SAP2000®

Integrated
Finite Elements Analysis
and
Design of Structures

GRAPHIC USER INTERFACE MANUAL



Computers and Structures, Inc.
Berkeley, California, USA

Version 7.0
August 1998

COPYRIGHT

The computer program SAP2000 and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *support@csiberkeley.com*
web: *www.csiberkeley.com*

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMP-TIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table of Contents

New Interface **13**
Noun-Verb **13**
2 Modes of SAP2000 ..... **13**
Select Mode **13**
Setup Coordinate System **14**
Frame Element **14**
Shell Element **15**
FINT = F intermediate **15**
Asolid Element **15**
SINT = F intermediate **15**
Solid Element **16**
Nonlinear Analysis **16**
Nonlinear Static Pushover Analysis **17**
See Also 18
Getting a Head Start With Templates **19**
Import/Export Capabilities **20**
Import a SAP90 text input file **20**
Import a SAP2000 text input file **20**
Export a SAP2000 text input file **21**
Import a DXF file **21**
Export a DXF file **22**
Import a SAP2000 Job Run in DOS mode **23**
Export an AVI file **23**
Export an Enhanced Metafile **23**
View Real Time Animation for Time History Results **24**
Print Setup **24**
Print Selected Graphical Output to a Printer or a File **24**
Text Output **25**
Print Output to a Printer or a File **25**
Print Design Results to a Printer or a File **26**
Undo and Redo Capabilities **27**
Move **28**
Replicate **28**
Replicate in a Linear Array **28**
Replicate in a Radial Array **28**
Replicate in a Radial Array by Shifting the Origin **29**
Replicate by Using the Mirroring Option **29**
Merge Joints **30**
Automatic Meshing **30**
Divide or Break Frames **31**
Mesh Shells **31**
Join Frames **32**
Disconnect **32**
Connect **32**
Show Duplicates **32**
Joint and Element Labels **33**
Re-Label Previously Assigned Labels **33**
Select 3-D Views **34**

Select 2-D Views **34**
Set Elements **35**
Set Limits **35**
Define Materials **37**
Add a New Steel Material Type **37**
Add a New Concrete Material Type **38**
Add a New Other Material Type **38**
Define Section Properties **39**
Define Frame Section Properties **39**
Add a Frame Section by defining the dimensions and or properties manually **40**
Add a Nonprismatic Frame Section **41**
Add an Auto Select Frame Group **42**
Add an NLLink Property **42**
NL Damper, Gap and Hook Properties **43**
NL Plastic 1 Properties **43**
NL Isolator 1 Properties **44**
NL Isolator 2 Properties **44**
Define Static Pushover Cases **45**
See Also 45
Static Pushover Case Data **46**
See Also 46
Push To Load Level Defined By Pattern **47**
Save Positive Increments Only **47**
Restart When Unloading **48**
Push To Displacement Of **52**
Minimum and Maximum Saved Steps **52**
Event Force And Deformation Tolerances **53**
Maximum Failed Steps **53**
Load Pattern **54**
Define Frame Hinge Properties **54**
See Also 54
Default Hinge Properties **55**
Steel Axial Hinge **55**
See Also 55
Steel Moment Hinge and Steel P-M-M Hinge **56**
See Also 56
Steel Shear Hinge **56**
See Also 56
Concrete Axial Hinge **57**
See Also 57
Concrete Moment Hinge and Concrete P-M-M Hinge **57**
See Also 58
Concrete Shear Hinge **58**
See Also 58
Types Of Hinge Properties (Pushover) **58**
Frame Hinge Property Data **59**
See Also 60
Hinge Types **60**
Elastic and Strain-Hardening Slopes **60**
Frame Hinge Property Data for XXX **60**
See Also 61

Stress Versus Strain **62**
Frame Hinge Interaction Surface **62**
See Also 62
Concrete Interaction Surface **63**
Interaction Surface Definition **63**
See Also 63
Spreadsheet **64**
Define Group Names **64**
Define Static Load Cases **64**
Bridge Analysis **65**
Define Lanes **65**
Define Vehicles **66**
Standard Vehicles **67**
General Vehicle **70**
Define Vehicle Classes **70**
Define Bridge Response **71**
Define Moving Loads **71**
Define Joint Patterns **72**
Assign Dynamic Loads **72**
Define Time History Functions **72**
Define Time History Cases **73**
Define Response Spectrum Functions **74**
Define Response Spectrum Cases **74**
Define Load Combinations **75**
Draw Mode **76**
Draw Members **76**
Reshape Element **77**
Add Special Joints **77**
Draw a Frame Element from Joint to Joint **78**
Draw a Quick Frame Element **78**
Draw a Shell Element between 4 Joints **79**
Draw a Rectangular Shell Element **79**
Draw an NLLink Element **80**
Edit Grids **80**
To Move a Grid **80**
To Add a new Grid **81**
To Delete a Grid **81**
Snap Tools **81**
Snap to Midpoints and Ends **82**
Snap to Intersections **82**
Snap to Perpendicular **83**
Snap to Lines and Edges **83**
Drawing Constraints **84**
Selection Procedures **86**
Select Objects by Window **86**
Select Objects by Intersecting Line **87**
Select Objects by 2D Planes **87**
Select Objects by Groups **87**
Select Objects by Frame Sections **87**
Select Objects by Shell Sections **87**
Select Objects by NLLink Properties **88**

Select Objects by Constraints **88**
Select Objects by Labels **88**
Select All Objects **88**
Assign Options **89**
Assign Section Properties **89**
Assign Joint Constraints **90**
Add Joints to an existing Constraint **90**
Delete or Remove Joints from an existing Constraint **90**
Generalized Constraints **91**
Body Constraint **91**
Diaphragm Constraint **92**
Plate Constraint **92**
Rod Constraint **93**
Beam Constraint **93**
Equal Constraint **94**
Partial Connection **95**
Local Constraint **95**
Symmetry About a Plane **96**
Anti-symmetry About a Plane **96**
Axisymmetry **96**
Cyclic symmetry **97**
Symmetry About a Point **97**
Assign Joint Masses **98**
Assign Joint Restraints **98**
Assign Joint Patterns **98**
Assign Local Axes **99**
Assign Local Axes for Joints **99**
Assign Local Axes for Frame Elements **100**
Assign Frame Releases **100**
Assign Frame End Offsets **101**
Assign Frame Output Segments **101**
Assign Frame Prestress **102**
Assign Frame Initial P-Delta Force **102**
Assign Frame Lanes **103**
Assign Frame Hinges (Pushover) **103**
See Also 103
Relative Distance **104**
Hinge Property **104**
Assign Prestress to Frame Static Load **104**
Assign Local Axes for Shell Elements **105**
Assign Local Axis For NLLinks **105**
Assign Static Loads **105**
Assign Loads or Displacements to Joints **106**
Assign Gravity Loads to Frames **106**
Assign Trapezoidal Loads to Frames **107**
Assign Temperature Loads to Frames **107**
Assign Gravity Loads to Shells **107**
Assign Uniform Loads to Shells **108**
Assign Pressure Loads to Shells **108**
Assign Temperature Loads to Shells **108**
Assign Gravity Loads to NLLinks **109**

Display Options **112**
Graphical Output **113**
Displaying Undeformed Geometry **113**
Display Static Loads **113**
Display Joint Patterns **113**
Display Bridge Lanes **114**
To graphically display lanes: **114**
Display Input in Tabular Format **114**
Display Mode Shape **115**
Display Member Force or Stress Diagram **115**
Display Energy Diagrams **116**
View Time History Results **116**
Display Input Time History Functions **117**
Display Joint Output Time History Trace **117**
Display Frame Element Force Output Time History Trace **119**
Display Shell Element Stress Output Time History Trace **120**
Display Structural Energy Time History Trace **121**
Display Base Function Time History Trace **122**
Display Group Summation Forces Time History **122**
F(t) vs. F(t) **123**
Scale Factor, Line Type and Color **124**
Zoom into the Plot **124**
Print Time History or Response Spectrum Plots or Tables **124**
View Generated Response Spectra Curves **125**
Spectrum Generation Define Tab **125**
Spectrum Generation Axes Tab **125**
Spectrum Generation Options Tab **126**
Spectrum Generation Frequency Tab **126**
Spectrum Generation Damping Tab **126**
Display Group Joint Force Sums **126**
Display Frame Influence Lines **127**
Display Joint or Member Text Output on Screen **128**
Pushover Curve **128**
See Also 130
Structural Behavior Type / Kappa Factor **130**
Viscous Dampers **130**
Single Demand Spectrum (Variable Damping) **130**
Override Axis Labels And Range **131**
See Also 131
Override Structural Behavior Type **131**
See Also 132
Kappa Factor **132**
Acceleration-Displacement Response Spectrum (ADRS) **132**
Color Coded Hinge State **133**
Step-By-Step Hinge Formation **133**
Step-By-Step Pushover Forces **133**
Steel Design **134**
Group Elements for Design **135**
Start Design/Check of Structure **135**
Selecting Design Load Combinations **136**
Overwrite Element Design Data **136**

