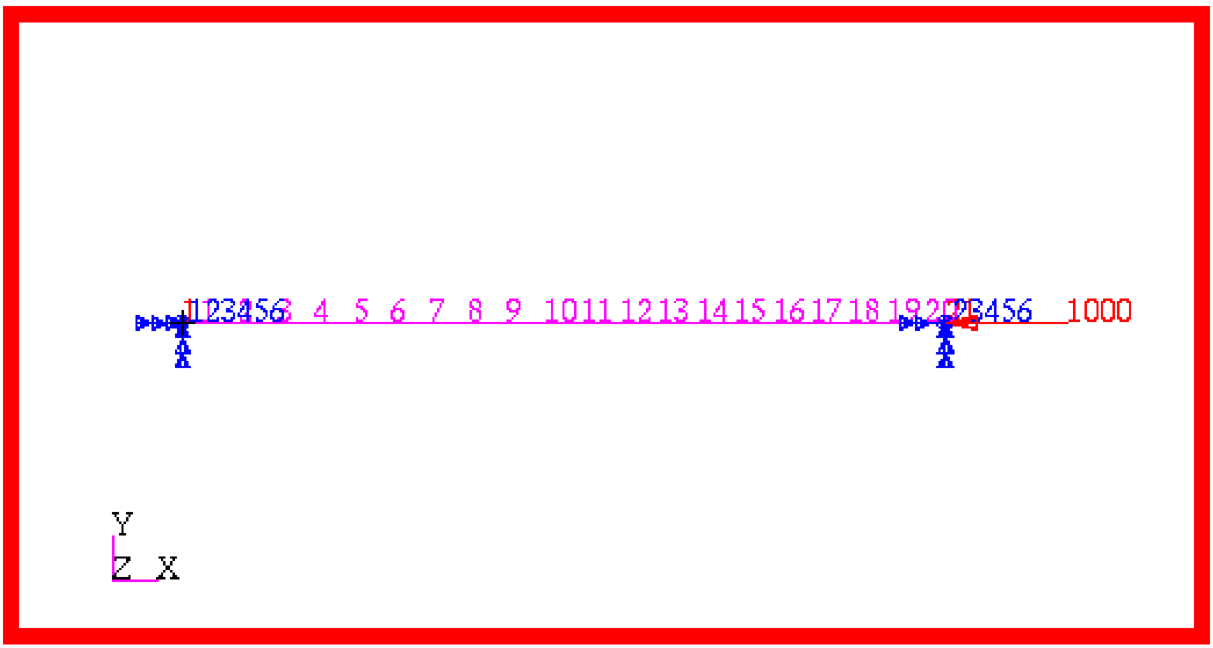


Figure 15-36 Basic CROD Model to Investigate Stress Wave.

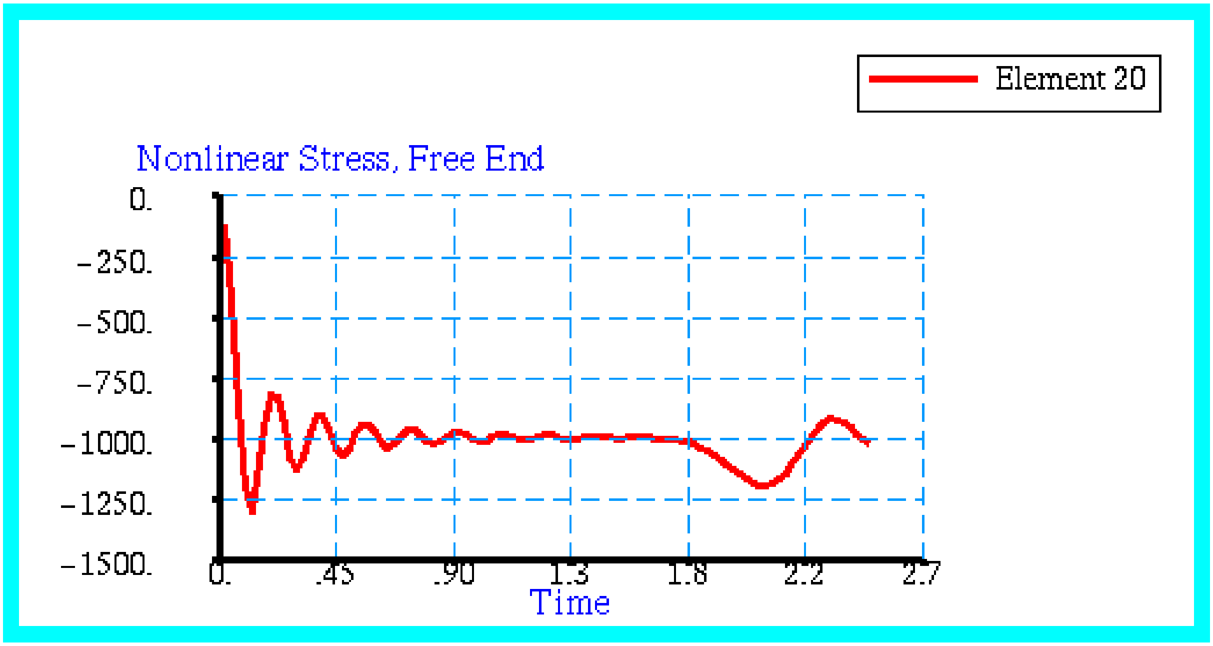


Figure 15-37 Stress History Plot at Element 20, the Free End.

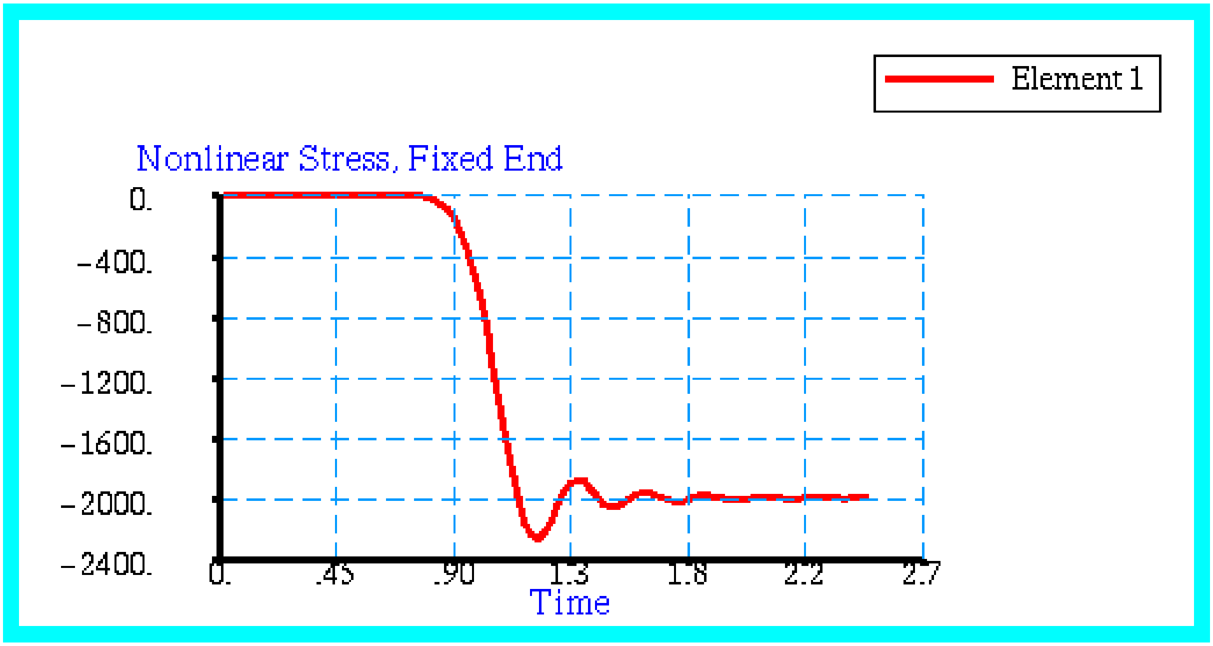


Figure 15-38 Stress History Plot at Element 1, the Fixed End.

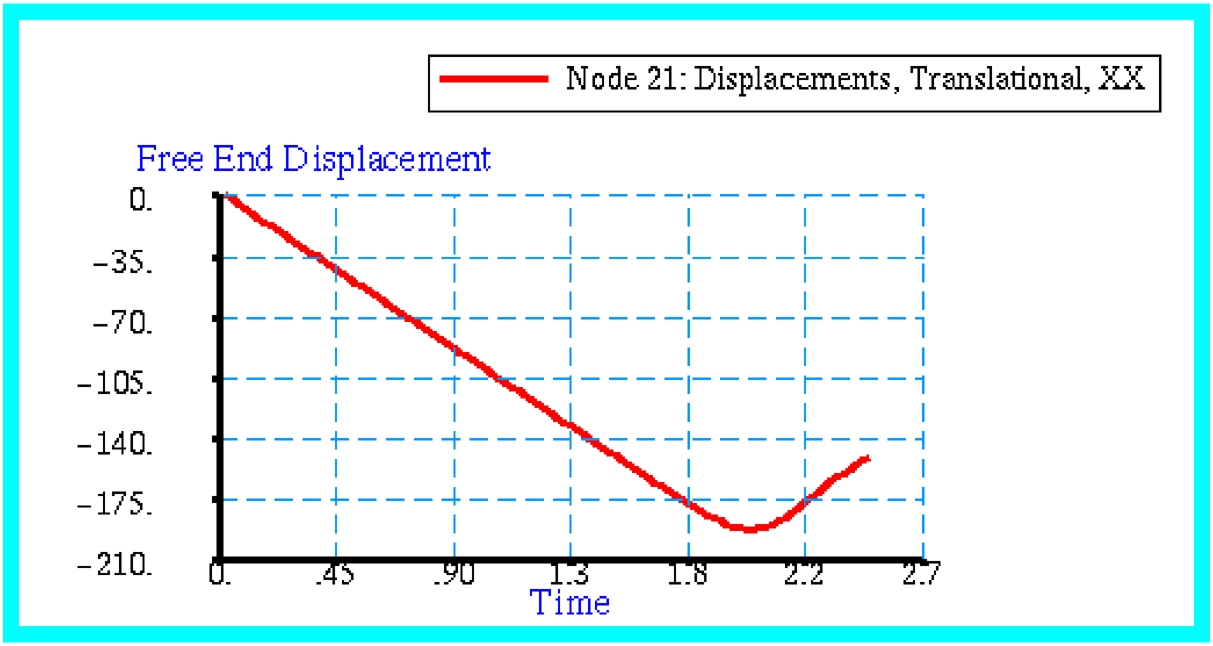


Figure 15-39 Free End Displacement of CROD Model.

Problem 22: Direct Nonlinear Transient, Impact with 1D, Concentrated Mass and Gap Elements

Solution/Element Type:

MSC.Nastran, Direct Nonlinear Transient, Solution 129, CROD, CGAP, CONM2 Elements.

Reference:

Timoshenko, S., and Goodier, J.N., *Theory of Elasticity*, 2nd ed., McGraw-Hill Book Co., 1951, pp. 497-504

April 1986 Application Note, Application Manual Section 5.

Problem Description:

A rod with a fixed end is struck by a moving mass at its other end. Let v_0 be the initial velocity of the mass prior to impact. Consider the mass of the body to be infinitely rigid and the velocity at the free end of the rod at the instant of impact to also be equal to v_0. Determine the time history of stress and displacement at the free end of the rod as well as the duration of the impact and the maximum stress in the rod.

Engineering Data:

L = rod length = 100.0 m

A = cross-sectional of the rod = 1.0 m^2

E = elastic modulus = $10^3 \ N / m^2$

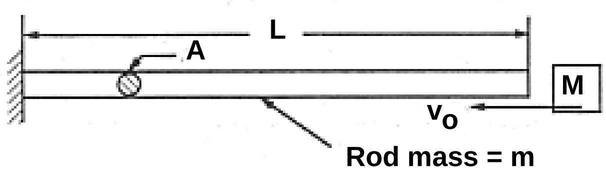
ν = Poissons Ratio = .3

$\rho = 0.1 \ kg / m^3$

m = rod mass = $\rho AL = 10.0$ kg

M = moving mass = 10.0 kg

$v_0 = -0.1 \ m / sec$



Theoretical Solution

At the instant of impact, an initial compressive stress is generated at the free end of the rod equal to:

$$\sigma_0 = v_0\sqrt{E\rho}$$

or upon substitution:

$$\sigma_0 = -1.0 \ \ N / m^2$$

After impact, the resistance of the bar will cause the velocity of the moving body to decrease and hence the pressure on the bar will decrease as well, causing a reduction in the compressive stress in the bar. Thus, there exists a compressive wave characterized by a decreasing compressive stress traveling along the length of the bar. The change in compressive stress with time can easily be determined from the equation of motion of the body. Letting σ denote the variable compressive stress at the free end of the bar and v the variable velocity of the body yields:

$$M\frac{dv}{dt} + \sigma = 0$$

or upon substituting

$$v = \frac{\sigma}{\sqrt{E\rho}}$$

yields:

$$\frac{M}{\sqrt{E\rho}}\frac{d\sigma}{dt} + \sigma = 0$$

from which

$$\sigma = \sigma_0 e^{\frac{-t\sqrt{E\rho}}{M}}$$

This equation is only valid so long as $t < 2l / \sqrt{E\rho}$, or $t < 2$ seconds. At t = 2 seconds, the compressive wave with an intensity of σ_0 returns to the free end of the bar, which is still in contact with the body. The velocity of the body cannot change suddenly. As a consequence, the stress wave will be reflected from the end of the bar, resulting in an immediate increase in the compressive stress to $2\sigma_0$. This sudden change in compressive stress occurs during impact at the end of every two second interval. However, during every subsequent interval, there will be an increasing number of stress waves either moving away or toward the struck end of the rod. The compressive stress at the free end of the rod at any instant will simply be the sum of the stress induced by the reflected waves produced during that particular interval with the stress produced by the returning waves from the previous interval. The actual peak stress produced in the rod throughout impact will occur in a different interval depending upon the ratio α of the mass of the rod to the mass of the body. For the case $\alpha = 1$, the max stress occurs at t = 2 seconds and is equal to $2\sigma_0$.

The instant when the stress at the free end of the rod becomes zero indicates the end of impact. In general the duration of the impact increases as α decreases. Calculations of Saint-Venant give the following values for impact duration:

$\alpha =$	1/6	1/4	1/2	1
Duration	7.419	5.900	4.708	3.068

MSC.Nastran Results:

To determine stress and displacement history at the free end of the rod, the model shown in **Figure 15-40** was generated using MSC.Patran. The model consisted of 20 CROD elements. The end of one CROD element was completely restrained while the remainder were constrained to motion in the x, or axial, direction only. The mass was modeled with a CONM2 concentrated mass element which was connected to the free end of the rod with a gap element having a zero initial opening. An initial velocity of $v_0 = -0.1 \text{ m/sec}$ was specified for both the CONM2 and the node situated at the free end of the rod. An integration time step of 0.025 seconds was chosen and a small damping of 0.4% was introduced to reduce any high frequency oscillations in the solution.

The MSC.Nastran results are shown in **Figure 15-41** and **Figure 15-42**. Here, the stress history and displacement history at the free end of the rod were plotted using MSC.Patran. Also shown in **Figure 15-42** is the displacement history of the mass, designated as Node 22. The results show the expected behavior; namely, an initial compressive stress of -1.0 N / m^2 that rapidly decays until 2 seconds at which time it immediately doubles and then decays to a value of zero. The time at which the stress reaches zero also correlates in **Figure 15-42** with the mass and rod tip no longer moving in unison, which signifies the end of impact.

Table 15-14 Rod Impact Results

Source	Max Stress (N / m^2)	Impact Duration (sec)
Theory	-2.00	3.068
MSC.Nastran	-1.84	3.075
%, Difference	-8.0%	0.23%

File(s): <install_dir>/results_vv_files/prob022.bdf, prob022.op2

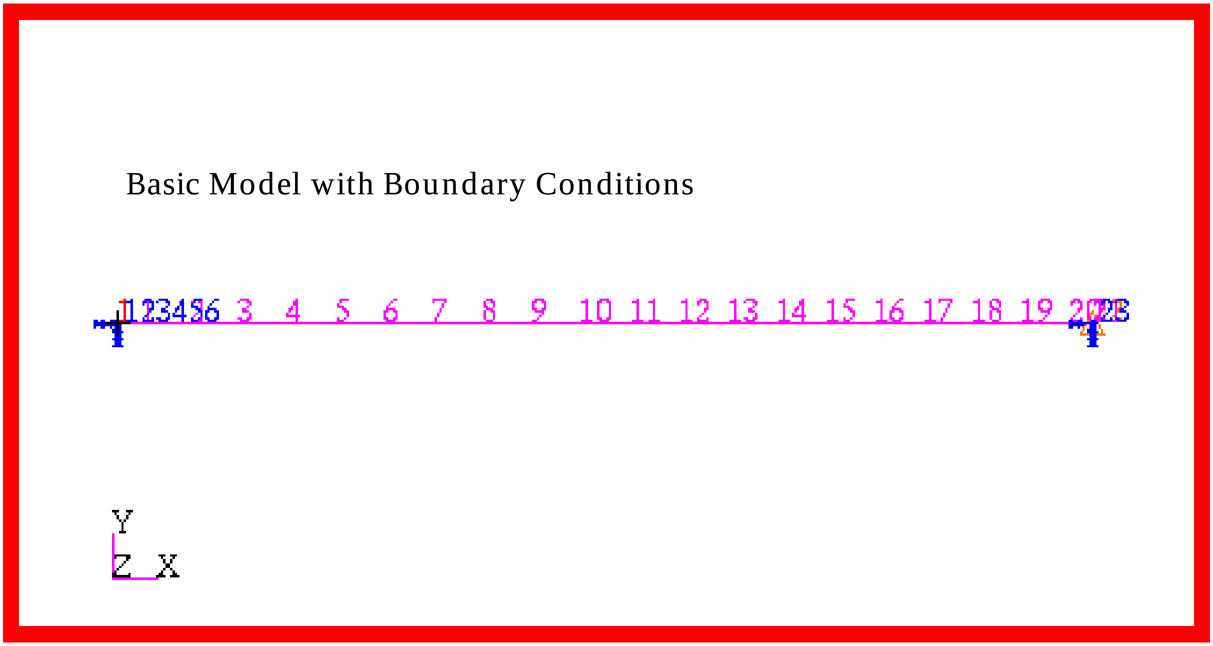


Figure 15-40 Basic Model of Impact Analysis.

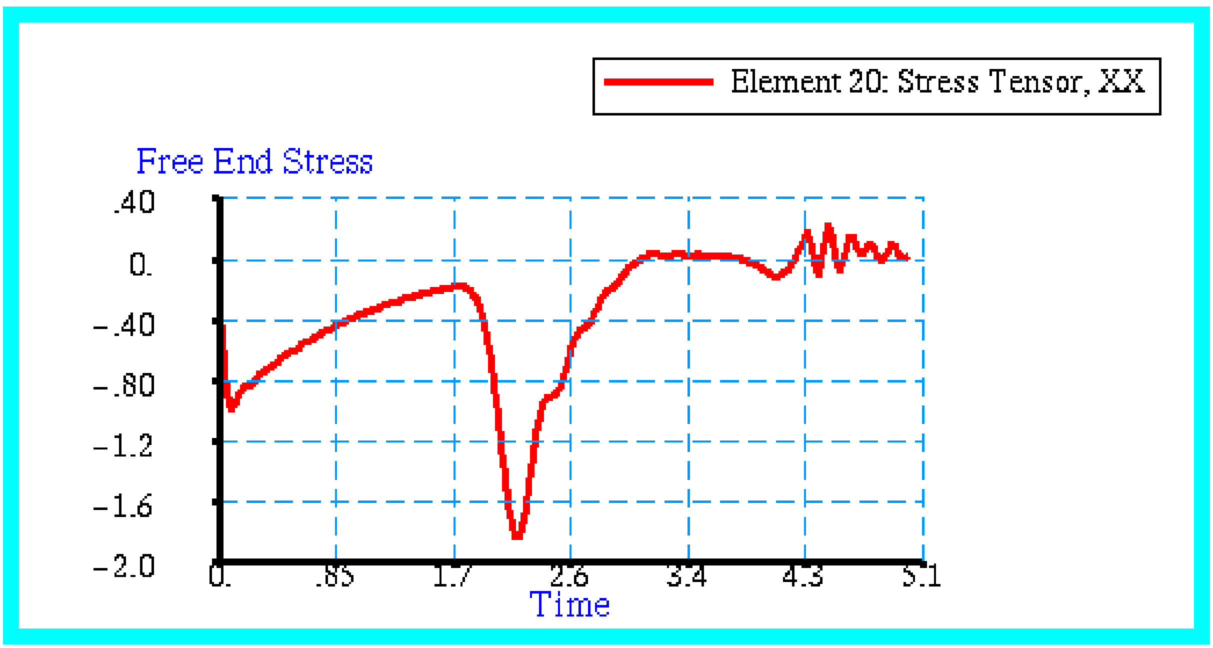


Figure 15-41 Free End Stress Response of Impact Model.

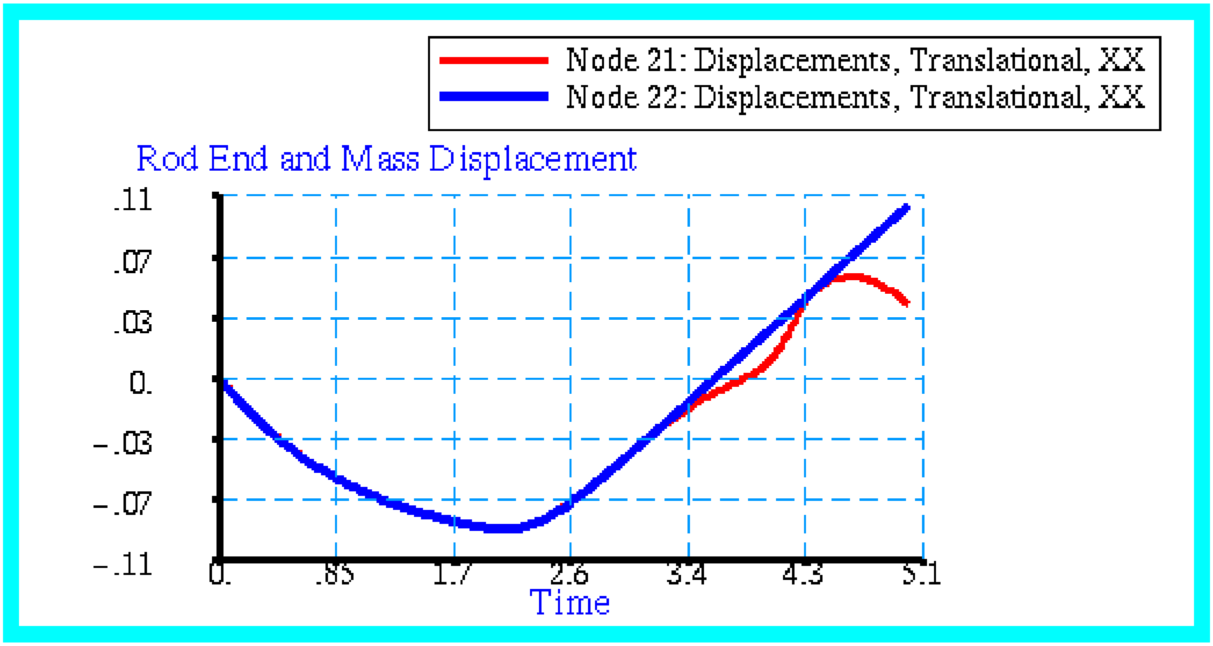


Figure 15-42 Rod End and Mass Displacement Responses.

Problem 23: Direct Frequency Response, Eccentric Rotating Mass with Variable Damping

Solution/Element Type:

MSC.Nastran, Direct Frequency Response, Solution 108, CBAR, CELAS1, CDAMP1, and CONM2 Elements.

Reference:

Thomson, W. T., *Theory of Vibration with Applications*, Prentice-Hall, Inc., 1972, pp. 45-52.

Problem Description:

A spring mass system is constrained to move in the vertical direction and is excited by a rotating machine that is unbalanced as shown below. The unbalance is represented by an eccentric mass, m, with eccentricity, e, which is rotating at an angular velocity, ω. Determine the undamped resonant frequency as well as the amplitude at this frequency when the damping is varied.

Engineering Data:

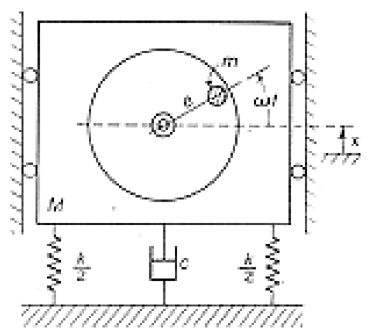
k = spring constant = 1000.0 lbf / inch

c = 2.5, 10.0 or 30.0 lbf-sec / inch

e = 0.1 inch

m = eccentric mass = .0633 lb-sec^2 / inch

M = total mass = .25 lb-sec^2 / inch



$$\frac{M}{m}\frac{X}{e} = \frac{\left(\frac{\omega}{\omega_n}\right)^2}{\sqrt{\left[1-\left(\frac{\omega}{\omega_n}\right)^2\right]^2 + \left[2\zeta\frac{\omega}{\omega_n}\right]^2}}$$

where:

$$c_c = 2m\omega_n = \text{critical damping factor}$$

$$\zeta = \frac{c}{c_c} = \text{critical damping ratio}$$

The resonant amplitude is then given by

$$X = \frac{\frac{me}{M}}{2\zeta}$$

For c = 2.5, $\zeta = 0.07906$ $X = 0.1602036$ inches

For c = 10.0, $\zeta = 0.31622$ $X = 0.0400509$ inches

For c = 30.0, $\zeta = 0.94868$ $X = 0.0133503$ inches

MSC.Nastran Results:

To model the eccentrically rotating mass system, the model shown in **Figure 15-43** was generated using MSC.Patran. To improve visualization, the nonrotating mass was modeled as a rod using two CBAR elements. The rod had a total mass equal to that of the nonrotating mass (M - m) and the eccentric mass *m*, or 0.25 lb-sec^2/inch. The ends of the rod were constrained to move in the y, or vertical, direction only. The ends of the rod were attached to grounded springs, each with a stiffness of 500 lbs/inch. The center of the rod was attached to a grounded damper. In addition, at the center of the rod, the force due to the rotation of the eccentric mass was applied. A plot of the applied force versus frequency is shown in **Figure 15-44**.

Using this model, a frequency response analysis was performed whereby the frequency and rotational loading was varied over a range of 0 to 30 hertz. The displacement at the center of the rod was recovered and is shown plotted in **Figure 15-45**. Here the MSC.Nastran results have been plotted with the aid of MSC.Patran for each of the damping ratios that were examined. The magnitude of the complex results is plotted. The results clearly reveal the presence of a resonant spike occurring at about 10 hertz that becomes rapidly attenuated as the damping is increased. At the highest damping level, a critically damped response is highly evident that is characterized by the absence of any amplification in the displacement at the resonant frequency.

Table 15-15 Amplitude at Resonant Frequency

Source	$\zeta = 0.0790569$	$\zeta = 0.0400509$	$\zeta = 0.948683$
Theory	0.160236	0.0400509	0.0133503
MSC.Nastran	0.1586090	0.0397802	0.0132626
%, Difference	-0.995%	-0.676%	-0.657%

File(s):`<install_dir>/results_vv_files/prob023_1.bdf, prob023_1.op2, prob023_2.bdf, prob023_2.op2, prob023_3.bdf, prob023_3.op2`

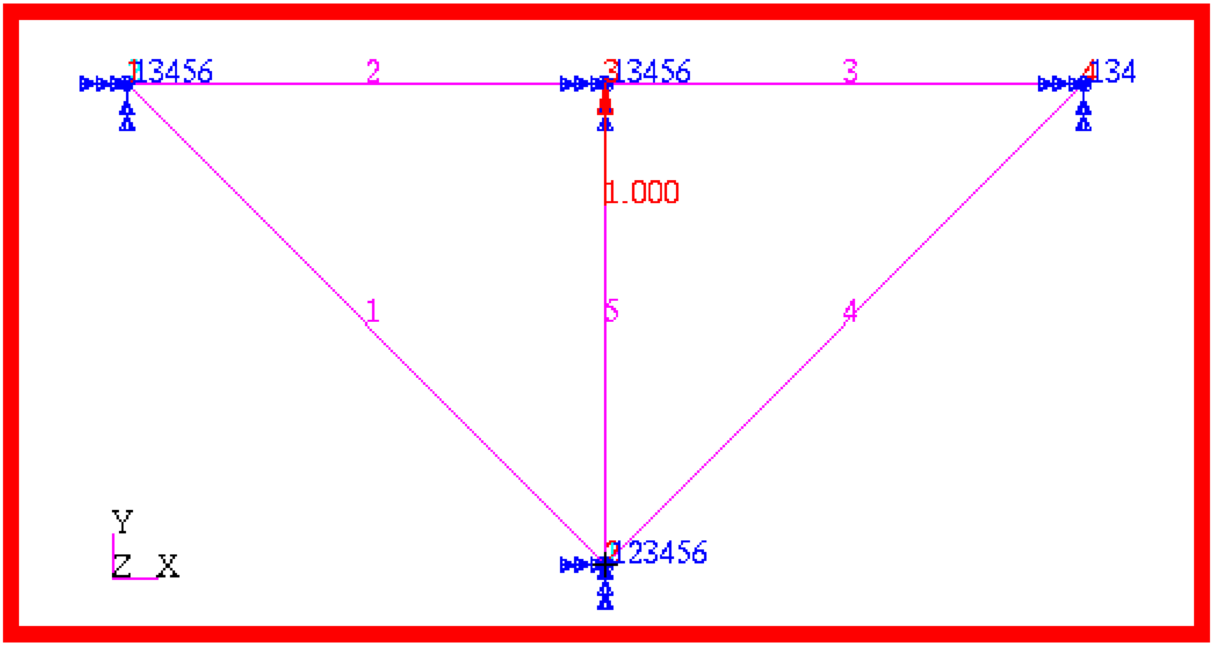


Figure 15-43 Basic Model of Direct Frequency Response Analysis.

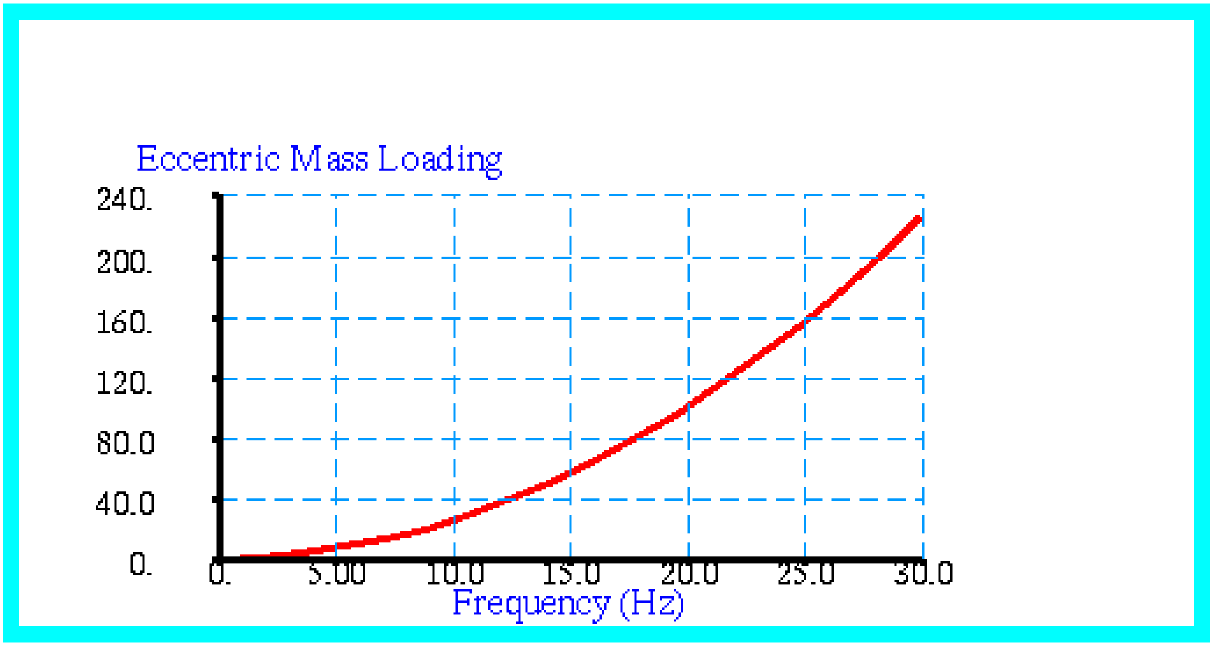


Figure 15-44 Applied Force versus Frequency.

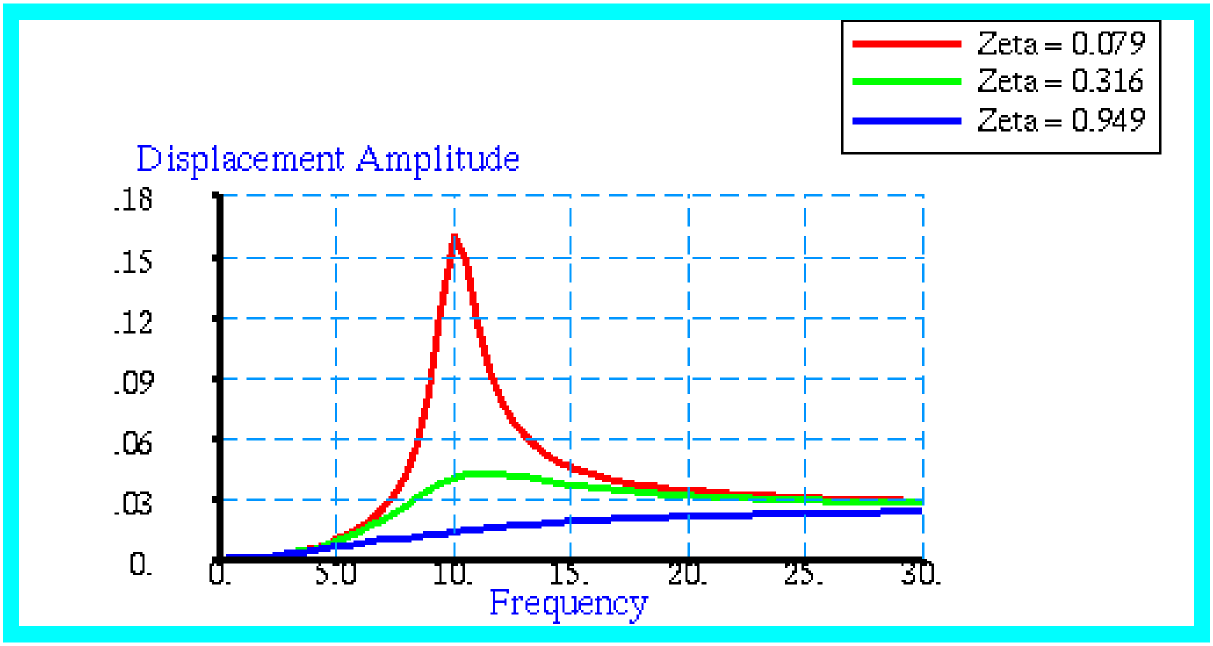


Figure 15-45 Amplitude Displacements at Center of Rod.