Defining Elements as a Single Member for Bending 137
Display Design Input/Results on Screen 138
Display Steel Stress Check/Design Ratios on Screen 138
Display Concrete Design/Check Output on Screen 138
Display Steel Stress Check/Design Input on Screen 138
Display Concrete Design/Check Input on Screen 139
Review Stress Check/Design Details 139
Review Concrete Design/Check Detail 139
View Interaction Diagram for a Concrete Column 140
Redesign a Steel Member 140
Redesign a Concrete Member 141
Update Analysis Sections 141
Reset Design Sections 141
Preferences 142
Setting Steel Design Parameters 142
Setting Concrete Design Parameters 143
Setting Colors 143
Editing Toolbars 143
Aerial View 144
End 144
General Tips 145
Specific Tips 146
SAP2000 Versions and Limitations 146
Error Messages 146
Phone and Fax Support 146
Online Support 147
Frequently Asked Questions 147
Program Manuals 148
Primers 149
Newsletters 149
Text Books 149
References 149
End User License Agreement for CSI Software 152
Questions and Answers about the End User License Agreement 154
Can I sell my end user license agreement for SAP2000 to someone else? 154
Can I sell or give away old versions of SAP when I acquire an upgrade? 154
Can I make a second copy for my personal use? 154
Can I rent or lease the Program to someone else? 154
When I upgrade from SAP90, do my license rights for SAP2000 change? 155
In what ways can I use the software over a network? 155
Modal Combinations 156
EIGENVECTORS 156
RITZ 156
Rigid-body behavior 156
Equal-displacement behavior 156
Symmetry Conditions 156
Anti-symmetry Conditions 157
Von Mises Stress 157
Analysis Cases 157
Load Combination Type 157
Linear Time History Analysis 157

Periodic Time History Analysis 158
NonLinear Time History Analysis 158
Ground Acceleration 158
UBC94 Response Spectrum 158
Energy Functions 158
Base Functions 158
DXF 158
Joint Stress Averaging 159
Paste Coordinates 159
Insertion Point 159
Coordinate System 160
Set Coordinate System 160
Coordinate System (Advanced) 161
Shell Element (4 Joints) 161
Shell Element (4 Joints 3-D) 161
Shell Element (Quick) 161
Frame Element (2 Joint) 162
Frame Element (2 Joint 3-D) 162
Side Toolbar 162
SAP2000 Screen 163
Dimension Tolerances 163
Concrete Design Parameters 164

GENERAL

New Interface

1. SAP2000 is now fully integrated into Windows.
2. Model building, Analysis, Design and Display of results may be carried out in the same window.
3. The model may be viewed in multiple windows (up to 4).
4. Zooming is possible either in steps or with a mouse defined window 
5. Members may be extruded on their centerlines.
6. The model may be viewed in Perspective 
7. Context sensitive help is available in the “forms” with a right mouse button click.
8. Detailed information about joints and members in the model is also available with a right mouse button click (e.g. bending moment diagrams, joint displacements or connectivity etc.)

Designer’s Convention

The designer’s sign convention is quite different from the typical structural analysis sign convention described in most text books. The moment which produces compression in the positive face/fiber is called a positive moment and the axial force which produces tension in the member is called a positive axial force.

Noun-Verb

All commands in SAP2000 work in the “Noun-Verb” mode i.e. you make a selection first and then perform an action on it. As an example, if you want to delete a member from the model you would first select that member and then press the delete key.

2 Modes of SAP2000

SAP2000 has two very distinct modes i.e. DRAW and SELECT. SELECT mode is the default mode. DISPLAY and ASSIGN functions are carried out in the SELECT mode.

Select Mode

When SAP2000 is not in the DRAW mode, SELECT is the default mode. It is possible to make multiple selections and then perform a task on the selected entities. Display, Assign, Design, Output or even Delete functions are carried out in this mode.

Tell me more about

 Selection Procedures

Display Options
Assign Options

Setup Coordinate System

To set up a Coordinate System for creating a new model

1. On the File menu, click New Model .
2. Choose either the Cartesian or the Cylindrical tab.
3. Type in the number of grids and their spacing and then click OK.

To add another Coordinate System

1. On the Options menu, click Set Coordinate System.
2. Click Add new System.
3. Choose either the Cartesian or the Cylindrical tab.
4. Type in System Name, to define the new Coordinate System.
5. Type in the number of grids and their spacing.
6. Click on Advanced to specify Location and Orientation.
7. Click Ok, Ok, and OK.

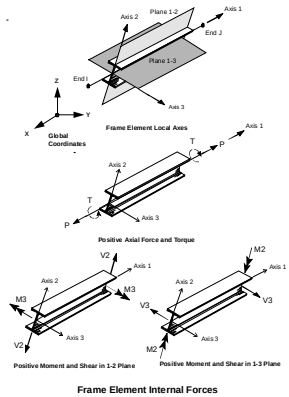
Tip: You can add a new grid by selecting Reshape Element from the tool bar, pressing the CTRL key and dragging an existing grid to a new location, or by using the Edit Grid command from the Draw menu. Double clicking on an existing grid opens the Edit Grid box where you may add or modify the location of a grid.

Also See

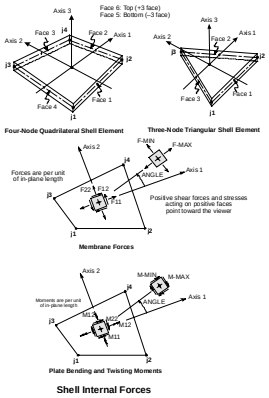
Edit Grids

Frame Element

FRAME element follows the designer's convention.



Shell Element

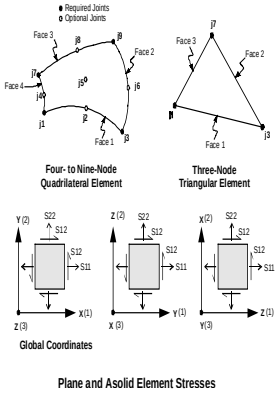


FINT = F intermediate

FVM =√{1/2[(FMAX - FINT)^2+(FMAX - FMIN)^2+(FINT - FMIN)^2]}.

Also See: Von Mises Stress

Asolid Element

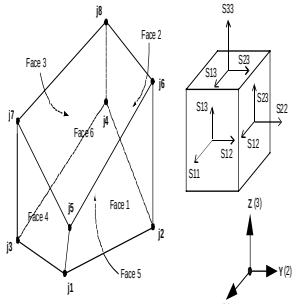


SINT = F intermediate

SVM =√{1/2[(SMAX - SINT)^2+(SMAX - SMIN)^2+(SINT - SMIN)^2]}.

Also See: Von Mises Stress

Solid Element



Solid Element Stresses

Nonlinear Analysis

Nonlinear analysis capabilities are available through a nonlinear link element (NLLink). This link element is used to model local structural nonlinearities such as gaps, dampers, isolators, and the like. Nonlinear behavior is exhibited only during nonlinear time-history analyses. For all other analyses, this link element behaves linearly.

Each nonlinear link element may be either:

- a one-joint grounded spring, or
- a two-joint link

Properties for either type of element are defined in the same way.

Each element is assumed to be composed of six separate nonlinear springs, one for each of the six deformational degrees-of freedom. The force-deformation relationships of these springs may be coupled or independent of each other.

The types of nonlinear behavior that can be modeled with the link element include:

1. Viscoelastic damping
2. Gap (compression only) and hook (tension only)
3. Uniaxial plasticity
4. Biaxial-plasticity base isolator
5. Friction-pendulum base isolator

Each element has its own local coordinate system for defining the force-deformation properties and for interpreting output.

Each NLLink element may be loaded by gravity (in any direction).

Available output includes the deformation across the element, and the internal forces at the joints of the element.

Nonlinear Static Pushover Analysis

Nonlinear static pushover analysis capabilities are provided only in the nonlinear version of SAP2000. The nonlinear behavior occurs in discrete user-defined hinges. Currently hinges can only be introduced into frame elements; they can be assigned to a frame element at any location along that element. Uncoupled moment, torsion, axial force and shear hinges are available. There is also a coupled P-M2-M3 hinge which yields based on the interaction of axial force and bending moments at the hinge location. More than one type of hinge can exist at the same location, for example, you might assign both a M3 (moment) and a V2 (shear) hinge to the same end of a frame element. Default hinge properties are provided based on ATC-40 and FEMA-273 criteria.

A pushover analysis can consist of more than one pushover load case. Each pushover load case can have a different distribution of load on the structure. For example, a typical pushover analysis might consist of three pushover load cases. The first would apply gravity load to the structure, the second would apply one distribution of lateral load over the height of the structure, and the third would apply another distribution of lateral load over the height of the structure. There are four different methods of describing the distribution of load on the structure for a pushover load case:

1. A uniform acceleration can be automatically applied. In this case, the lateral force automatically applied at each node is proportional to the mass tributary to that node.
2. A lateral force that is proportional to the product of a specified mode shape times its circular frequency squared (ω^2) times the mass tributary to a node can be automatically applied at each node. You may specify the mode shape to be used in this instance.
3. You may define an arbitrary static load pattern.
4. You may combine any of the methods described in 1, 2 and 3 above.

Several types of output can be obtained from the nonlinear static pushover analysis:

1. Base shear versus displacement at a specified control joint can be plotted.
2. Base shear versus displacement at a specified control joint can be plotted in the ADRS format where the vertical axis is spectral acceleration and the horizontal axis is spectral displacement. The demand spectra can be superimposed on this plot.
3. The sequence of hinge formation and the color-coded state of each hinge can be viewed graphically, on a step-by-step basis , for each step of the pushover.
4. The member forces can be viewed graphically, on a step-by-step basis, for each step of the pushover.
5. Tabulated values of base shear versus displacement at each point along the pushover curve, along with tabulations of the number of hinges beyond certain control points on their hinge property force-displacement curve can be viewed on the screen, printed, or saved to a file.
6. Tabulated values of the capacity spectrum (ADRS capacity and demand curves), the effective period and the effective damping can be viewed on the screen, printed, or saved to a file.

The following general sequence of steps is involved in a nonlinear static pushover analysis:

1. Create a model just like you would for any other analysis.
2. Define arbitrary static load patterns, if needed, for use in the pushover analysis (Define > Static Load Cases). Note that the program also has built-in capability to define the distribution of lateral load over the height of the structure based on both uniform acceleration, and mode shapes.
3. Define the pushover load cases (Define > Static Pushover Cases).
4. Define hinge properties (Define > Hinge Properties)
5. Assign hinge properties to frame elements (Assign > Frame > Hinges (Pushover)).
6. Run the pushover analysis (Analyze > Run Static Pushover). The Run Static Pushover option will be active only if there is at least one frame element with a hinge property assigned to it, and there is at least one pushover load case defined. If frame elements are specified to be designed by the program, this design automatically will be performed before the pushover analysis routine begins.
7. Review the pushover results (Display > Show Static Pushover Curve).
8. If necessary, revise the model and repeat steps 2 through 8.

See Also

Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Pushover Curve
Override Axis Labels And Range

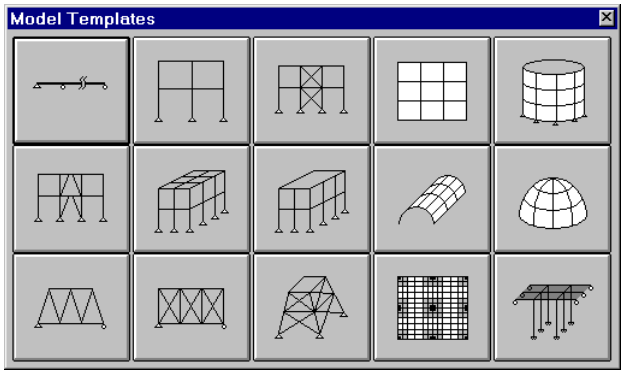
Override Structural Behavior Type

FILE MENU

Getting a Head Start With Templates

New models may be created with very little effort using pre-programmed templates.

Click on any of the template buttons to learn more about them.



To Base a new model on a template:

1. On the File menu, click New Model from Template… This will display the possible templates shown above.
2. Click on a template most appropriate for your model.
3. Choose or change the default values in the edit boxes.
4. Click OK.

To add one or more templates to the existing model:

1. On the Edit menu, click Add to Model From Template… This will display the template options similar to the one shown above.
2. Click on the template you want.
3. Fill in the parameters in the edit boxes and then click on Advanced to specify an insertion point.
4. Click OK.

Note: Remember to save your model periodically. Click on .

Import/Export Capabilities

Import and Export are available through the File pull down menu. SAP2000 now works around a central database created for the model. The database is maintained in a binary format and does not operate with text input files. However, text input files may be read or written using the Import Export facility.

What do you want to do?

- Import a SAP90 text input file
- Import a SAP2000 text input file
- Export a SAP2000 text input file
- Import a DXF file
- Export a DXF file
- Import a SAP2000 Job Run in DOS mode
- Export an AVI file
- Export an Enhanced Metafile

Import a SAP90 text input file

1. On the File menu, click Import and then SAP90. This will display the Open SAP90 Data File dialog box.
2. Choose the file from the list box (extension not allowed).
3. Click Open.

Import a SAP2000 text input file

1. On the File menu, click Import and then SAP2000.S2K. This will display the Open SAP2000 Data File dialog box.
2. Choose the file from the list box (must have either .S2K or .$2K extension).
3. Click Open.

Caution: The text file with an extension . $2k is automatically written out whenever the model is saved. This file is good for analysis purposes only or recovering the model in emergency crash situations and is not a substitute for the database file (.SDB). It does not contain all the information about the model.

Note: The analysis and design information provided in the SDB file is also saved in the S2K file. This gives the user the ability to modify the model using a text editor like WordPad. To find the format used in the S2K file for the design data. Create a frame element in SAP2000 with the design information that you desire. Export the model and open it in a text editor to see the format used for the defined design information in the S2K file.

Export a SAP2000 text input file

1. On the File menu, click Export and then SAP2000.S2K. This will display the Save Model File As dialog box.
2. Type in the file name in the File name edit box.
3. Click Save. This will save the text input file with an extension *filename*.S2K.

Note: A text file with an extension *filename*. $2k is also saved at the same time as the model database is saved (as a file with .SDB extension). This text file is good for analysis purposes only or for recovering the model in emergency crash situations and is not a substitute for the database file. It does not contain all the information about the model.

Import a DXF file

The DXF file import is intended to facilitate importing model geometry from AutoCAD and other DXF compatible programs. DXF for AutoCAD r12, r13 and r14 is supported.

1. Start a new model or open an existing model. You can not import a DXF file if you do not have a model or coordinate system defined first.
2. From the File menu, click Import and then .DXF. This will display the Import DXF File dialog box.
3. Choose the file from the list box and press the Open button.
4. From the Import Data dialog box select the Global Up direction of the DXF file elements being imported and the units that the file was drawn in.
5. Press the OK button in the Import Data dialog box to see the DXF Import dialog.
6. Select from the dropdown list boxes the layer to import from the DXF file for each SAP2000 element type. A layer can only be used for one element type.
7. If there is no data to be imported for a given element type, then the corresponding list box should read NONE
8. Click the OK button to import the DXF data.

The SAP2000 element types correspond to the following DXF element types.

SAP2000 Elements	**DXF Elements**
Grids	Lines – Currently Not Active
Special Joints	Points
Frames	Lines
NL Links 1-Noded	Points
NL Links 2-Noded	Lines
Shells	3Dfaces
Plane elements	3 to 8 sided Polyline – Currently Not Active
Asolids	3 to 8 sided Polyline – Currently Not Active
Solids	Solids NOT 3Dsolds and Meshed Polylines

Note: Point and Line elements can be on the same NL Link layer.

The imported DXF objects are assigned to a new group called DXFINx to facilitate editing of the objects.

The SOLID entity can be drawn by selecting 2D Solid Surface from the Draw menu. The Meshed Polyline can be drawn by typing “3d” in the Command line and entering a Box or Wedge.