Problem 24: Modal Frequency Response, Enforced Base Motion with Modal Damping and Rigid Body Elements

Solution/Element Type:

MSC.NastranModal Frequency Response, Solution 111, CQUAD4, CELAS1, CONM2, and RBE2 Elements.

Reference:

Thomson, W.T., *Theory of Vibration with Applications*, Prentice-Hall, Inc., 1972, pp. 45-49.

Problem Description:

A long thin cantilevered plate has a concentrated mass suspended from its free end by two parallel springs. The base of the plate is excited by a sinusoidal excitation producing a peak unit acceleration. Find the resultant acceleration response of the suspended mass and the tip of the plate over a frequency range of zero to 100 hertz.

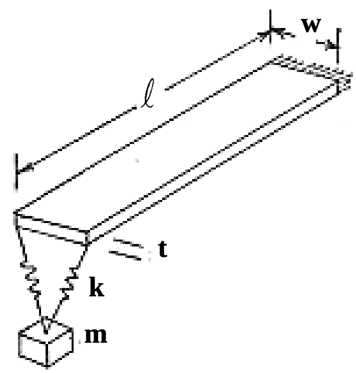
Engineering Data:

l = 12.0 inches

w = 2.0 inches

t = 0.5 inches

k = 50 lbf / inch



E = elastic modulus = 10^7 psi

ν = Poissons ratio = .33

ρ = mass density of plate = 0.026 lbf-sec^2 / inch^4

m = end mass = 0.5 lbf-sec^2 / inch

ζ = critial damping ratio = 0.1

Theoretical Solution:

For the cantilevered plate and suspended mass, the application of a sinusoidal base excitation will produce a resonant spike at each of the mode shapes associated with the end mass and with the tip of the plate. The amplitude induced at resonance will be a function of the amount of damping present. To determine where the expected resonant points occur, it is first necessary to estimate the natural frequencies for the end mass and plate.

First consider the end mass. This mass is suspended by two parallel springs whose total combined stiffness is simply the sum of the two. However, this combined spring is in series with another spring determined by the stiffness of the plate. For a cantilevered plate, the spring stiffness is given by:

$$k_{plate} = \frac{3EI}{l^3}$$

Substituting the assumed engineering data gives:

$$k_{plate} = 361.69 \text{ lbf / inch}$$

The total effective combined stiffness of the plate and the end springs is given by

$$k_{total} = \frac{2k \times k_{plate}}{2k + k_{plate}} = 78.34 \text{ lbf / inch}$$

The resonant frequency for the end mass then becomes:

$$f_{mass} = \frac{1}{2\pi}\sqrt{\frac{k}{m}} = 1.993 \text{ hertz}$$

For the cantilevered plate, the resonant frequencies are given by

$$f_i = \frac{\lambda_i}{2\pi l^2}\left(\frac{EI}{m_{plate}}\right)^{1/2}$$

where m_{plate} is the mass per unit length.

For the first two modes:

$$\lambda_1 = 1.875 \qquad \lambda_2 = 4.694$$

This gives for the plate:

$$f_1 = 10.99 \text{ hertz} \qquad f_2 = 68.89 \text{ hertz}$$

Since there exists a single resonant frequency associated with the motion of the end mass that lies within the frequency range of interest, the application of a unit base acceleration should cause a resonant response only at frequencies in the vicinity of 1.933 hertz. In contrast, for a point situated on the end of the plate, two additional resonant spikes will occur at frequencies near 10.99 hertz and 68.89 hertz.

For a harmonically excited damped oscillator, the width of the resonant spike is expressed by the factor Q which is defined as:

$$Q = \frac{1}{2\zeta} = \frac{\omega_n}{\Delta\omega} = \frac{\omega_n}{\omega_1 - \omega_2}$$

where ω_1 and ω_2 refer to those frequencies above and below ω_n where the response is approximately 70 percent of the response that occurs when $\omega = \omega_n$. These frequencies are referred to as the half power points. If the damping remains fixed over the entire frequency range of interest, then the bandwidth about the resonant points must be noticeably greater at the higher natural frequencies. Thus, a very narrow band response should be observed at low frequencies becoming increasingly broadband at the natural frequencies associated with the higher modes.

MSC.Nastran Results

To determine the frequency response of the system, the model shown in **Figure 15-46** was generated using MSC.Patran. The plate was modeled using standard CQUAD4 elements. The end mass was modeled using a lumped mass CONM2 element which was attached to the plate with two CELAS1 linear spring elements. To simulate the base, the other end of the plate was attached to an RBE2 rigid body element. At the independent node for the RBE2 element, a large concentrated mass was placed and excited by a force of sufficient magnitude to cause a unit acceleration throughout the frequency range of interest. The magnitude of the mass for the base was chosen to be 10^6 times as great as the entire mass of the plate and suspended end mass. This would ensure that the inertia of the base would dominate and impose the desired enforced motion. For the purpose of this analysis, the model was purposely constrained to prevent any lateral motion.

Before determining the frequency response of the plate-mass system, a modal analysis was performed to determine the natural frequencies and corresponding mode shapes. The first three predicted modes were plotted using MSC.Patran and are shown in **Figure 15-47** through **Figure 15-49**. Also displayed in these figures are the corresponding natural frequencies. Examination of these figures shows excellent agreement with the predicted modes. The first mode is predominately associated with motion of the end mass, whereas the higher modes mainly entail excitation of the plate. On the basis of the modal analysis, a frequency response analysis should exhibit a resonant peak in the acceleration occurring 2.0054, 12.84 and 71.134 hertz.

A subsequent frequency response analysis was performed with the preceding model using the modal method. For the purpose of this analysis, only the first five modes were retained, which included frequencies up to 195 hertz. This was deemed satisfactory since the maximum frequency of interest was only 100 hertz. The accelerations were recovered at the base, the suspended end mass and at the tip of the plate. These accelerations are shown plotted in **Figure 15-50** where the magnitude of the complex result has been plotted. The resultant accelerations show the correct behavior. The end mass shows a single resonant spike at the first mode, or approximately 2 hertz, whereas the plate tip shows three distinct resonance points occurring at each of the three natural frequencies that were previously calculated. In addition, the base shows a uniform unit acceleration over the entire frequency band, clearly indicating that a sufficiently large mass was chosen to obtain the desired enforced motion. Lastly, the resonant spikes for the plate tip shows an increasingly broadband behavior that would be consistent with a constant Q, or damped system.

For the problem at hand, there exists no rigorous closed form solution. The presence of the elastically suspended mass on a cantilevered plate creates a multi degree of freedom system. Closed form solutions only exist for single degree of freedom damped oscillators. Nevertheless, the estimated modes show the correct mode shapes and are occurring at approximately the expected natural frequencies. In addition, resonant peaks are occurring in the vicinity of the predicted natural frequencies, with an increasing broadband response at the higher modes that would be indicative of uniform modal damping.

File(s): <install_dir>/results_vv_files/prob024_1.bdf, prob024_1.op2, prob024_2.bdf, prob024_2.op2

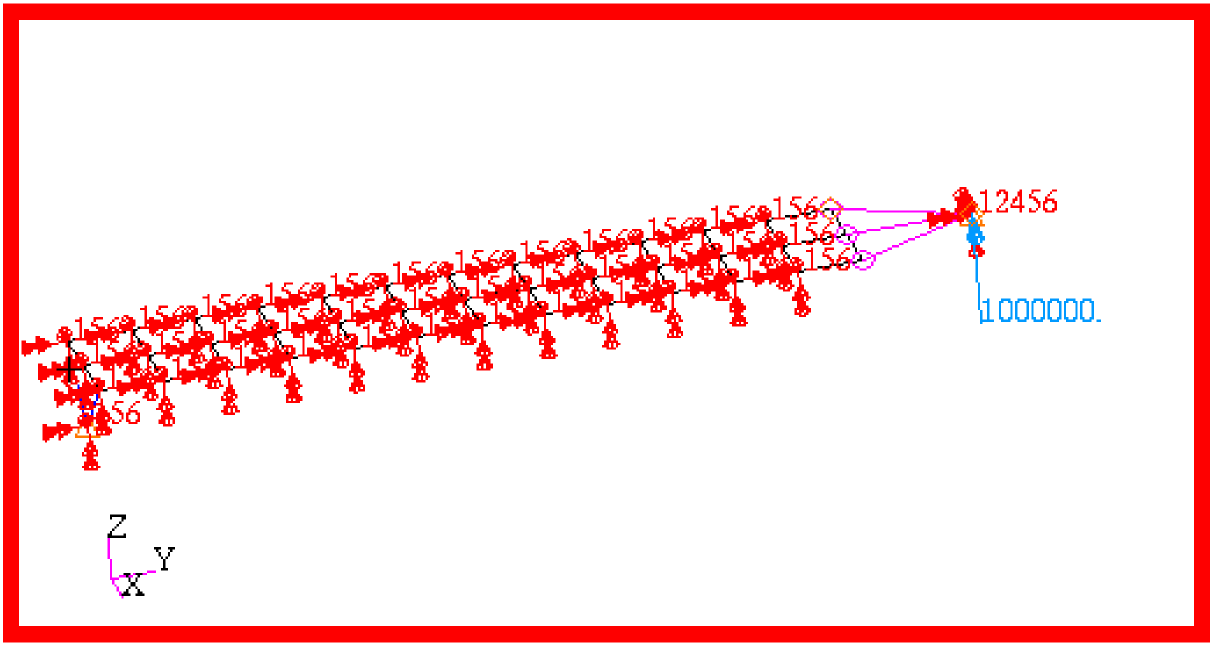


Figure 15-46 Model of Beam with Suspended Mass.

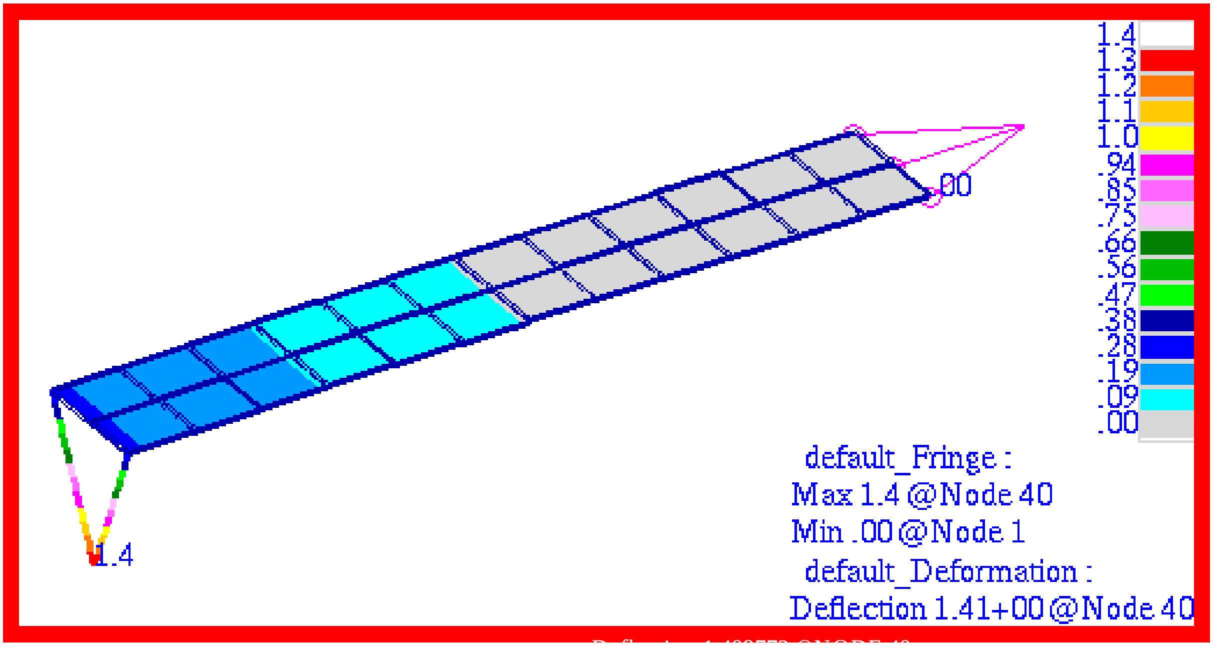


Figure 15-47 First Mode Shape of Beam, Freq=2.0054 Hz.

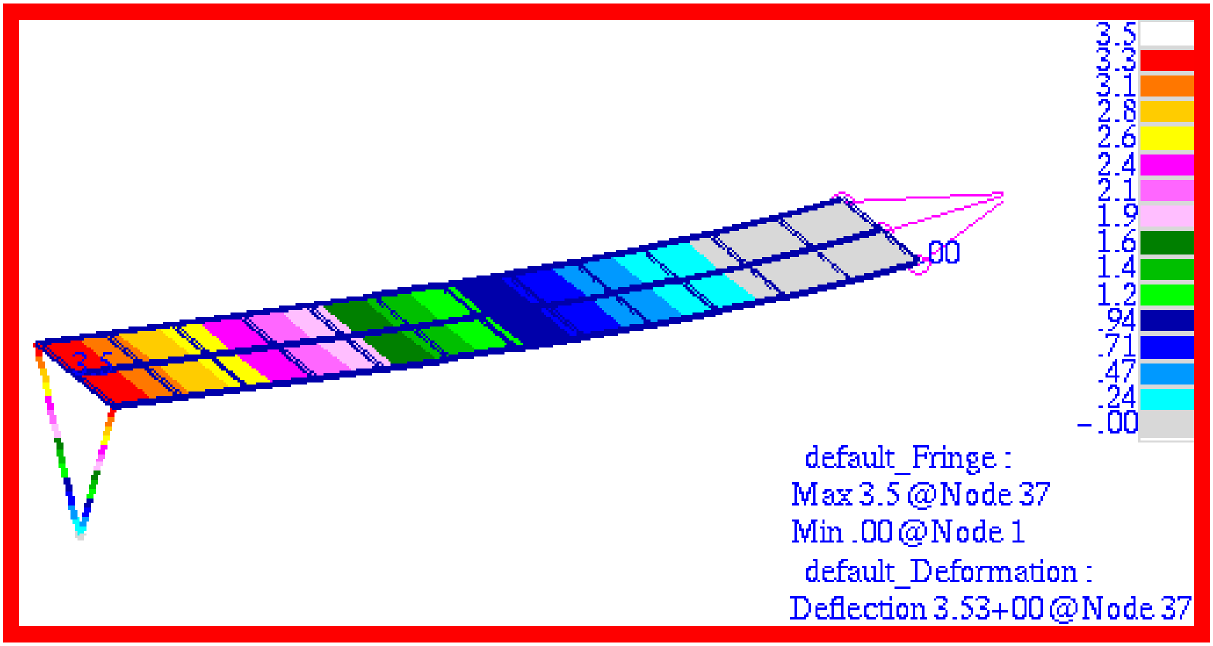


Figure 15-48 Second Mode Shape of Beam, Freq = 12.84 Hz.

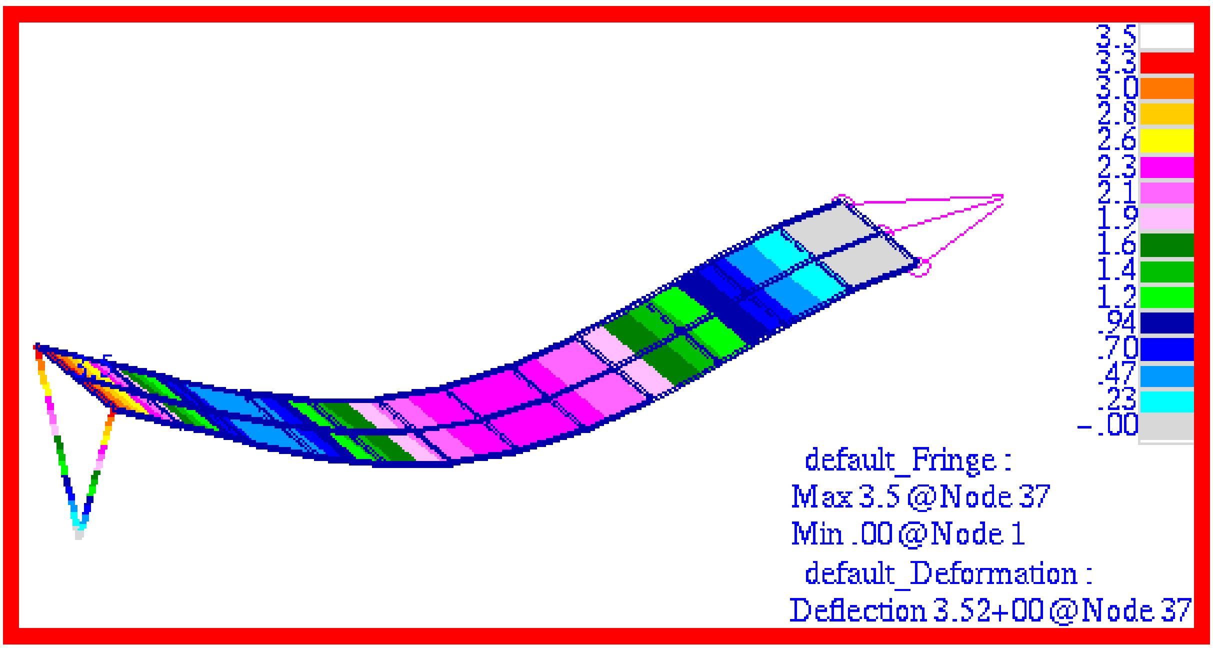


Figure 15-49 Third Mode Shape of Beam, Freq = 71.134 Hz.