Export a DXF file

The DXF file Export is intended to facilitate exporting geometry data into AutoCAD and other DXF compatible programs. DXF for AutoCAD r12, r13 and r14 is supported.

1. If you want to export a limited number of elements then select the elements first; otherwise go directly to step 2.
2. From the File menu, select Export and then .DXF.
3. This will display the Select Template DXF file. The Template DXF file is used to set up all the basic data for the exported DXF file. You can select any DXF file that you want. Then press the Open button.

Note: For your convenience there is a file SAPDXF.DXF provided for you in the SAP2000 directory.

4. This will display the DXF Export dialog box, which is used to select the layer to export to, for each element type. The layer names are the same as the ones in the Template DXF file.
5. Select from the dropdown list boxes the layer to export to in the DXF file. A layer can only be used for one element type.
6. If you want to add another layer name, type its name in the text box at the bottom of the dialog box and press the Add Layer button. Then the new layer can be selected from the drop down list boxes.
7. If there is no data to be exported for a given element type then the corresponding list box should read NONE.
8. Press the OK button when finished with the layer assignments.
9. This will display the Save AutoCAD DXF File As dialog box.

10 Type in the file name in the File name edit box.

11 Click Save. This will save the DXF file, which may be imported into AutoCAD.

The SAP2000 element types correspond to the following DXF element types.

SAP2000 Elements	DXF Elements
Grids	Lines – Currently Not Active
Joints	Points
Frames	Lines
NL Links 1-Noded	Points
NL Links 2-Noded	Lines
Shells	3Dfaces
Plane elements	3 to 8 sided Polyline – Currently Not Active
Asolids	3 to 8 sided Polyline – Currently Not Active
Solids	Meshed Polylines
Text	Text

Note: Point and Line elements can be on the same NL Link layer.

The template file can be used to set up all the layer names, colors, line types and so on so that the exported DXF files are consistent. The Template DXF file can be any AutoCAD r13 or r14 file. There is a file SAPDXF.DXF file provided in the SAP2000 directory.

The text will only be exported if the current view is a 2D view with visible text. The text will be exported and displayed in the same plane as the 2D view.

Hint: If you want to only export a 2D plane of the model then set the 2D view that you are interested in. Select all the elements that are visible, and then select Export… DXF.

Import a SAP2000 Job Run in DOS mode

1 On the File menu, click Import and then SAP2000.JOB. This will display the Open SAP2000 File dialog box.

2 Choose the file from the list box (with extension .JOB).

3 Click Open.

Export an AVI file

1 On the File menu, click Create History Video… This will display the Time History Video File Creation dialog box.

2 Click on AVI File Name button to change the AVI file name or accept the default name.

- In the Time History Data area:
  - Select the History Name from the drop down list box.
  - Specify the Start and End Time and the Time Increment in the edit boxes.
- In the Display Options area:
  - Click to check if Wire Shadow is to be retained in the Video file capture.
  - Click to check if Cubic Curve is to be used.
  - Type in the Magnification Factor in the edit box.
- In the AVI Options area:
  - Type in the animation speed in the Frames per Second edit box or accept the default value.
  - Specify the Frame size in Pixels in the edit boxes or accept the default values.

4 Click OK

Note: The AVI file may be played back using Windows Media Player.

Tip: To play the AVI file in **real time animation** make sure that the number of frames per second is equal to 1/Time Increment. The value of number of frames per second may be adjusted to speed up or down the animation

Export an Enhanced Metafile

The Enhanced Metafile is a vector file format that can be read by many graphics programs as well as some office tools like word processors.

1. On the File menu, click Export and then Enhanced Metafile. This will display the Save Enhanced Metafile dialog box.
2. Type in the file name in the File name edit box.
3. Click Save. This will save the text input file with an extension *filename*.EMF.

View Real Time Animation for Time History Results

1. On the File menu, click Create History Video… This will display the Video File dialog box.
2. Type in a file name or accept the default name.
3. Click Save. This will display the Time History Video File Creation dialog box.
4. Select the Time History case from the History Name drop down box.
5. Enter the Start Time, End Time and Time Increment in the edit boxes.
6. Checking the Wire Shadow box will also display the undeformed shape as a reference.
7. Checking the Cubic Curve box will display the deformed elements with cubic curve fit.
8. Changing the Magnification Factor will scale the vector components. The default factor is 10.
9. Enter Frame Speed in frames per second for animation and the Frame Size in Pixels. The default for Frame Speed is 10 and Frame Size is 320x240.
10. Click OK. Creating AVI File window will appear and the animation will be saved as an AVI file which may be played back through the Windows Media Player.

Note: If you want to view real time animation then the value for frames per second should be the same as 1/Time Increment. A larger value will speed up the animation and a smaller value will slow it down.

Print Setup

Print Setup is where you can set number of lines of text to be printed per page, project information and notes to be printed with text and graphics output. You can also do regular printer setup.

1. From the File menu select Print Setup.
2. In the Print Page Setup dialog box:
   - Check the No Page Eject if you do not want SAP2000 to break the printed text into pages and place page headings.
   - If you want to have SAP2000 break the printed text into pages and place page headings then:
     - Select Default to use the default number of lines per page
     - Or select User Defined and enter in the provided text input box the number of lines per page.
   - Enter under the Titles area, Project and other comments in the provided text input boxes.
   - Click the Setup button to set printer options. (See windows documentation for instructions on using this dialog box.)
   - Click OK on the Print Page Setup dialog box when finished.

Print Selected Graphical Output to a Printer or a File

1. Plot the desired deformed or undeformed shape on the Screen.
2. On the File menu, click Print Graphics...
3. The plot will be printed to the default printer. If the default Printer has been set to print to a file then the file

dialog box will be displayed.

Note: If more than one Window is open the highlighted window will be printed.

Text Output

Text output for a Joint or a member for the chosen Load Case(s) and/or Load Combination(s) may be either viewed on the screen or printed to a printer or a file.

What do you want to do?

Display Joint or Member Text Output on Screen

Print Output to a Printer or a File

Print Input Tables to a Printer or a File

1. Select the members or Joints for which the input data is desired. If no members are selected the input data is generated for all the members in the model.
2. On the File menu, click Print Input Tables... This will display the Print Input Tables dialog box.
3. In the Print Input Tables dialog box:
   - Check the information you want printed.
   - If you want to generated output for all the members in the model even though some members were selected then remove the check for Selection Only check box.
   - If Output to a File is desired check the Print to File check box.
     - If you want the output appended to an existing file check the Append check box.
   - Click OK.

Print Output to a Printer or a File

1. Select the members or Joints for which the output is desired. If no members are selected the output is generated for all the members in the model.
2. On the File menu, click Print Output Tables... This will display the Print Output Tables dialog box.
3. In the Print Output Tables dialog box:
   - Click on Select Loads to select Load Cases or Combos.
   - Select the Load Cases and/or the Load Combinations labels for which the output is to be displayed in a tabulated form. Loads may be selected by clicking on the Load labels in the list box. For selecting a range of loads, click and drag, or for multiple loads hold down the Ctrl Key and click on different load labels.
   - Click OK to close the Select Output dialog box.
   - Click on the check boxes to select the Type of Analysis Results desired, e.g. Displacements, Reactions, Spring Forces, Frame Forces, NLLink Forces, Shell Resultants, Shell Stresses, Plane Stresses, ASOLID Stresses and Solid Stresses, Element Nodal Forces, Group Forces.
     - If Group Forces is checked, you can use the Select Group button that appears to select predefined Groups for which you want forces reported.
   - If Output to a File is desired check the Print to File check box.
     - If Spreadsheet output format is desired check the Spreadsheet check box.
     - If you want the output appended to an existing file check the Append check box.
   - If only Envelopes of Load Combos are desired check the Envelopes Only check box.
   - If you want to generated output for all the members in the model even though some members are selected then remove the check for Selection Only check box.
   - Click OK.

Note: You can not select Envelope results in a Spreadsheet format.

Print Design Results to a Printer or a File

1. Select the members or Joints for which the design results are desired. If no members are selected the design results are generated for all the designed members in the model.
2. On the File menu, click Print Design Tables... This will display the Print Design Tables dialog box.
3. In the Print Design Tables dialog box:
   - Check the information you want printed for steel and concrete members.

- If you want to generated output for all the members in the model even though some members were selected then remove the check for Selection Only check box.
- If Output to a File is desired check the Print to File check box.
  - If you want the output appended to an existing file check the Append check box.
- Click OK.

User Comments and Session Log

SAP2000 provides the user with a way to keep notes with the model file.