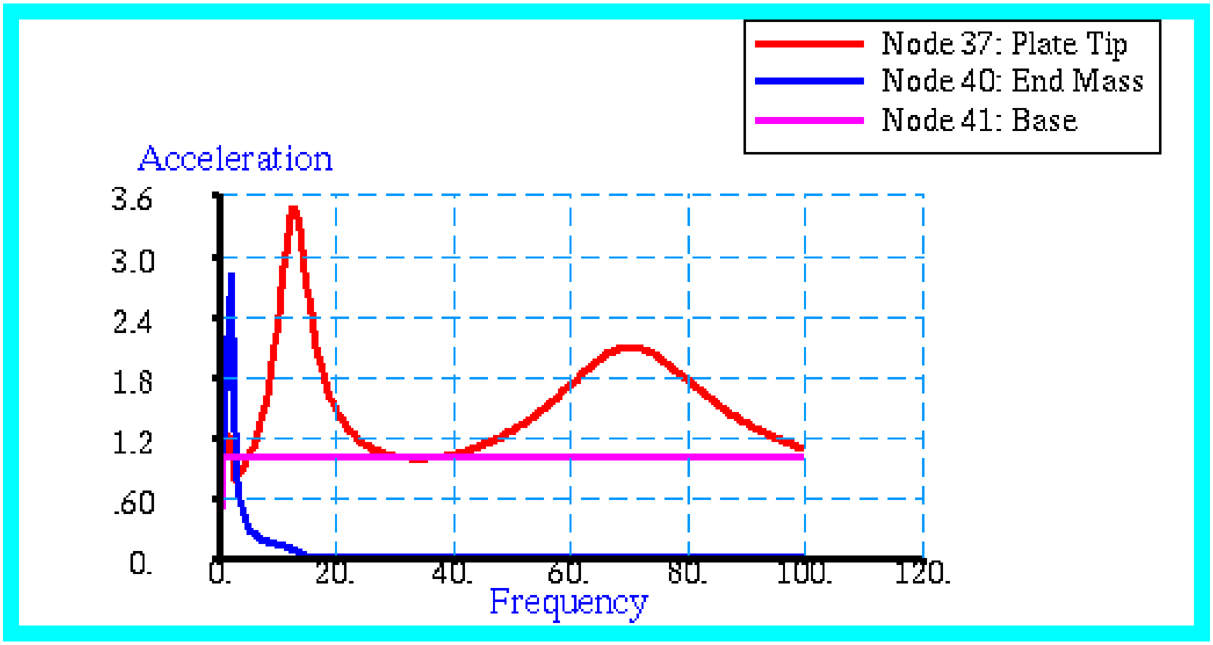


Figure 15-50 Acceleration Responses of Plate Tip, End Mass, and Base.

Problem 25:Modal Frequency Response, Enforced Base Motion with Modal Damping and Shell P-Elements

Solution/Element Type:

MSC.Nastran, Modal Frequency Response, Solution 111, CQUAD16, CELAS1, CONM2

Reference:

Thomson, W.T., *Theory of Vibration with Applications*, Prentice-Hall, Inc., 1972, pp. 41, 108, 158.

Problem Description

This is a repeat of **Problem 24: Modal Frequency Response, Enforced Base Motion with Modal Damping and Rigid Body Elements** (p. 444) with the exception that cubic QUAD16 pshell elements were used to model the plate. See this problem for a description of the model and all relevant engineering data.

Theoretical Solution

See **Problem 24: Modal Frequency Response, Enforced Base Motion with Modal Damping and Rigid Body Elements** (p. 444)

MSC.Nastran Results:

To determine the frequency response of the suspended end mass and the tip of the plate, the model shown in **Figure 15-51** was generated using MSC.Patran. In this instance, the plate was modeled using two cubic QUAD16 elements. The suspended mass was attached to the end of the plate using CELAS1 elements. Since RBE2 elements are not functional with p-elements, this necessitated that two large masses be attached directly to the corners at the base of the plate. At the site of the large masses, two sinusoidal forces were applied that were of sufficient magnitude to impart a unit acceleration throughout the entire frequency range of interest, or zero to 100 hertz. As before, base masses were chosen that were on the order of 10^6 times the mass of the plate and the suspended end mass. This would ensure that the desired enforced motion was being imparted at the base of the plate. In addition, a constant 10% critical modal damping was assumed.

Using the preceding model, a modal analysis was performed to determine all of the relevant modes that existed in the frequency range between zero and 100 hertz. The results of this analysis showed the existence of three distinct modes at the following frequencies:

Table 15-16 Predicted Natural Frequencies

Mode Number	Natural Frequency (Hertz)
1	2.009
2	12.984
3	72.613

The corresponding mode shapes were identical to those shown in **Figure 15-47** through **Figure 15-49** of **Problem 24: Modal Frequency Response, Enforced Base Motion with Modal Damping and Rigid Body Elements** (p. 444). In addition, the predicted natural frequencies closely matched those that were obtained with linear CQUAD4 elements, which were 2.0054, 12.840 and 71.134 hertz. The results of a frequency response analysis are shown in

Figure 15-52. Here the predicted accelerations at the base and tip of the plate and as well as at the suspended mass have been plotted versus frequency. The results show the correct response. Namely, resonant spikes are clearly evident at each of the predicted natural frequencies. The acceleration plot for the end mass shows a single resonance point since only one modes exit below 100 hertz. In contrast, the acceleration plot for the plate tip shows three distinct resonance peaks that correspond to the three predicted modes. A constant unit acceleration is predicted at the base of the plate, indicating that a large enough mass was used when modeling the base in order to impart the desired enforced motion. A comparison of the acceleration profiles plotted in Figure 15-47 with those shown in Figure 15-50 of Problem 24, clearly indicate that nearly identical results were obtained when using either QUAD4 elements with a standard h-shell formulation or QUAD16 cubic elements with a p-formulation. The actual peak accelerations that were obtained with both element types are summarized below.

Table 15-17 Peak Accelerations for Suspended Mass

Element Type	Peak Acceleration, Mode 1
QUAD4	2.798
QUAD16	2.795
%, Difference	-0.107%

Table 15-18 Peak Accelerations for Plate Tip

Element Type	Mode 1	Mode 2	Mode 3
QUAD4	1.170	3.448	2.084
QUAD16	1.170	3.480	2.168
%, Difference	0.0%	0.928%	4.03%

File(s):`<install_dir>/results_vv_files/prob025.bdf, prob025.op2`

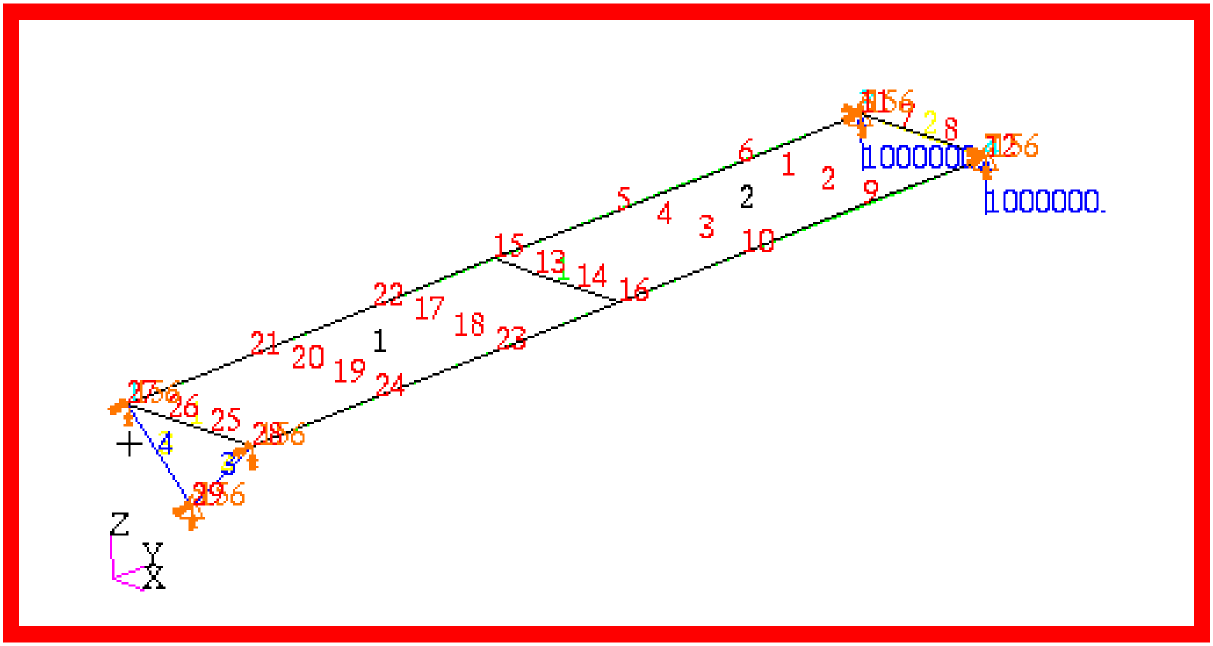


Figure 15-51 p-Element Model of Plate with Suspended Mass.

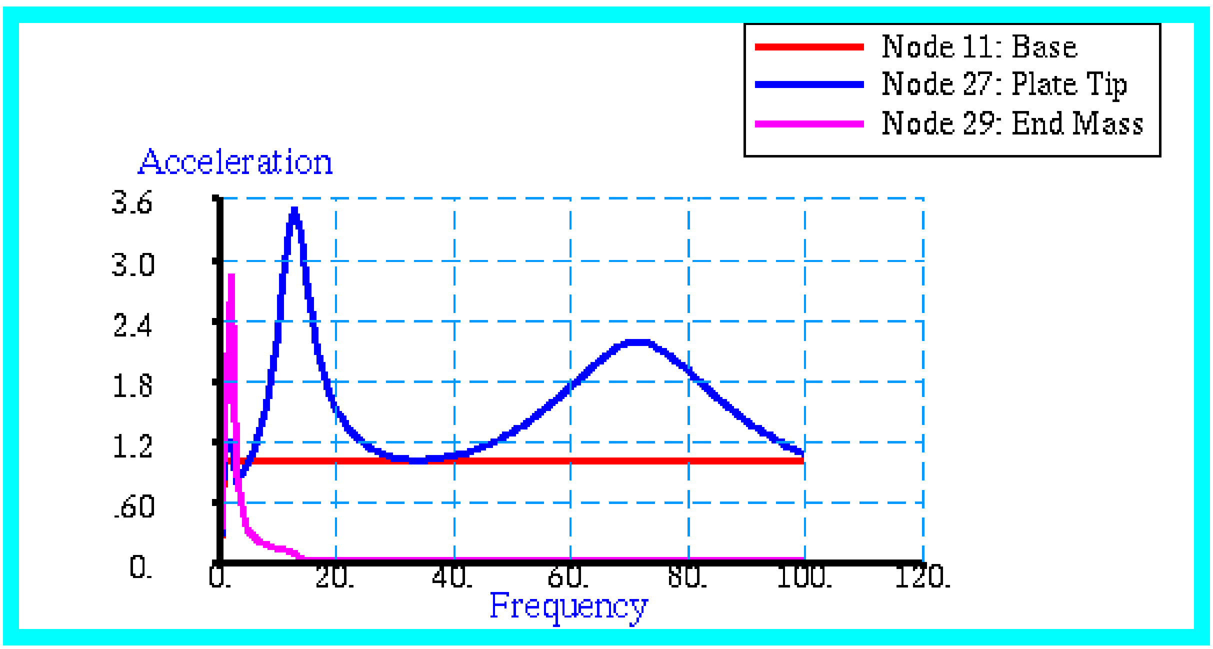


Figure 15-52 Acceleration Responses of Plate Tip, End Mass, and Base.

Problem 26: Complex Modes, Direct Method

Solution/Element Type:

MSC.Nastran, Complex Modes, Solution 107, CBAR, CONM2, CELAS1, and CDAMP Elements.

Reference:

Thompson, W.T., *Theory of Vibration With Applications*, Prentice-Hall,1972, pp. 23-27.

Blevins, R.D., *Formulas For Natural Frequency And Mode Shape*, Kreiger Publishing Co., 1979, p. 77.

Problem Description:

Three equal masses are each supported by a massless pendulum. Each pendulum is in turn coupled to the other two pendulums by a linear spring. In addition, there is a rotational damper located at the pivot point for each pendulum. Assuming that each damper has a different strength, determine the complex modes and natural frequencies for this coupled spring-mass system.

Engineering Data:

$L = 5$ inches

$a = 5$ inches

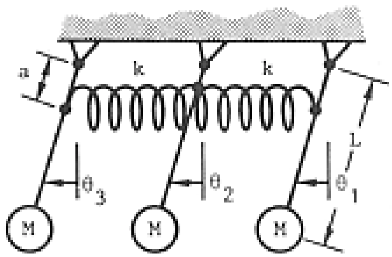
$k = 500\ \text{lbf} / \text{inch}$

$M = 0.01\ \text{lbf} - \text{sec}^2 / \text{inch}$

$C_1 = 120$ lbf - sec

$C_2 = 60$ lbf - sec

$C_3 = 20$ lbf - sec



Theoretical Solution:

For the undamped case, there are three modes associated with the angular displacements for each pendulum. The relative angular displacement for each mode are as follows:

$$\begin{bmatrix}\theta_1\\ \theta_2\\ \theta_3\end{bmatrix} = \begin{bmatrix}1\\1\\1\end{bmatrix},\ \begin{bmatrix}1\\0\\-1\end{bmatrix},\ \begin{bmatrix}1\\-2\\1\end{bmatrix}$$

The corresponding natural frequencies associated with each mode are

$$f_1 = \frac{1}{2\pi}\left(\frac{g}{L}\right)^{1/2},\quad f_2 = \frac{1}{2\pi}\left(\frac{g}{L}+\frac{ka^2}{ML^2}\right)^{1/2},\quad f_3 = \frac{1}{2\pi}\left(\frac{g}{L}+\frac{3ka^2}{ML^2}\right)^{1/2}$$

where g refers to gravitational constant, or 386.4 inches / sec^2. Substituting in the assumed engineering values yields the following undamped natural frequencies:

$$f_1 = 1.399\ \text{hertz},\quad f_2 = 35.616\ \text{hertz},\quad f_3 = 61.656\ \text{hertz}$$

When damping is present, the damped natural frequency is given by the expression:

$$\omega_d = \omega_n\sqrt{1-\zeta^2}$$

where

$$\omega_d = \omega_n\sqrt{1-\zeta^2}$$

$$\zeta = \text{critial damping ratio} = \frac{c}{c_{crit}}$$

and c_{crit} is the critical damping required to just suppress any oscillatory motion. For a simple pendulous mass, c_{crit} can be expressed as:

$$c_{crit} = 2ML^2\omega_n$$

To estimate the natural frequencies for the three damped oscillators, the average critical damping ratio is computed.

$$\bar{c} = \frac{120 + 60 + 20}{3} = 66.67 \text{ lbf - sec}$$

For the first mode, all of the masses are moving in unison. Thus, the most highly damped pendulum would dominate. However, since this pendulum has a damping rate of 120 lbf - sec, this yields a value of $\zeta > 1$, indicating that an overdamped situation should exist so that a non oscillatory mode with essentially zero frequency should result.

Similarly, the second pendulum has a damping rate of 60 lbf - sec which is approximately equal to the average overall damping. Thus, $\zeta \approx 1$ when the second oscillator dominates. When the ζ is equal to one, a critically damped situation exists which is once again characterized by non-oscillatory motion. Thus, there should be a second damped mode characterized by a natural frequency essentially equal to zero.

In the case of the second undamped mode, the motion is confined principally to the first and third pendulums. The lack of motion at the center pendulum effectively isolates the more lightly damped pendulum, thereby permitting an oscillatory motion to occur. Estimating the critical damping on the basis of the second undamped natural frequency gives:

$$c_{crit} = (2ML^2) \times (2\pi f_2) = 111.891 \text{ lbf - sec}$$

and

$$\zeta \approx \frac{c}{c_{crit}} \approx \frac{66.667}{111.891} \approx 0.595$$

This gives an estimated damped frequency for the second mode that is equal to:

$$f_{d,\ mode2} \approx f_2\sqrt{1-\zeta^2} \approx 28.626 \text{ hertz}$$

Similarly, for the third undamped mode, the critical damping is

$$c_{crit} = 193.698 \text{ lbf - sec}$$

and

$$\zeta \approx \frac{c}{c_{crit}} \approx \frac{66.667}{193.891} \approx 0.344$$

This gives an estimated damped frequency of

$$f_{d,\ mode3} \approx f_3\sqrt{1 - \zeta^2} \approx 57.896 \text{ hertz}$$

MSC.Nastran Results:

To compute the complex modes for the coupled pendulums, the model shown in **Figure 15-53** was generated using MSC.Patran. The pendulums were modeled with a single massless CBAR element while the end masses were modeled using lumped CONM2 concentrated mass elements. The springs were modeled with CELAS1 linear springs while grounded CDAMP1 elements were used for the rotational dampers.

Using this model, the complex modes were recovered. The following natural frequencies were predicted for each complex mode:

Table 15-19 Predicted Complex Modal Frequencies

Mode	Frequency (Hertz)
1	1.994×10^{-20}
2	4.776×10^{-12}
3	27.129
4	55.802

The corresponding mode shapes were plotted using MSC.Patran and are shown in **Figure 15-54** through **Figure 15-57**. Because of the presence of nonuniform damping, animation of any one of these modes will show a distinct phase difference in the motion of the individual pendulums.

The complex modal frequencies agree quite closely with the estimated values. As expected, the first two modes involve motion of all three pendulums. The higher damping associated with two of the pendulums produces an overdamped condition which results in a modal frequency of essentially zero. The third mode primarily involves motion of the outermost pendulums, which corresponds to the second undamped mode shape. The predicted value of 27.129 hertz for this mode agrees very closely with the estimated value of 28.626 hertz. Similarly, the fourth mode is characterized primarily by motion at the center pendulum which corresponds to the third undamped mode. The predicted modal frequency of 55.802 hertz once again agrees very favorably with the estimated value of 57.896 hertz.

File(s):`<install_dir>/results_vv_files/prob026.bdf, prob026.op2`

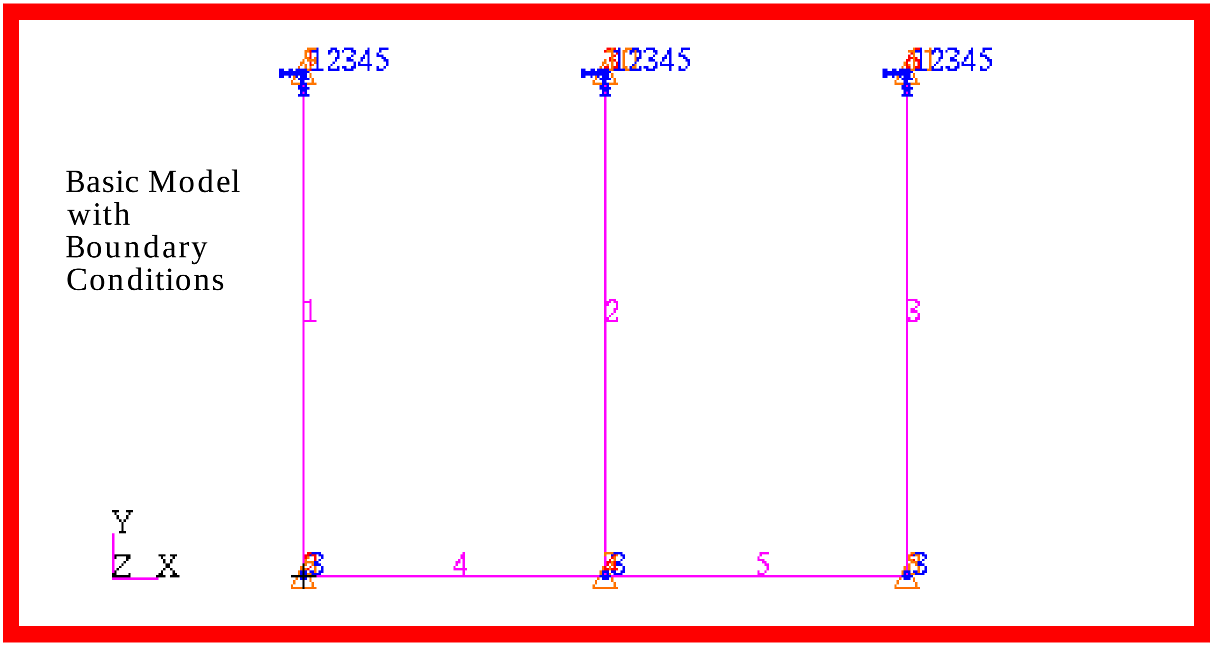


Figure 15-53 Model of Coupled Pendulums.

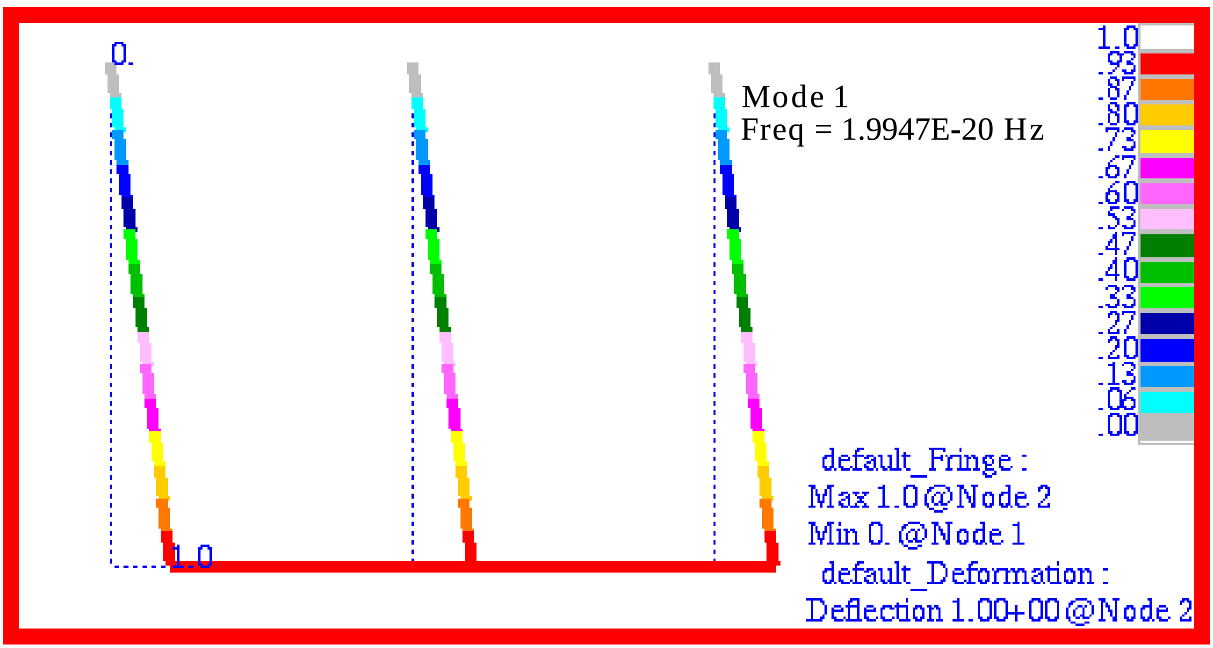


Figure 15-54 Complex Mode 1 of Pendulums, Freq = 1.9994E-20 Hz.

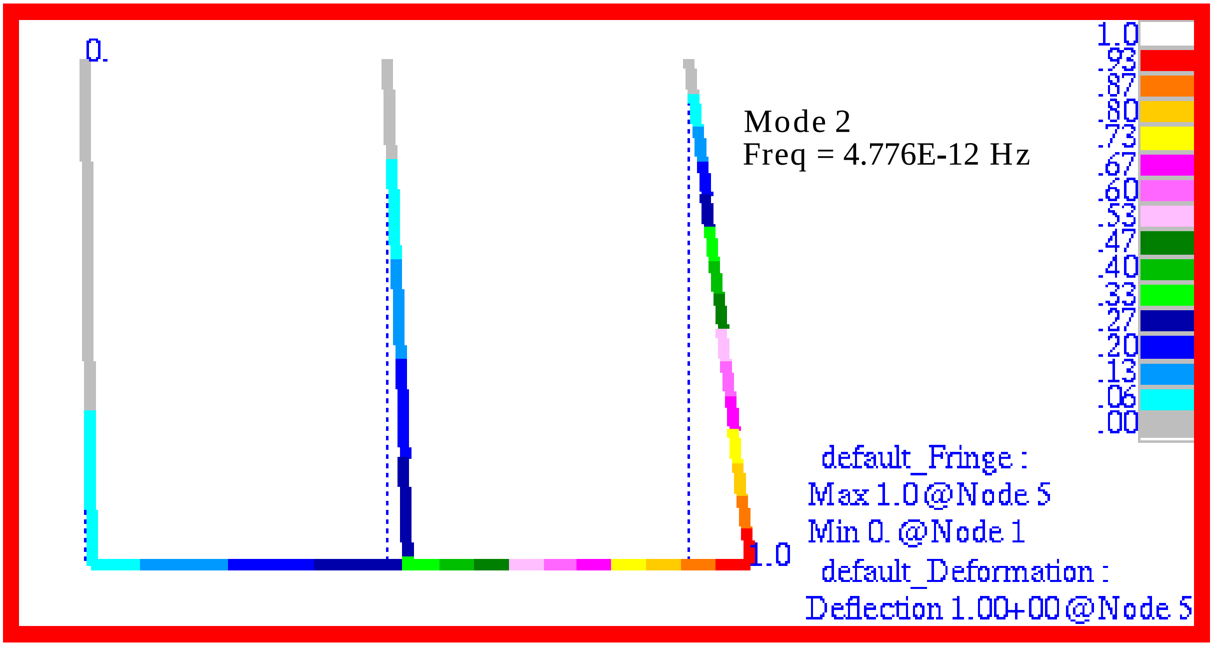


Figure 15-55 Complex Mode 2 of Pendulums, Freq = 4.776E-12 Hz.

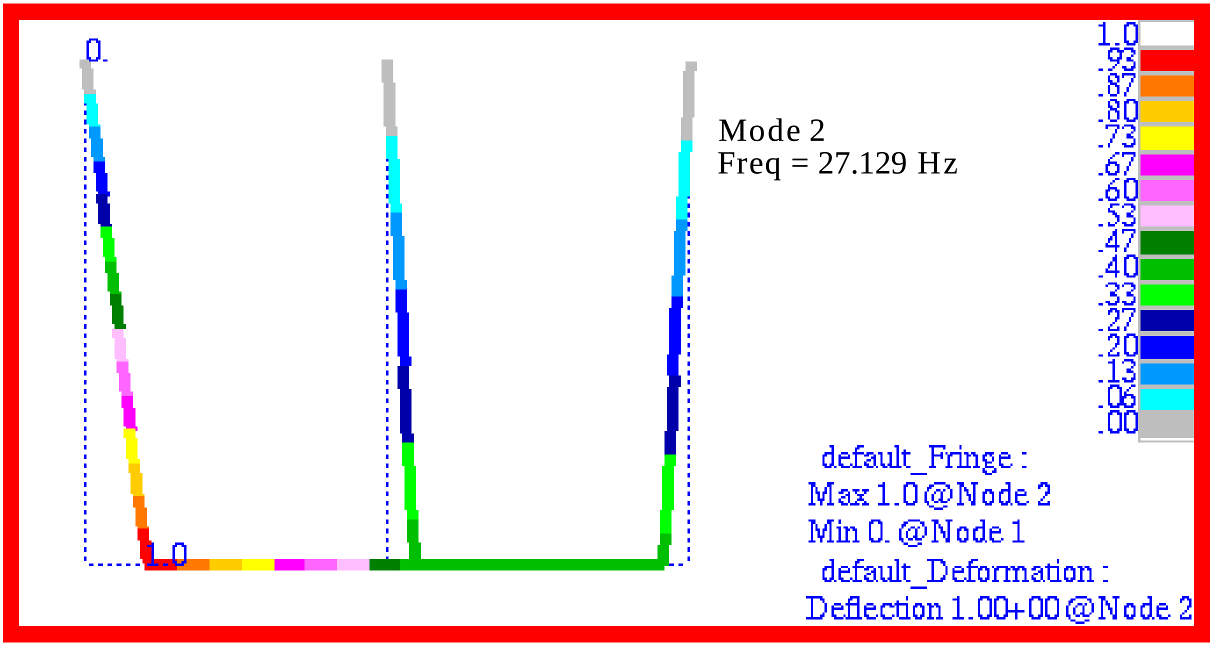


Figure 15-56 Complex Mode 3 of Pendulums, Freq = 27.129 Hz.

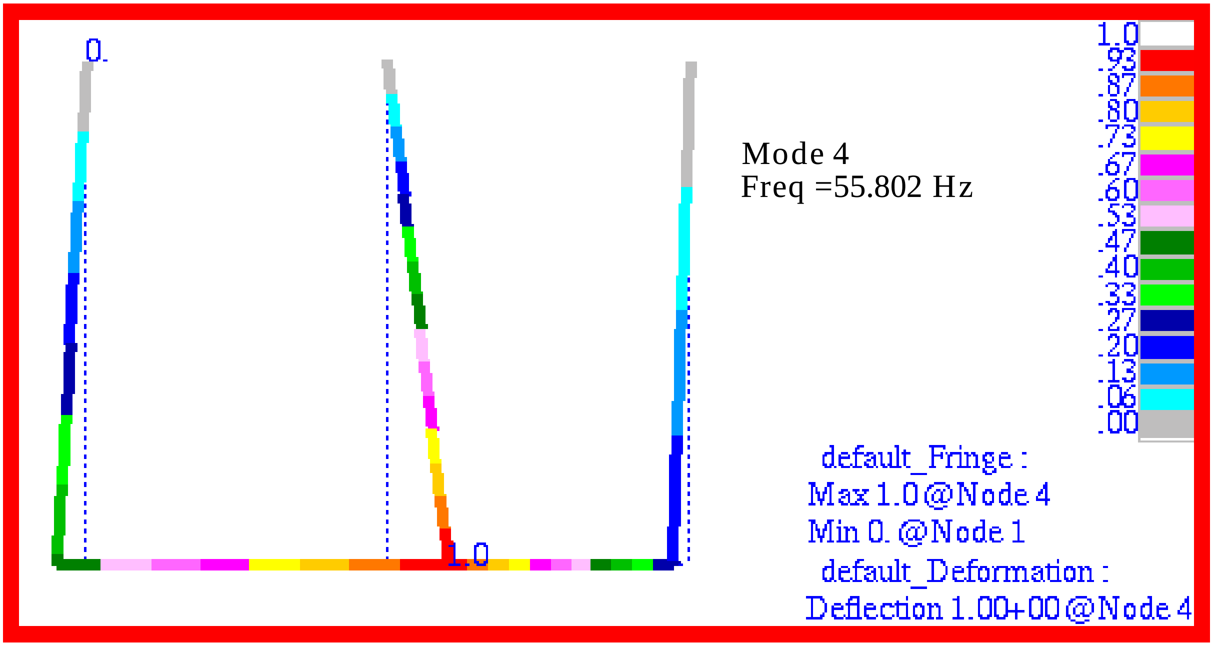


Figure 15-57 Complex Mode 4 of Pendulums, Freq = 55.802 Hz.

Problem 27: Steady State Heat Transfer, Multiple Cavity Enclosure Radiation

Solution/Element Type:

MSC.Nastran, Steady State Thermal Analysis, Solution 153, CQUAD4, CHBDYG, and RADCAV Elements

Reference:

MSC.Nastran *Thermal Analysis User's Guide*, Vol. 68, pp. 154-158.

Problem Description:

Four parallel plates radiate to each other and to the external ambient environment, assumed to be a perfect blackbody at a temperature of zero degrees K. If one of the plates is held at a constant temperature of $2000\ {}^\circ K$, find the temperature of the other three plates.

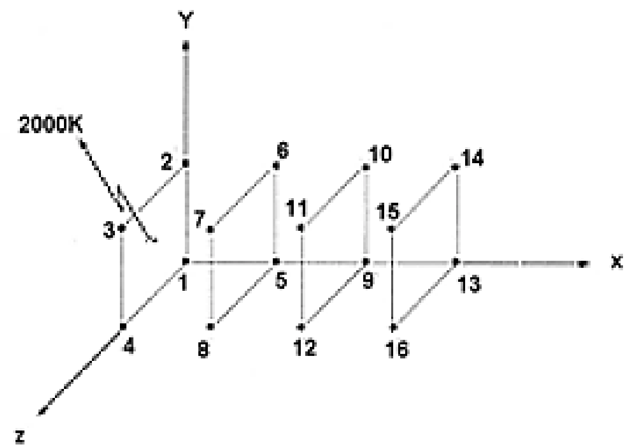
Engineering Data:

$$A \;=\; \text{area of plates} = 1.0 \text{ meters}^2$$

$$\varepsilon \;=\; \text{emmisivity} = \alpha \;=\; \text{absorptivity} = 1.0$$

$$T_\infty \;=\; 0.0\ {}^\circ K$$

$$\sigma \;=\; 5.67 \times 10^{-8}\ \text{W} \;/\; \text{m}\ {}^\circ K^4$$



Theoretical Solution:

For the four parallel plates, the theoretical value of the view factor, F, between successive plates is F = 0.2

Thus,

$$\frac{1}{AF} \;=\; 5.0$$

$$\frac{1}{A(1-F)} \;=\; \frac{1}{0.8} \;=\; 1.25$$

$$T_1 \;=\; 2000.0\ \ {}^\circ K$$

$$\varepsilon \;=\; 1.0$$

$$\frac{1-\varepsilon}{\varepsilon A} \;=\; 0.0$$

$$\frac{\sigma T_1^{\ 4}}{1.0206} = \frac{\sigma T_1^{\ 4} - \sigma T_2^{\ 4}}{5.0} + \frac{\sigma T_1^{\ 4}}{1.25}$$

$$\frac{\sigma T_1^{\ 4} - \sigma T_2^{\ 4}}{5.0} = \frac{\sigma T_2^{\ 4} - \sigma T_3^{\ 4}}{5.0} + \frac{\sigma T_2^{\ 4}}{0.625}$$

$$\frac{\sigma T_2^{\ 4} - \sigma T_3^{\ 4}}{5.0} = \frac{\sigma T_3^{\ 4} - \sigma T_4^{\ 4}}{5.0} + \frac{\sigma T_3^{\ 4}}{0.625}$$

$$\frac{\sigma T_3^{\ 4} - \sigma T_4^{\ 4}}{5.0} = \frac{\sigma T_4^{\ 4}}{1.25}$$

Solving for $T_1 T_2 T_3$ and T_4 gives:

$$T_2 = 1127.57 \quad °K$$

$$T_3 = 637.28 \quad °K$$

$$T_4 = 426.18 \quad °K$$

MSC.Nastran Results:

To solve for the temperatures of the plates, the model shown in **Figure 15-58** was generated using MSC.Patran. The model consisted of 4 CQUAD4 elements. The radiation heat transfer was modeled using three separate radiation cavities that ignored the effects of any shading caused by the intermediate plates. In addition, no ambient element was specified which in effect simulated the effect of having a perfect black body at zero degrees kelvin to model the far field ambient conditions.