1 This is done by selecting from the File menu, User Comments and Session Log.

- This will bring up a text editor that works just like Notepad.

Uses for the User Comments and Session Log Text Editor:

- Keeping track of what has been done on the model to date.
- Keeping a To Do list for the model.
- Retaining key results to see the effect of changes on the model.
- Paste a Cut or Copied model into the text editor. Modifying the information, Cut/Copying it and then Pasting the modified model back into SAP2000.
- The Log file automatically provides a log of all file saves by the user.

EDIT MENU

Undo and Redo Capabilities

SAP2000 allows you to go back one step at a time in an editing session. Therefore, it is possible to Undo a series of actions previously performed. If you go too far in the Undo process you may Redo those actions.

1 Click  to **Undo** (in steps) the most recent actions performed. **Undo** is also available in the Edit menu.

If you later decide you didn't want to undo an action:

1 Click  to **Redo** the action(s). **Redo** is also available in the Edit menu.

Note: Undo only works on Objects which have been moved, deleted or added. Once a file is saved Undo and Redo actions are no longer available.

Cut, Copy and Paste

1 SAP2000 now supports standard Windows Cut, Copy and Paste commands.

2 The entire structure or any selected part may either be cut or copied and then pasted back into the model area at any user specified location.

3 It is also possible to Paste the Cut or Copied model into Microsoft Excel, modify, Cut/Copy in Excel and then Paste the modified model back in the SAP2000 window.

Also see:

Replicate

Deleting Members

1 You can delete members by selecting them and pressing the Delete button on your keyboard.

2 You can also delete members by selecting them and clicking on Delete From the Edit menu.

Move

The move feature is a way to select parts of the structure and relocate them in the model.

1 Select the members and joints you want to move.

2 On the Edit menu, click Move. This will display the Move dialog box.

- Enter the relative movement of the selected members in the global X, Y and Z direction.
- Click OK to move the selected elements.

Replicate

Replicate is another very powerful way of generating a large model from a small model when the elements and/or joints form a linear or radial pattern or are symmetrical about a plane. When joints or elements are replicated the assignments on those joints and elements are also replicated, e.g. member section assignments, member loads, joint loads and joint restraints. This is a major benefit of using Replicate over Cut, Copy and Paste, which only operates on lines and joints.

What do you want to do?

Replicate in a Linear Array

Replicate in a Radial Array

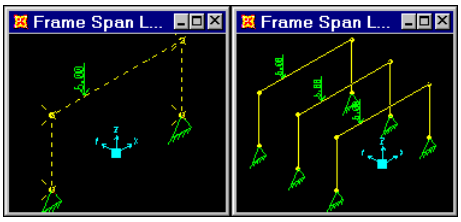
Replicate in a Radial Array by Shifting the Origin

Replicate by Using the Mirroring Option

Replicate in a Linear Array

1. Select the members and joints you want to replicate.
2. On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
3. Click on the Linear Tab.
4. Fill in the X, Y and Z offset distances in the Distance edit boxes.
5. Type in the number of times you want the selected entities replicated in the Number edit box.
6. Click OK.

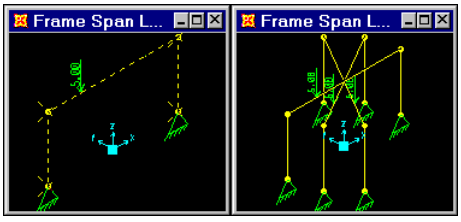
EXAMPLE:



Replicate in a Radial Array

1. Select the members and joints you want to replicate.
2. On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
3. Click on the Radial Tab.
4. Check the axis about which the selected entities are to be rotated, i.e. X, Y or Z in the Rotate About section.
5. Type in the increment angle and the number of times you want the selected entities replicated in the Increment Data box.
6. Click OK.

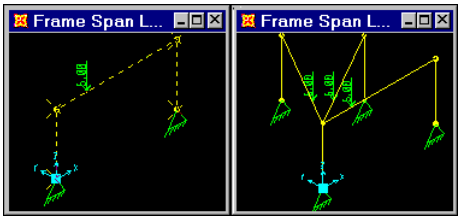
EXAMPLE:



Replicate in a Radial Array by Shifting the Origin

1. On the Options menu, click Set Coordinate System… This will display the Coordinate Systems dialog box.
2. Select the coordinate system from the Systems list box.
3. Click on Modify/Show System. This will display the Location and Orientation dialog box.
    - In the Location and Orientation dialog box
        - Type in the values by which you need to shift the coordinate system in Translations edit boxes and/or Rotations in Degrees edit boxes.
        - Click OK
4. Click OK
5. Select the members and joints you want to replicate.
6. On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
7. Click on the Radial Tab.
8. Check the axis about which the selected entities are to be rotated, i.e. X, Y or Z in the Rotate About section.
9. Type in the increment angle and the number of times you want the selected entities replicated in the Increment Data box.
10. Click OK.

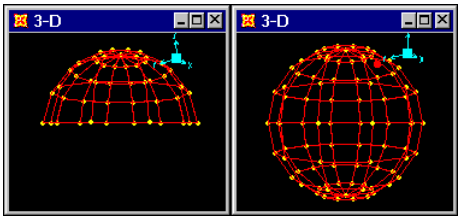
EXAMPLE:



Replicate by Using the Mirroring Option

1. Select the members and joints you want to replicate.
2. On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
3. Click on the Mirror Tab.
4. Check the plane about which the selected entities are to be rotated, i.e. XY, YZ or XZ in the Mirror About section.
5. Type in the ordinate by which you want to shift the mirrored replication in the Ordinate section.
6. Click OK.

EXAMPLE:



Merge Joints

Joints within the default Auto Merge Tolerance are automatically merged. Those Joints which fall outside this tolerance may be merged as follows:

1. Select the Joints you want merged.

2 On the Edit menu click Merge Joints… This will display the Merge Selected Joints dialog box.
3 Accept or change the Merge Tolerance.
4 Click Ok

Hints: This is a very useful command for merging two separate models together.

If elements are drawn with Snaps turned off there may be extraneous joints at common locations. These extraneous Joints may be easily eliminated by this command.

Models developed in CAD programs may have beam joints drawn 6 inches or so away from column joints. This is a powerful option to merge beam joints to column joints in such models imported from CAD.

Automatic Meshing

It is not necessary to define Joints prior to defining elements. Joints are automatically created when elements are drawn. Typically, draw the boundary of your structure and then use the following meshing techniques to create a detailed model.

What do you want to do?

» Divide or Break Frames
» Mesh Shells
» Join Frames
» Merge Joints
» Disconnect
» Connect
» Show Duplicates

Divide or Break Frames

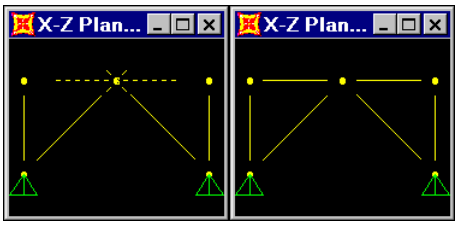
To Divide

1 Select the Frame Elements you want divided.
2 On the Edit menu click Divide Frames… This will display the Divide Selected Frames dialog box.
3 Select Divide into option.
4 Type in the number of elements the Frame Elements should be divided into.
5 Type in the ratio of the last element to the first, if the division is not to be in equal lengths.
6 Click OK.

To Break

1 Select the Frame Elements you want to break into multiple elements and also select the intersecting Elements or Joints.
2 On the Edit menu click Divide Frames… This will display the Divide Selected Frames dialog box.
3 Select Break at intersections with selected Frames and Joints option.
4 Click Ok

EXAMPLE:



Mesh Shells

To Mesh

1. Select the Shell Elements you want meshed.
2. On the Edit menu click Mesh Shells. This will display the Mesh Selected Shells dialog box.
3. Select Mesh into option.
4. Type in the number of elements (in both directions) the Shell Elements should be meshed into.
5. Click OK.

To Mesh using selected Joints on edges

1. Select the Shell Elements you want meshed into multiple elements and also select the Joints on the edges of the Shell Elements.
2. On the Edit menu click Mesh Shells. This will display the Mesh Selected Shells dialog box.
3. Select Mesh using selected Joints on edges option.
4. Click Ok

To Mesh using grid intersections

1. Select the Shell Elements you want meshed into multiple elements.
2. On the Edit menu click Mesh Shells. This will display the Mesh Selected Shells dialog box.
3. Select Mesh at intersection with grids.
4. Click Ok

Note: In order for the mesh using selected Joints on edges to work you have to have an equal number of joints on the opposite edges. Also, if both sides are not divided this option will not work.