The resultant steady state temperatures that were calculated for each of the plates are shown plotted in **Figure 15-59**. The corresponding radiation heat fluxes are plotted in **Figure 15-60** along with the actual nodal values for the radiation heat flux. Closer examination of the nodal fluxes reveals the tremendous decrease that occurs in the heat flux when multiple parallel plates are present.

Table 15-20 Steady State Temperatures in Parallel Plates

Source	T_2	T_3	T_4
Theory	1127.57	637.28	426.18
MSC.Nastran	1127.46	637.17	426.06
%, Difference	0.01%	0.02%	0.03%

File(s):`<install_dir>/results_vv_files/prob027.bdf, prob027.op2`

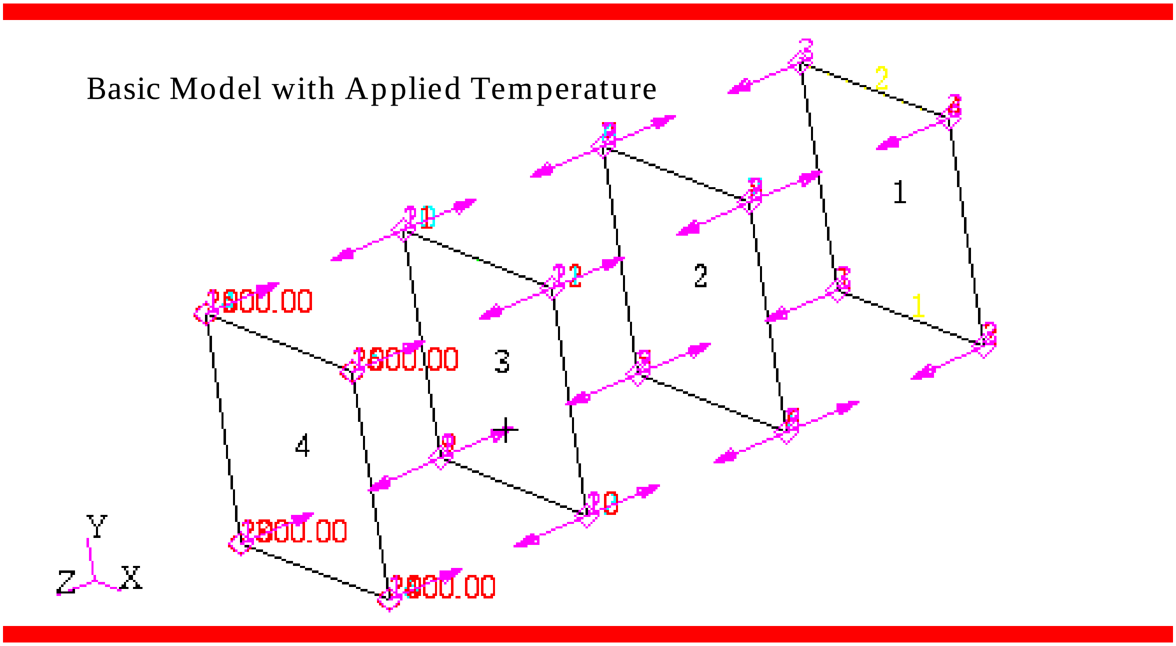


Figure 15-58 Model of Four Parallel Plates.

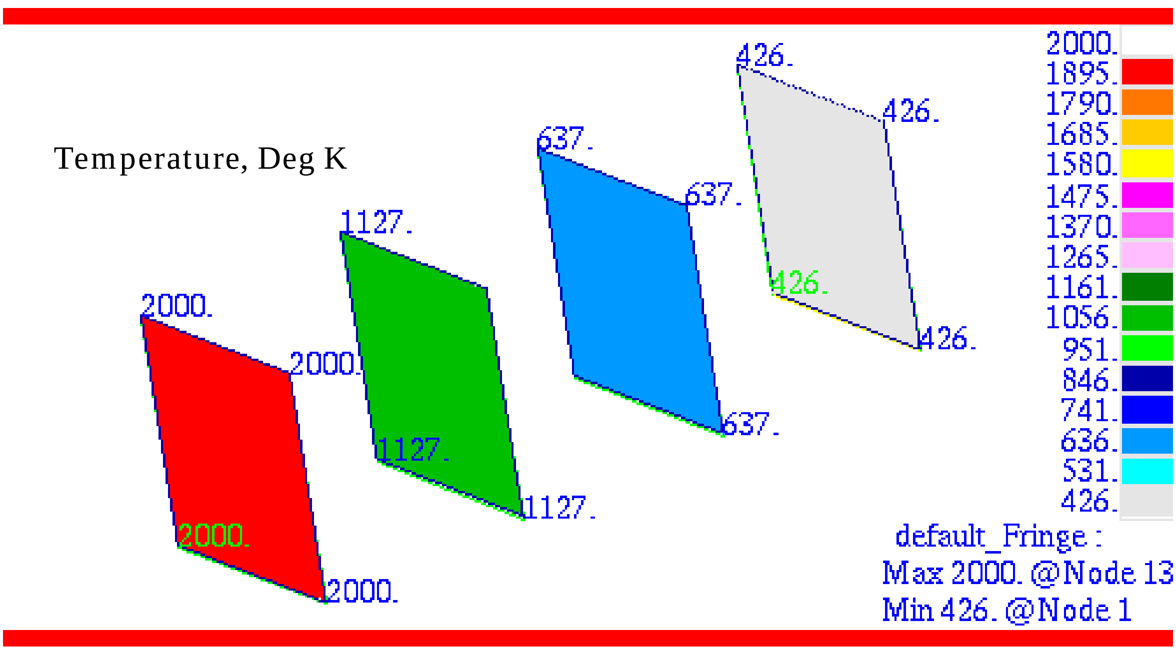


Figure 15-59 Steady State Temperatures of the Four Parallel Plates.

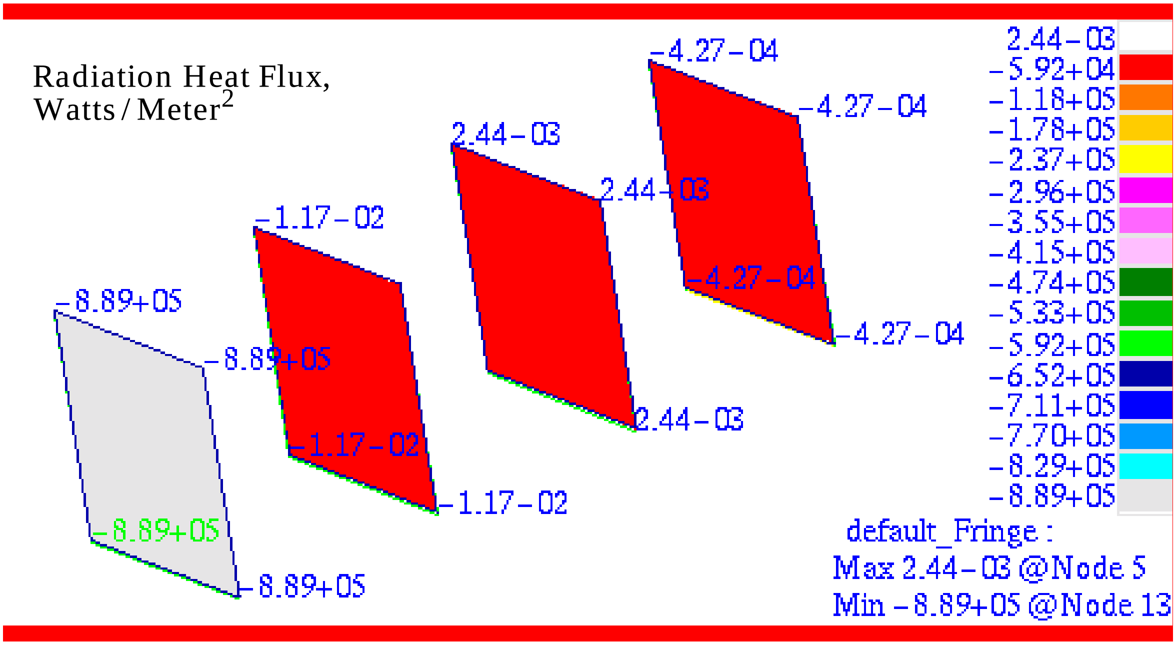


Figure 15-60 Radiation Heat Fluxes of the Four Parallel Plate.

Problem 28: Transient Heat Transfer with Phase Change

Solution/Element Type:

MSC.Nastran, Transient Thermal Analysis, Solution 159, CHEX8, CHBDYG, and CONV Elements.

Reference:

MSC.Nastran *Thermal Analysis User's Guide*, Vol. 68, pp. 188-191.

Problem Description:

A cube of water is exposed to an ambient environment that is below the freezing point of water. In addition, the water is at an uniform initial temperature that is above the freezing point. Forced convection occurs along the lateral faces of the cube. Determine the time it takes for freezing to first occur as well as the total time required for all of the water to freeze.

Engineering Data:

$l = 0.1$ meter

h = convective film coefficient = $100\ \mathrm{W/m^2}\ °C$

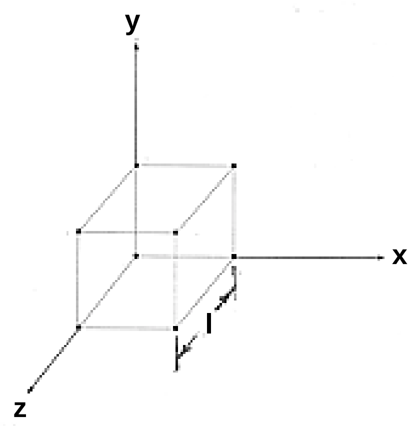
L = latent heat of formation = $3.34 \times 10^5\ \mathrm{J/kg}$

T_c = freezing poing = $0\ °C$

$\Delta T = 2.0\ °C$ = temperature range for freezing

T_∞ = ambient temperature =-20.0 $°C$

$T_{initial}$ = initial temperature



Theoretical Solution:

If no phase change occurs, the heat balance equation is:

$$\rho c_p V \frac{dT}{dt} = -hA(T - T_\infty)$$

Assuming constant properties, then

$$\rho c_p V \ln[(T - T_\infty) / (T_2 - T_\infty)] = hA(T_2 - T_1)$$

The elapsed time required for complete freezing to occur is calculated from the following expression:

$$\Delta t_c = \rho V(L + c_p \Delta T) / (hA|T_c + 0.5\Delta T - T_\infty|)$$

For this specific problem:

$$A = 0.06\ \mathrm{m}^2,\ V = 0.001\ \mathrm{m}^3,\ T_c = 0.0\ °C,\ \Delta T = 2.0\ °C,$$

$$L = 3.34 \times 10^5 \text{ J / Kg, h} = 100.0 \text{ W / m}^2\text{, } \rho_w = 1000.0 \text{ Kg / m}^3$$

$$c_{p,w} = 4217 \text{ J / Kg } {}^\circ C,$$

Substituting these values into the preceding equations gives:

$$\text{time to initiate freezing} = t_c = 420 \text{ seconds}$$

$$\text{total phase change time} = \Delta t_c = 2718 \text{ seconds}$$

MSC.Nastran Results:

To model the freezing of the unit cube of water, the model shown in **Figure 15-61** was generated using MSC.Patran. The cube was meshed with a single CHEX8 element. A uniform convective boundary condition was imposed along all of the exposed faces of the cube along with an initial temperature of -20 ^{o} C. The properties of water were assigned to the solid element and it was assumed that freezing would initiate at 0 ^{o} C and occur over a temperature range of 2 ^{o}C. A transient analysis was then performed with MSC.Nastran using a fixed time step 5 seconds for a total duration of 5000 seconds.

Using MSC.Patran, the temperature history was plotted for a single node situated at the base of the cube. The results are shown in **Figure 15-62**. Examining this figure, it is quite evident when the water began to freeze since the temperature will level off due to the release of the latent heat associated with the formation of ice. Once freezing has completed, the temperature will continue to decline until it reaches the ambient temperature, or $-20°C$.

The corresponding rate of change in enthalpy is shown in plotted in **Figure 15-63**. Since the total enthalpy is just the product of the specific heat and temperature, during a phase change the rate of change of enthalpy should remain fairly constant. Afterward, the enthalpy will continue to decline at a decreasing rate as the ice asymptotically approaches the ambient temperature. This behavior is clearly evident in **Figure 15-63**.

Table 15-21 Transient Temperature Results

Source	t_c ,(seconds)	Δt_c ,(seconds)
Theory	420	2718
MSC.Nastran	430	3100
%, Difference	2.38%	14.05%

File(s):`<install_dir>/results_vv_files/prob028.bdf, prob028.op2`

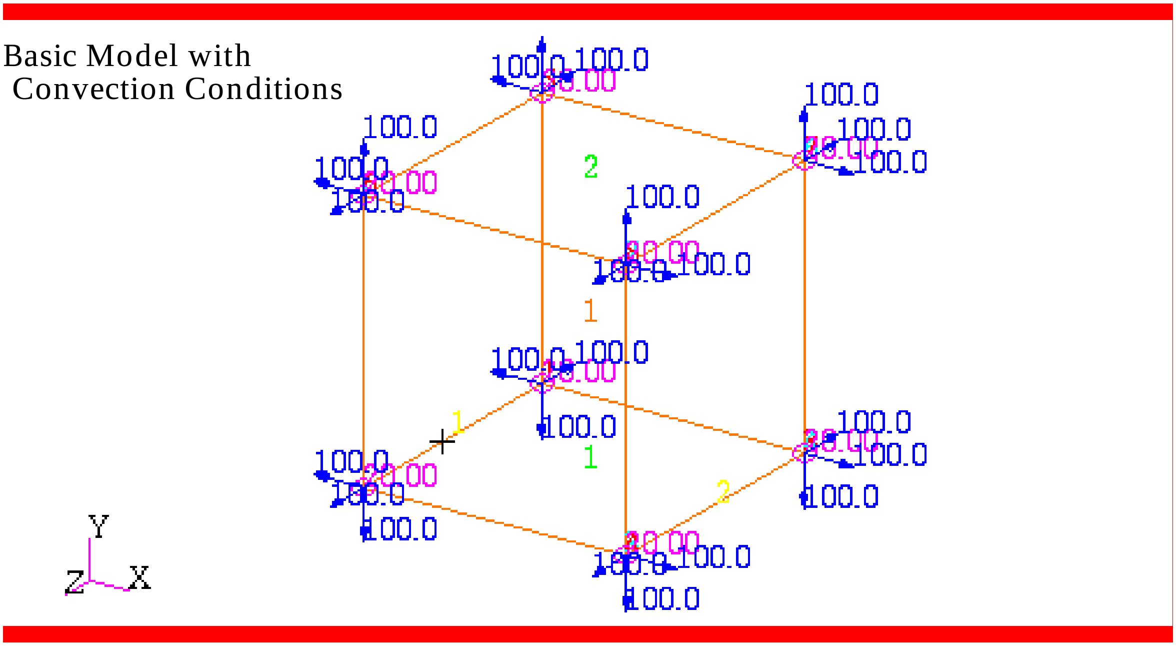


Figure 15-61 Model of Freezing Cube.

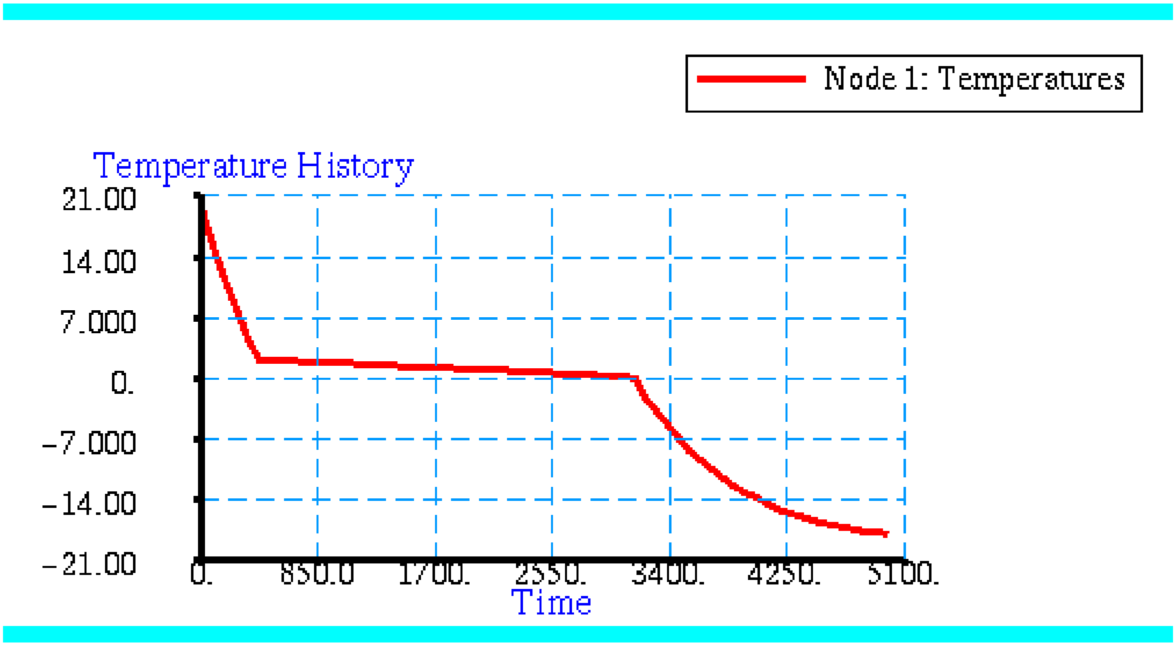


Figure 15-62 Temperature Response of Freezing Cube.

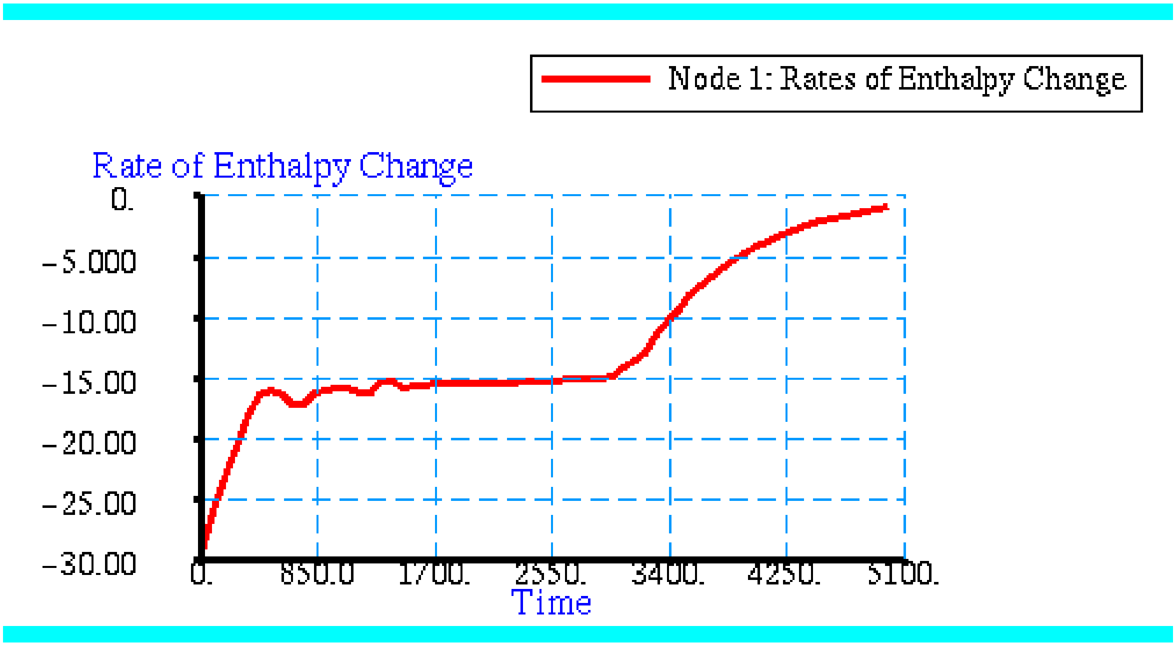


Figure 15-63 Rate of Enthalpy Change of Freezing Cube.

Problem 29: Steady State Heat Transfer, 1D Conduction and Convection

Solution/Element Type:

MSC.Nastran, Steady State Thermal Analysis, Solution 153, CROD, CHBDYP, and CONV Elements.

Reference:

Chapman, A.J., *Heat Transfer*, 3rd ed., New York, MacMillan Publishing Co., Inc., 1974, p. 76.

Problem Description:

A fin of circular cross section is maintained at 250 ^{o} F at one end. The fin extends into air at an ambient temperature of 70 ^{o} F. Assuming a material conductivity of 132 Btu / hr-ft-^{o}F and a surface convective film coefficient of 1.6 Btu / hr-ft ^{2}-^{o} F, determine the steady state temperature distribution along the length of the rod.

Engineering Data:

L = 1 ft

d = diameter = 0.04167 ft

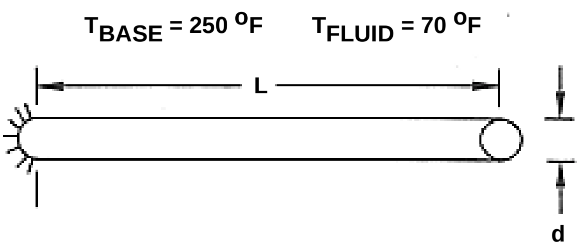
A = lateral area = 0.001365 ft ^{2}

k = thermal conductivity = 132 Btu / hr-ft-^{o} F

h = convective film coefficient = 1.6 Btu / hr-ft ^{2} -^{o} F

T_{base} = 250 ^{o} F

T_{fluid} = 70 ^{o} F



Theoretical Solution:

Assuming only 1D heat transfer along the length of the fin as well as no convective heat loss at the tip, the theoretical temperature distribution is

$$T = (T_{base} - T_{fluid})\left(\frac{\cosh(\mathrm{ml} - \mathrm{x})}{\cosh \mathrm{ml}}\right)$$

where

$$m = \sqrt{\frac{4h}{kd}} = \sqrt{\frac{4 \times 1.6}{0.63 \times \frac{1}{2 \times 12}}}$$

$$= 1.079\frac{1}{ft}$$

Substituting for m, gives the following steady state temperatures at four positions along the fin.

X / L	T (^{o} F)
.25	217.0243
.50	196.0559
.75	183.7156
1.00	179.7002

MSC.Nastran Results:

To compute the temperature distribution for the cooling fin, the model shown in **Figure 15-64** was generated using MSC.Patran. The fin was modeled using 4 CROD elements. By doing so, this enabled the convective boundary condition to be modeled with line type CHBDYP convective boundary elements which alleviated the need to compute the lateral surface area associated with each node. Instead, this was determined by MSC.Nastran from the cross-sectional area that was given for the rod.

Using the model shown in **Figure 15-64**, the nodal temperatures were computed. A fringe plot of the steady state temperature distribution that was generated by MSC.Patran is shown in **Figure 15-65**. Here the fringe labels have been included to better show the actual nodal values.

Table 15-22 Temperatures Along Cooling Fin

Source	X / L			
	.25	.50	.75	1.00
Theory	217.0243	196.0559	183.7156	179.7002
MSC.Nastran	217.7879	196.3183	184.0306	180.0316
%, Difference	0.035%	0.013%	0.172%	0.184%

File(s):<install_dir>/results_vv_files/prob029.bdf, prob029.op2

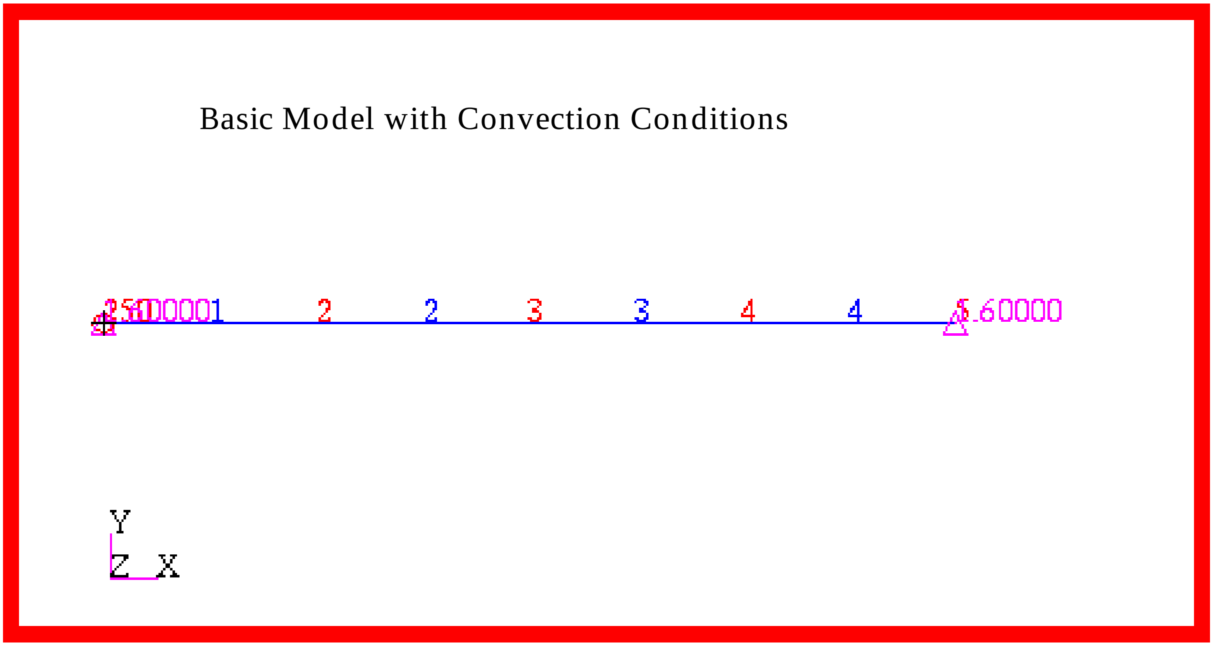


Figure 15-64 Model of Cooling Fin.

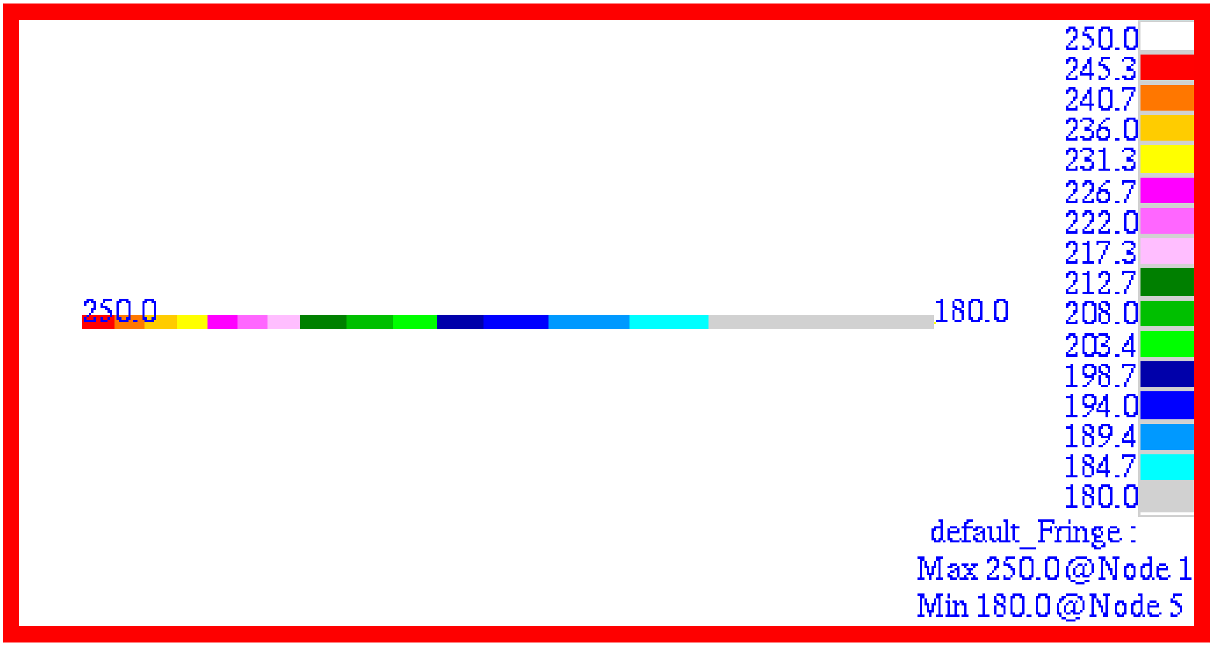


Figure 15-65 Nodal Temperatures of Cooling Fin.

Problem 30: Freebody Loads, Pinned Truss Analysis

Solution/Element Type:

MSC.Nastran, Linear Statics, Solution 101, CROD Elements.

Reference:

Przemieniecki, J.S., *Theory of Matrix Structural Analysis*, McGraw-Hill, Inc., 1968, p. 155.

Problem Description:

A pinned joint truss is loaded with a force at one end and one of the components is heated uniformly to an elevated temperature. Considering thermal effects, determine the freebody loads for each of the members in the truss.

Engineering Data:

This is a repeat of **Problem 8: Linear Statics, Pinned Truss Analysis** (p. 366) except that the freebody forces will be plotted.

$E = 1.0 \times 10^{7}$

$\alpha = 1.0 \times 10^{-6}$ / Deg F

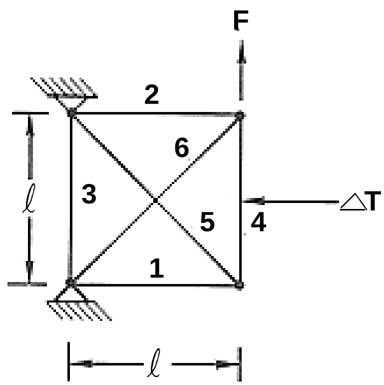
$A_1 = 1.0\ \text{in}^2$ (Elements 1, 3, 5 and 6)

$A_2 = 0.7071068\ \text{in}^2$ (Elements 2 and 4)

$\Delta T = +100.$ Deg F (Elements 3 only)

$F = 1000.0$ lb.

$\ell l = 20.$ in



Theoretical Solution:

See **Problem 8: Linear Statics, Pinned Truss Analysis** (p. 366).

MSC.Nastran Results:

Using the model shown in **Figure 15-66**, the same results were obtained with MSC.Nastran as in **Problem 8: Linear Statics, Pinned Truss Analysis** (p. 366). The MSC.Patran Freebody application was used to determine freebody force associated with every member of the truss. The resultant freebody forces computed by MSC.Patran are shown in **Figure 15-67** through **Figure 15-72**. The freebody forces should be identical to the predicted rod forces which they are. The freebody forces and moments for the entire truss are plotted in **Figure 15-73**. The freebody moments are labeled in blue and are all zero in order to maintain rotational equilibrium. The results can be compared with **Table 15-28**.

File(s):<install_dir>/results_vv_files/prob030.bdf, prob030.op2

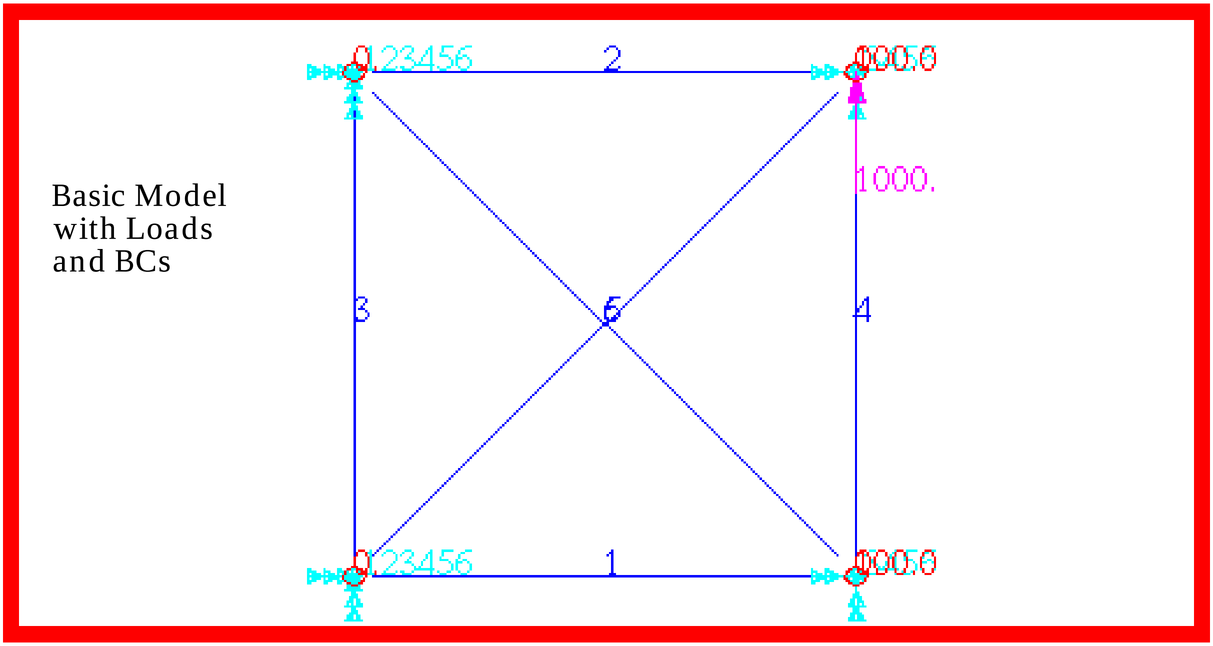


Figure 15-66 Model of Pinned Truss.