Join Frames

1. Select the Frame Elements you want joined.
2. On the Edit menu click Join Frames.
3. This will join the selected Frame Elements into a single element and remove unused joints left over from the joining process.

Disconnect

All elements connected to each other normally share a common Joint. Disconnect will break off the Elements from the Joint and will add duplicate Joints to each of those Elements. To use Disconnect

1. Select the Joints you want disconnected from the Elements.
2. On the Edit menu click Disconnect.

Hints: This is a very useful command for some special modeling conditions.

This feature adds separate joints to Elements sharing common Joints. After the Joint is disconnected special Constraints may be added to those Joints. e.g. You could use this feature to specify moment release Constraints between two adjacent edges of Shell elements or if you need to add a zero length non-linear link you would need separate Joints at the same location.

Connect

Connect facilitates re-connecting all selected elements which had been disconnected from each other and were connected to their own independent Joints. When Elements are connected back to each other the independent Joints are collapsed to a common joint which is shared by all the combined Elements. To use Connect

1. Select the Elements you want connected to each other.

2 On the Edit menu click Connect.

Show Duplicates

This is a useful command to select duplicate Joints, Frames, Shells, ASOLID and Solids from the entire structure. If the duplicates are unnecessary they may be deleted or merged.

1 Select the Joints and or Elements.
2 On the Edit menu click Show Duplicates.
3 Duplicate Joints and Elements will be re-drawn with a different color.

Joint and Element Labels

SAP2000 automatically labels JOINTS, FRAME and SHELL elements. However, it is possible to initialize a labeling scheme or to change labels after they have been assigned.

What do you want to do?

Initialize a New Labeling Scheme

Re-Label Previously Assigned Labels

Re-Label Previously Assigned Labels

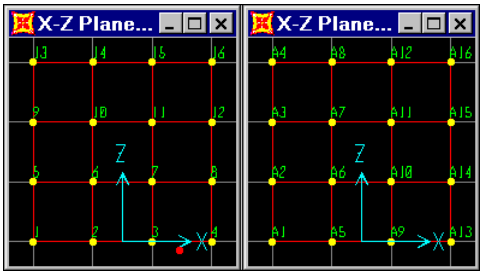
The labels in SAP2000 are alphanumeric. It is possible to select some Joints and Elements and change their labels by assigning them another labeling scheme.

To Change Labels:

1 Select the Joints and Elements (FRAME and SHELL) for which you want to change the labels.
2 On the Edit menu, click Change Labels... This will display the Re-Label Selected Items dialog box.
3 In the Re-Label Selected Items dialog box
- Enter an alphanumeric prefix, the starting numeric number for the scheme to be added to the prefix and a numeric increment for the numeric sequence for Joints, Frames and Shells.
- Click on the Joint, Frame and/or Shell check boxes to further select which type of members within the total selection will be affected by the label change.
- Select the first and second Re-Label order from the drop down boxes. See example below. The labels have been changed to add an alpha prefix and have been sorted first by X and then by Z.

4 Click OK.

EXAMPLE:



VIEW MENU

Select 3-D Views

It is sometimes helpful to view the model in 3-D views. It is possible to chose a quick 3D view by clicking on the  button in the Toolbar.

Setting 3-D View

1. On the View menu, click Set 3D View... This opens the 3D View dialog box.
2. Click the appropriate Fast View button.
3. Use the up/down scroll buttons or enter values in the text edit box for the Plan, Elevation and Aperture.
4. Click OK when finished.

Note: You can view the model orientation in the View window in the top left corner of the 3D View dialog box .

Tips: You can save time by saving 3D views by assigning them names and then recalling them later. Views may be saved through the View menu under Save Named Views.

Also See

 Select 2D Views

Select 2-D Views

It is sometimes easier to build a model while working in 2-D elevation or plan views.

Select an X-Y, X-Z or Y-Z Plane

1. On the View menu, click Set 2D View... This opens the 2D View dialog box.
2. Select the X-Y, X-Z or Y-Z plane.
3. Fill in the edit box with the appropriate Z, Y or X coordinate or click on a relevant plane within the window showing an outline of the model.
4. Click OK.

Note: It is also possible to choose a plane by clicking on one of the 2D view buttons  and then

using the up or down arrow buttons  to move to an appropriate plane.

You can use rubber band zoom to zoom into the window showing the outline of the model. Clicking once anywhere in the window will restore the original view.

Tips: You can save time by saving 2D views by assigning them names and then recalling them later. Views may be saved through the View menu under Save Named Views.

Also See

 Select 3D Views

Set Elements

Setting Elements allows you to selectively display various options associated with the elements. Use this method to selectively display various element types, associated element ID numbers and element section property types, labels and extrusions. You can also suppress element, element fills or shrink the elements about their corresponding centroids to enhance the clarity of the display.

1. On the View menu, click Set Elements or press the  button on the toolbar**.** This will bring up the Set Elements dialog box.
2. In the Set Elements dialog box:
   - Check the options to be activated using the check boxes. Check the Hide under each element's category if you don't want any of the elements and its options visible.
   - Check Shrink Elements if you want the elements shrunk in size to improve viewing of the model or to check element connectivity. You can also do this from outside the Set Elements dialog box by pressing the  button on the toolbar.
   - Check Show Extrusions if you want to see the frame element section shape on the screen. This can help in seeing if the member is oriented properly.
   - Check Fill Elements to see the elements shaded in as opposed to transparent.
   - Check show edges to see the outline of the elements.
   - Click the OK button to update the active screen.

Note: You Set Elements for individual windows not for the full model. So you can use this method to view different information in each window.

Set Limits

Use Set Limits menu when you do not want the whole structure to appear in the display. This option allows you to selectively display portions of the structure by defining limits on the X, Y and Z axes viewing range.

1. On the View menu, click Set Limits**.** This will bring up the Set Limits dialog box.
2. In the Set Limits dialog box:

   Method 1
   - Click on each of the XY, YZ and XZ planes.
   - Using the mouse, draw a box around the area you want to view in the viewing screen that appears in the top left corner of the dialog box.
   - Note that the limits of the box you define will be reflected in the Max and Min range for the corresponding Axis Limits at the bottom of the screen.

   Method 2
   - Alternately directly enter in the Max and Min Axis Limits for the X, Y and Z axes.
   - Click the Show All button to reset the full view for the corresponding axis.
   - Checking the Ignore Limit Settings allows you to view the full structure without resetting the limits.
   - Click OK.

Note: You can also use from the View menu the Show Selection Only option to view only the members that

you have selected. Use the Show All option to go back to viewing all the elements in the model.

DEFINE MENU

Define Materials

1 On the Define menu, click Materials… This will display the Materials dialog box with default materials, CONC, OTHER and STEEL listed in the Materials list box.

What do you want to do?

Add a New Steel Material Type

Add a New Concrete Material Type

Add a New Other Material Type

Add a New Steel Material Type

To Add a New Steel Material:

1 In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.

2 In the Material Property Data box

- Choose Steel from the Design Type drop down list.
- Enter the Material name in the Material Name edit box or accept the default name.
- In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume
- Select whether the material definition is for an Isotropic material or Orthotropic material and enter the:
  - Modulus of Elasticity, Poisson's Ratio and Coefficient of thermal expansion. If the material is defined to be Orthotropic then provide these values for the three local axis of the material. The material's local axis 1 and the element's local axis 1 may differ by the Material angle parameter defined in the Shell Section dialog box.

Note: Frame elements use isotropic material properties, so only the Modulus of Elasticity in Dir 1 and the Poisson's ratio in Plane 12 are used from the Orthotropic material definition.

- In the Design Property area, enter the value for Steel yield stress, fy or accept the default value.
- Click OK.

4 The new Material Name will be added to the Materials list box.

5 To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.

6 Click OK.

Add a New Concrete Material Type

To Add a New Concrete Material:

1. In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.
2. In the Material Property Data box
   - Choose Concrete from the Design Type drop down list.
   - Enter the Material name in the Material Name edit box or accept the default name.
   - In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume
   - Select whether the material definition is for an Isotropic material or Orthotropic material and enter the:
     - Modulus of Elasticity, Poisson's Ratio and Coefficient of thermal expansion. If the material is defined to be Orthotropic then provide these values for the three local axis of the material. The material's local axis 1 and the element's local axis 1 may differ by the Material angle parameter defined in the Shell Section dialog box.

   Note: Frame elements use isotropic material properties, so only the Modulus of Elasticity in Dir 1 and the Poisson's ratio in Plane 12 are used from the Orthotropic material definition.
   - In the Design Property area, enter the value for Reinforcing yield stress (fy), Concrete strength (fc), Shear steel yield stress (fys) and Concrete shear strength (fcs) or accept the default values.
   - Click OK.
3. The new Material Name will be added to the Materials list box.
4. To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.
5. Click OK.

Add a New Other Material Type

To Add a New Other Material:

1. In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.
2. In the Material Property Data box
   - Choose Other from the Design Type drop down list.
   - Enter the Material name in the Material Name edit box or accept the default name.
   - In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume
   - Select whether the material definition is for an Isotropic material or Orthotropic material and enter the:
     - Modulus of Elasticity, Poisson's Ratio and Coefficient of thermal expansion. If the material is defined to be Orthotropic then provide these values for the three local axes of the material. The material's local axis 1 and the element's local axis 1 may differ by the Material angle parameter defined in the Shell Section dialog box.

   Note: Frame elements use isotropic material properties, so only the Modulus of Elasticity in Dir 1 and the Poisson's ratio in Plane 12 are used from the Orthotropic material definition.
   - Click OK.
3. The new Material Name will be added to the Materials list box.
4. To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.
5. Click OK.

Define Section Properties

Section properties may be defined at any time before they are assigned to elements.

What do you want to do?

 Import a Frame Section from the Section property file

 Add a Frame Section by defining the dimensions and or properties manually

 Add a Nonprismatic Frame Section

 Add an Auto Select Frame Group

 Add a Shell Section by defining the dimensions and/or properties

 Add an NLLink Property

Define Frame Section Properties

Section properties may be defined at any time before they are assigned to elements. The define frame section dialog box is composed of a list of defined sections, a drop down list of sections that can be imported and a drop down list of sections that can be added by defining their dimensions. There are also two buttons, one to Modify/Show Sections and one to Delete sections.

What do you want to do?

Import a Frame Section from the Section property file

Add a Frame Section by defining the dimensions and or properties manually

Add a Nonprismatic Frame Section

Add an Auto Select Frame Group

Import a Frame Section from the Section property file (e.g. Sections.pro)

1. On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
2. In the Frame Sections dialog box click on the Import drop down button and choose importing I/Wide Flange or Channel, Tee, Angle etc. This will display the Section Property File selection box.
3. Choose the file name from the Section Property File selection box.
4. Click Open. This will display a multiple selection list box.
5. Select one or more sections from the list box:
   - Clicking on a section name will select only that section.
   - Clicking on a section name and dragging the mouse to other sections will select a range of sections.
   - Pressing the Ctrl key and clicking on different sections will select all of those sections.
6. Click OK. This will display a Section Property form that shows the physical shape and dimensions of the section.
7. The list of selected sections will appear in the Section Name drop down list box.
8. You can continue to load other sections by pressing the Import button which will display the multiple selection list box.
9. It is possible to look at the section properties by clicking on Section Properties.
10. You can modify the Analysis Section properties by pressing the Modification Factors button. There you can change the default factor of 1 that the Analysis Section properties are multiplied by.
11. Click OK to accept all the selected sections.
12. The new section names will be added to the Name list box.
13. To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.
14. Click OK.

Add a Frame Section by defining the dimensions and or properties manually

1. On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
2. In the Frame Sections dialog box click on the Add drop down button and choose Add I/Wide Flange or Channel, Tee, Angle etc. by double clicking on one of the section types. This will display a Section Property form that shows the physical shape and dimensions of the section.
3. Enter the section name or accept the default name.
4. Enter the physical dimensions of the section or accept the default values.
5. Choose the material type (e.g. Steel, Concrete or Other) from the Material drop down list box. For Concrete sections see below.
6. It is possible to look at the section properties by clicking on Section Properties.
7. You can modify the section properties by pressing the Modification Factors button. There you can change the default factor of 1 that the section properties are multiplied by.

8 Click OK to accept the selected section.

9 The new section name will be added to the Name list box.

10 To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.

11 Click OK.

Concrete Section

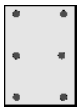
For concrete sections you will need to also perform the following steps:

1 When you select a Concrete material type, an additional Reinforcement button will appear at the bottom of the form. Click the Reinforcement button.

- In the Reinforcement Data dialog box, select Element Class e.g. Column or Beam.

For Columns

- Select the Configuration of Reinforcement e.g. Rectangular or Circular.
- Enter the Concrete Cover in the text edit box.
- Enter the Number of Bars in the 3-Direction in the text edit box.
- Enter the Number of Bars in the 2-Direction in the text edit box.



Example of Concrete Column
3 bars in the 3-Direction
2 bars in the 2-Direction

- If Circular Reinforcement Configuration is selected then enter total number of bars.
- Select and Enter the Area of 1 Bar or
- Select Design Area of Steel for SAP2000 to automatically find the area of steel required during design mode.

For Beams

- Enter the Top and Bottom Concrete Cover in the text edit boxes.
- If you want to specify top and bottom steel, enter reinforcement area for the section in the appropriate text edit box. Otherwise leave values of zero for SAP2000 to calculate the reinforcement automatically.

- Click OK to return to the Section Definition form.

Add a Nonprismatic Frame Section

1 On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.

2 In the Frame Sections dialog box click on the Add drop down button and choose Add Nonprismatic. This will display the Nonprismatic Section Definitions form.

Note: There must be at least two frame sections already defined for the Add Nonprismatic option to show in the add dropdown list box.

3 Enter a name for the Nonprismatic section if the default name is not acceptable.

4 Select from the drop down list box of already defined frame sections, the Start Section and End Section for the Nonprismatic member. These can both be the same section.

5 Enter the length of the section and Absolute Length Type or enter the Length ratio and Variable Length Type.

6 Select the EI variation in the 2-2 and 3-3 directions.

7 Press the Add button to add the section to the list.

8 Use the Insert, Modify and Delete buttons to edit the Nonprismatic section.

9 Press the OK button when finished.

Note: You can add more than one pair of frame sections to build the Nonprismatic section. By entering Absolute lengths (defined in the current units) you can define sections that are of fixed length. Then the sections defined as Variable Length Type will be extended between the fixed sections. The Variable Length value is the ratio of the length of the section relative to all the variable length sections defining the

Nonprismatic member. E.g. if there are two Variable Length sections with lengths of 1 and 2, the section with length 1 will be 1/3 of the total variable length, and the section with a length of 2 will be 2/3 of the total variable length.

This method of defining Nonprismatic sections can be used to define sections that have top and bottom fixed length Nonprismatic sections with a prismatic section defined between them, like a bridge pier, where the prismatic part of the section can "telescope" to fit the space between the fixed ends.

Add an Auto Select Frame Group

Auto Select section is a list of frame sections that are used for doing steel optimization design. SAP2000 will use the average properties of all the sections in the group for the first analysis run. Then selecting Start Design/Check of Structure will design each frame member assigned to an Auto Select group with the lightest section in the Auto Select group that satisfies all the stress checks. Then selecting Update Analysis Sections will make the selected section from the Auto Select group the new analysis section.

1. On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
2. In the Frame Sections dialog box click on the Add drop down button and choose Add Auto Select. This will display the Auto Selection Sections form.
3. Enter a name for the Auto Select section if the default name is not acceptable.
4. Select and Add sections from the List of Sections to the Auto Selections List.
5. Click OK.

Add a Shell Section by defining the dimensions and / or properties

1. On the Define menu, click Shell Sections... This will display the Shell Sections dialog box.
2. In the Shell Sections dialog box click on the Add New Section button. This will display Shell Sections Data dialog box.
3. In the Shell Sections Data dialog box
   - Enter the Section Name or accept the default name.
   - Select the material type by clicking on the Material drop down box and change the Material angle if desired. This angle is used for orthotropic materials to define the material's local 1 axis with respect to the section's local 1 axis.
   - Enter the Membrane and Bending Thickness.
   - Choose the shell type to be Shell, Membrane or Plate.
   - Uncheck the Thick Plate option if you do not want to consider thick plate behavior for the section.
   - Click OK.
4. The new section name will be added to the Shell Sections list box.
5. To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.
6. Click OK.