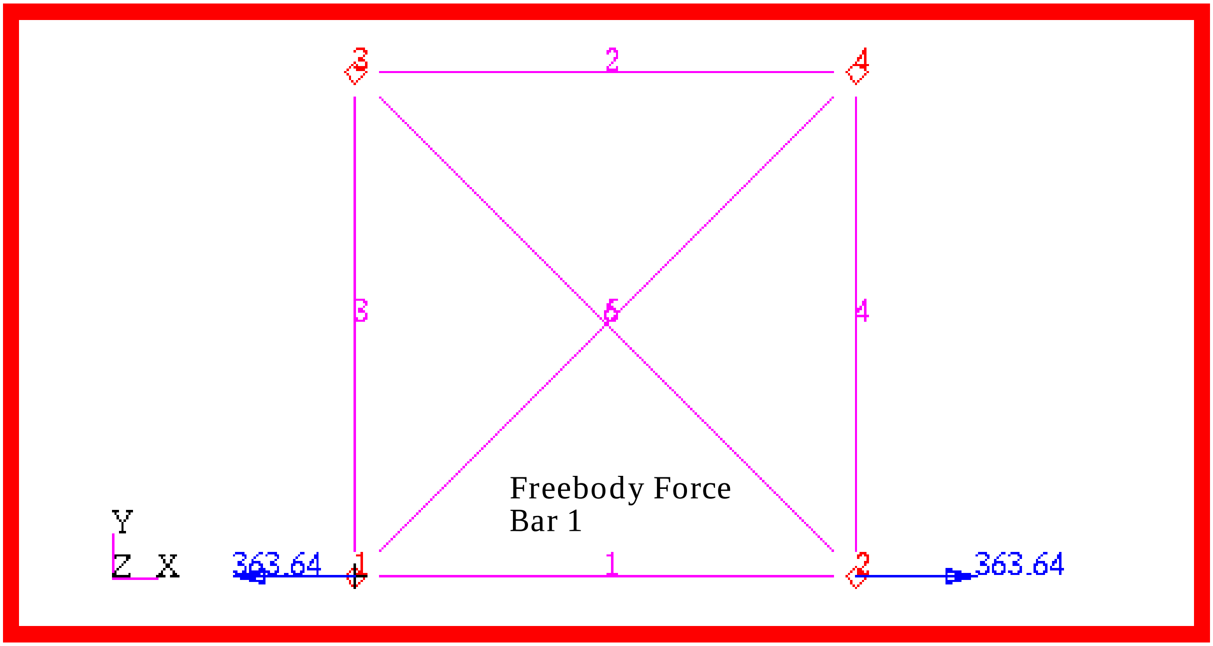


Figure 15-67 Freebody Forces in Bar 1.

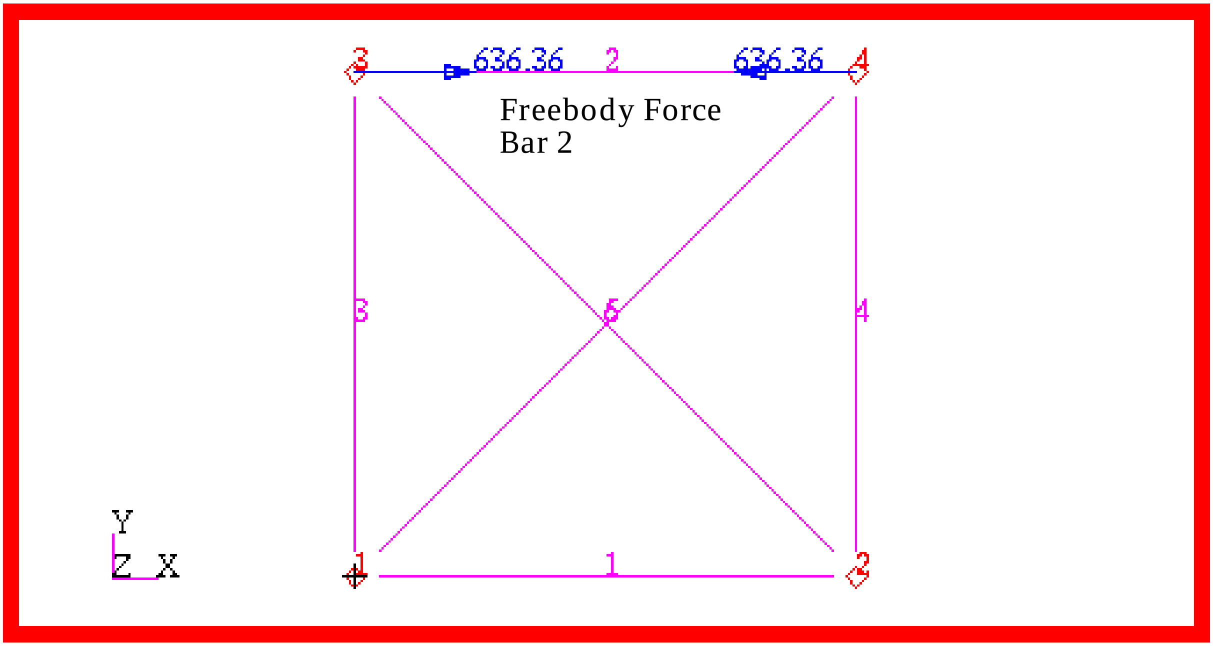


Figure 15-68 Freebody Forces in Bar 2.

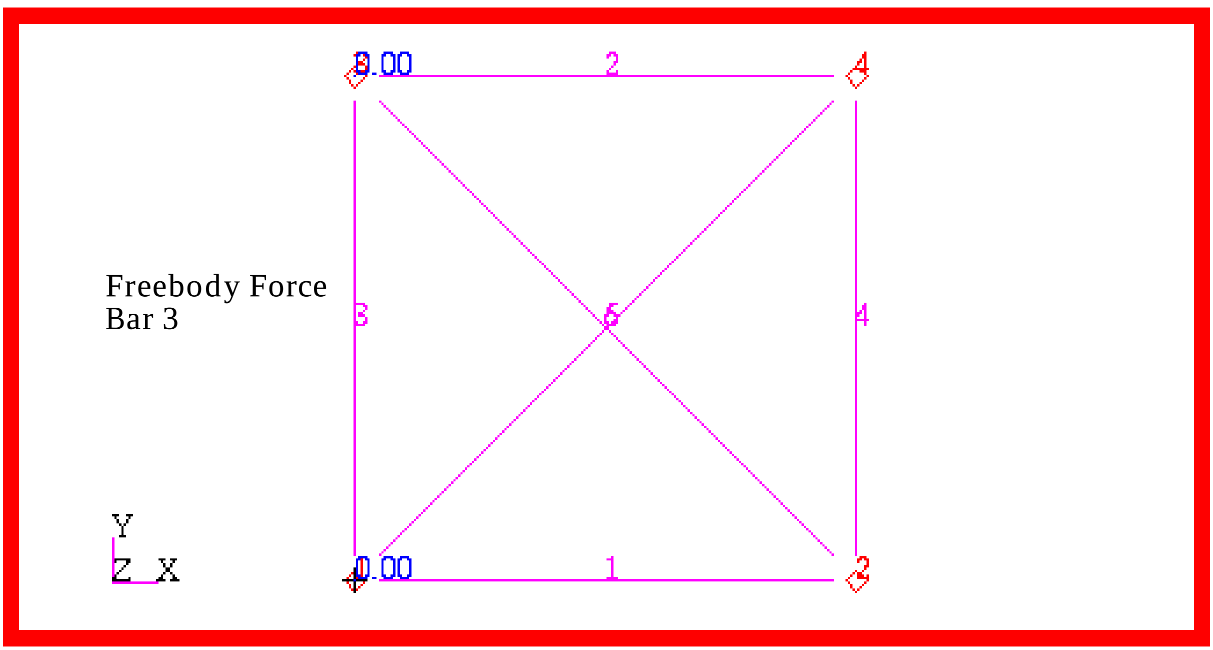


Figure 15-69 Freebody Forces in Bar 3.

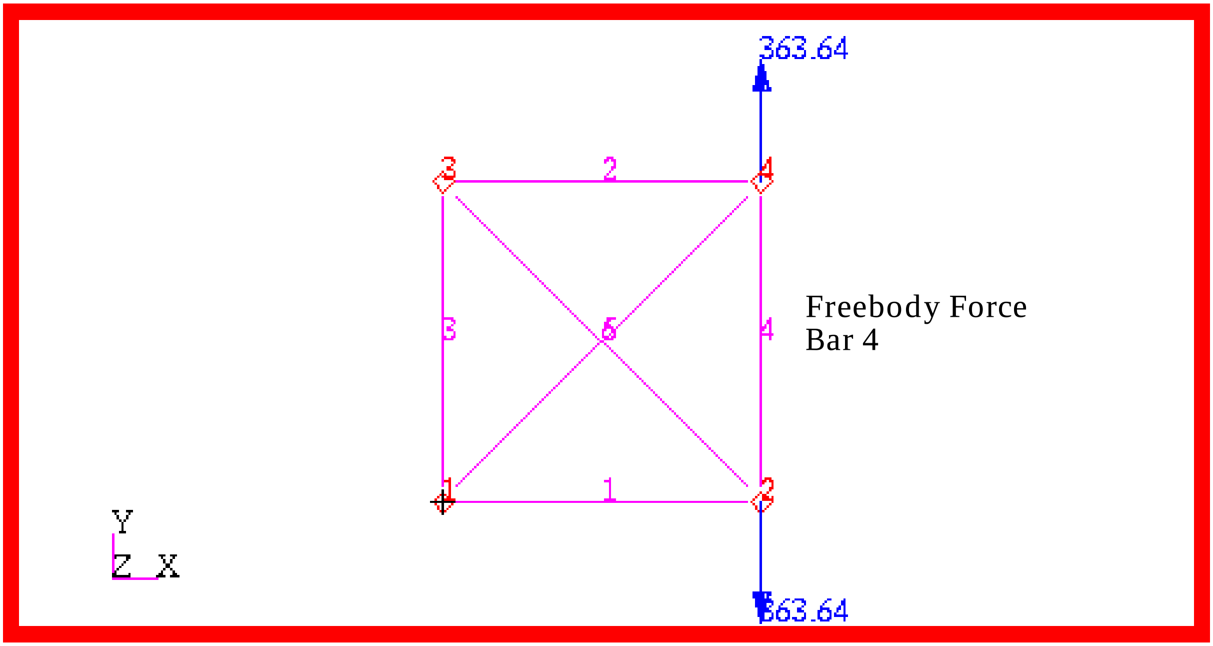


Figure 15-70 Freebody Forces in Bar 4.

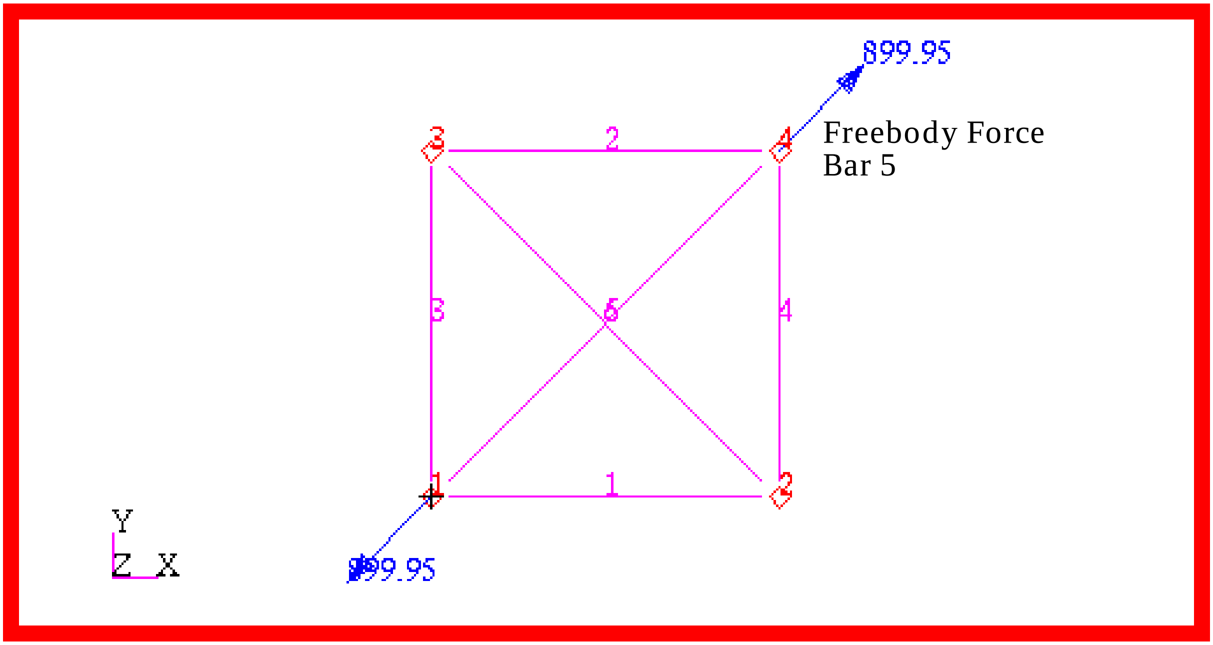


Figure 15-71 Freebody Forces in Bar 5.

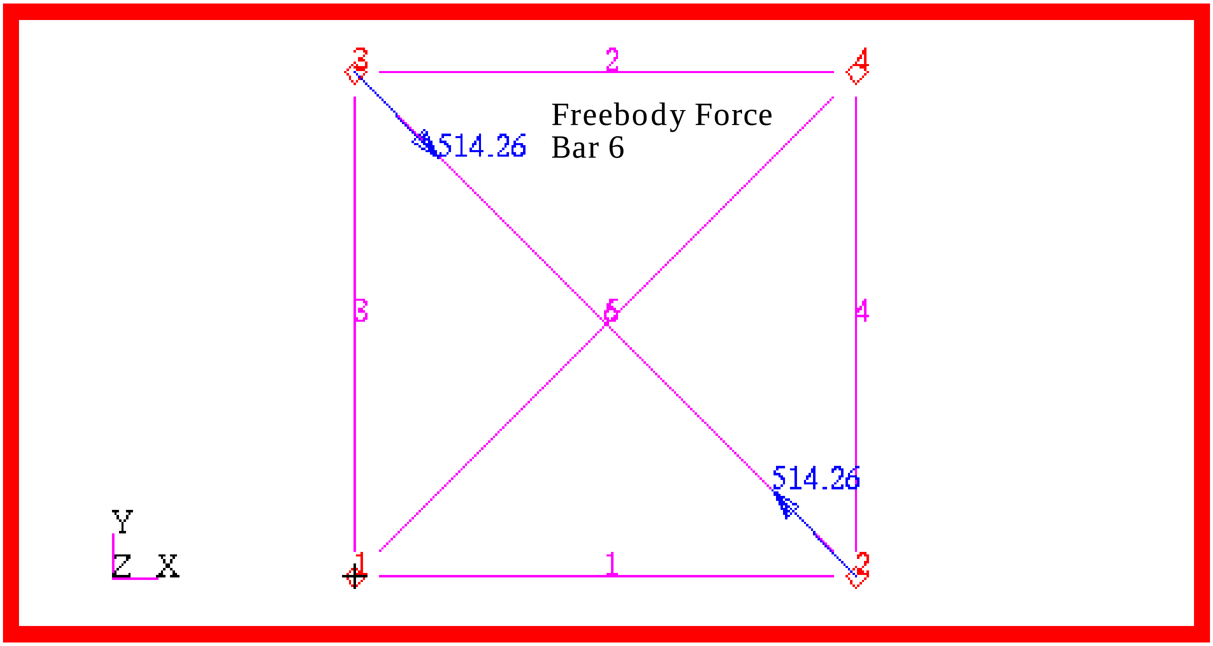


Figure 15-72 Freebody Forces in Bar 6.

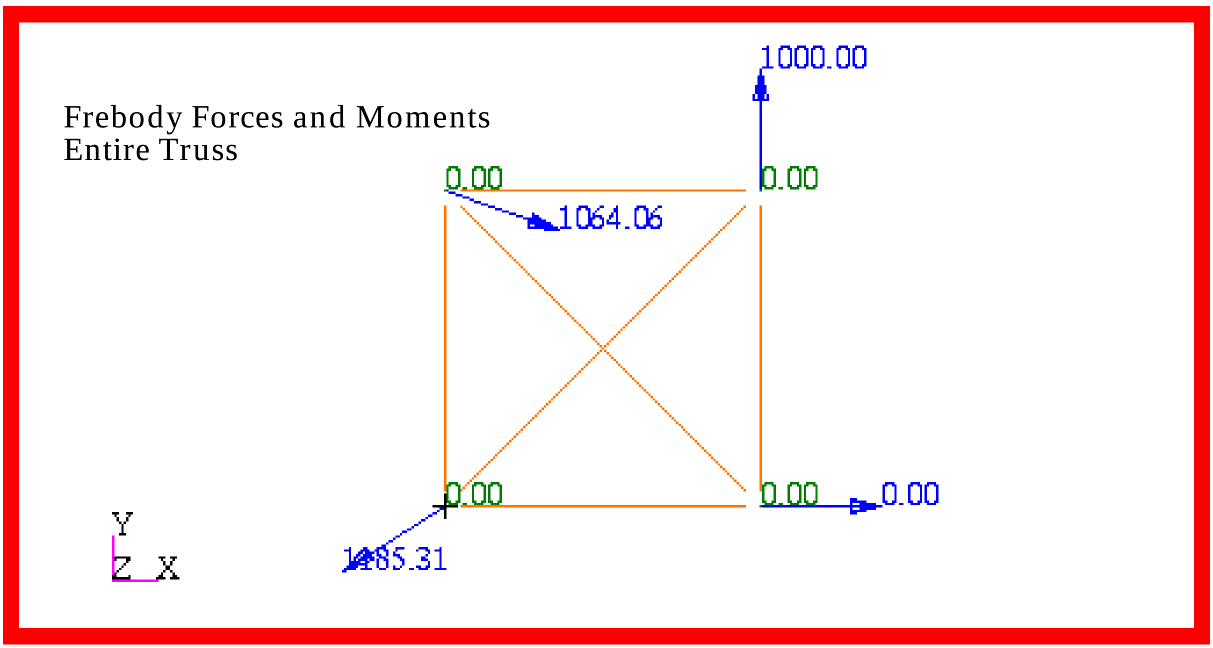


Figure 15-73 Freebody Forces and Moments for Entire Truss.

I N D E X

MSC.Patran Reference Manual
Part 6: Results Postprocessing

A

B

C

D

E

F

G

H

I

L

M

N

U

V

X