Note: Shell elements defined as Thick Plate include transverse shear deformation using the Mindlin/Reissner theory and only affect the plate behavior, not the membrane behavior of the element. The thin plate elements use the Kirchhoff theory.

Add an NLLink Property

1. On the Define menu, click NLLink Properties... This will display the NLLink Properties dialog box.
2. In the NLLink Properties dialog box click on the Add New Property button. This will display NLLink Property Data dialog box.
3. In the NLLink Property Data dialog box
   - Enter the Property Name or accept the default name.
   - Select the Property type to be Damper , Gap , Hook , Plastic1 , Isolator1 or Isolator2 by clicking on the Type drop down box.
   - Enter the Mass, Weight and Rotational Inertias in the edit boxes.
   - Choose the appropriate directions (U1,U2,U3,R1,R2,R3) for this property by clicking the check boxes and

also choose whether it is nonlinear or not.

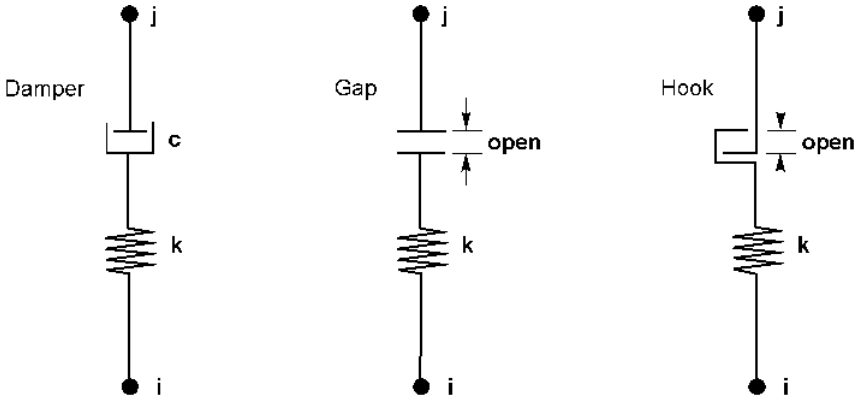
- Directional properties such as Stiffness, Damping, Yield Strength, Post Yield Stiffness Ratio and exponents etc. may be specified by clicking on the Modify/Show Properties button.
- Click OK.

4 The new section name will be added to the NLLink Props list box.

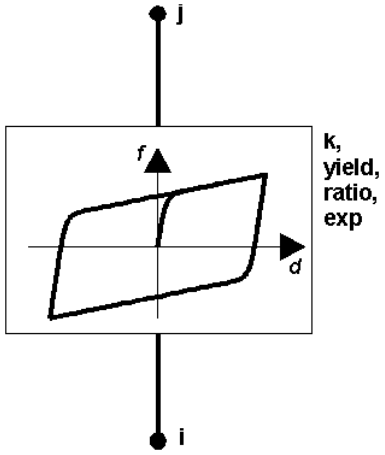
5 To Delete or Modify/Show Property, click on the Property name in the list box to select it and then click on the Modify/Show Property or Delete Property button.

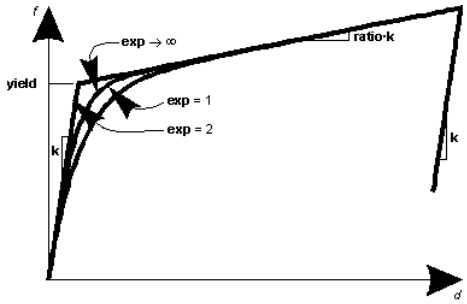
6 Click OK.

NL Damper, Gap and Hook Properties

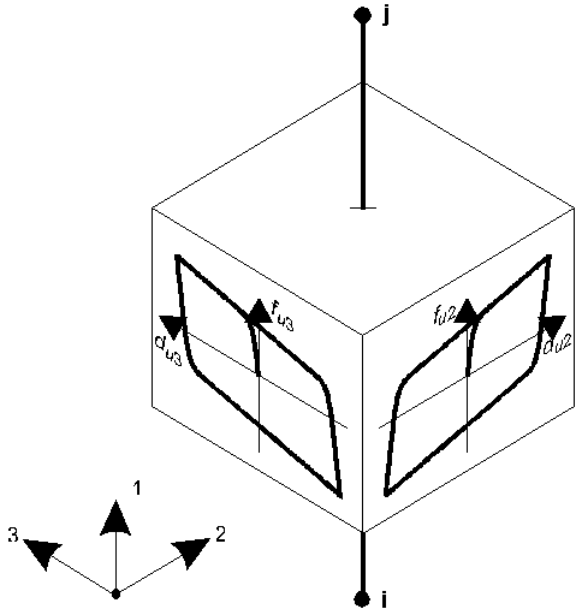


NL Plastic 1 Properties

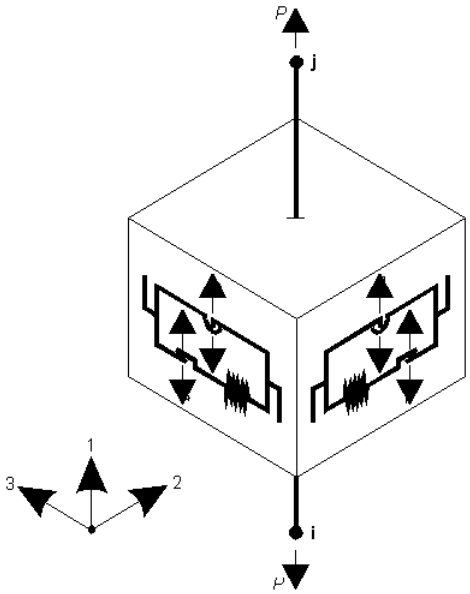




NL Isolator 1 Properties



NL Isolator 2 Properties



Define Static Pushover Cases

1. On the Define menu select Static Pushover Cases. This displays the Define Static Pushover Cases dialog box.
2. To add a new pushover case, click the Add New Pushover button and fill in the resulting Static Pushover Case Data dialog box.
3. To modify or show (i.e. view) an existing pushover case, highlight the pushover case name in the Pushover Name list box, click the Modify/Show Pushover button and modify or view the resulting Static Pushover Case Data dialog box.
4. To delete a pushover case, highlight the pushover case name in the Pushover Name list box and then click the Delete Pushover button.

See Also

- Nonlinear Static Pushover Analysis
- Define Frame Hinge Properties
- Frame Hinge Property Data
- Frame Hinge Property Data For XXX
- Frame Hinge Interaction Surface
- Interaction Surface Definition
- Static Pushover Case Data
- Assign Frame Hinges (Pushover)
- Pushover Curve
- Override Axis Labels And Range
- Override Structural Behavior Type

Static Pushover Case Data

1. On the Define menu select Static Pushover Cases. This displays the Define Static Pushover Cases dialog box. Click the Add New Pushover button, or select a pushover case and click the Modify/Show Pushover button to

display the Static Pushover Case Data dialog box.

2. In the Options area, select either the Push To Load Level Defined By Pattern option or the Push To Displacement Of option.
3. If the Push To Displacement Of option is selected, review the default displacement, Control Joint and Control Direction, changing them if necessary.
4. To start the pushover from the end condition of a previously specified pushover, select the name of the previous pushover from the Start From Previous Pushover drop down list. Typically this option is used for a lateral pushover case to specify that it should start from the end of a gravity load pushover case.
5. To include P-Delta affects in the pushover analysis check the Include P-Delta check box. The P-Delta affects are analyzed only in frame elements though other elements may be present in the model.
6. Check the Restart When Unloading check box if you want the analysis performed using this option
7. If you want only positive displacement increments of the pushover curve to be saved then check the Save Positive Increments Only check box
8. Review the Minimum Saved Steps , Maximum Saved Steps , Maximum Failed Steps , Event Force Tolerance , and Event Deformation Tolerance analysis control parameters, changing the default values if necessary. In most cases, the default values are adequate and it is not necessary to change them.
9. Define a Load Pattern for the pushover case:

- To add a Load to the Load Pattern definition, select the load from the Load drop down box, type in the appropriate scale factor in Scale Factor edit box, and click the Add button. If the selected load is a mode (only available if mode shapes have been requested for the analysis) then input the mode shape in the resulting dialog box and click OK.
- To modify the scale factor for a load specified as part of the load pattern, highlight the load in the Load/Scale Factor list box, edit the scale factor in the Scale Factor edit box, and click the Modify button.
- To delete a load specified as part of the load pattern, highlight the load in the Load/Scale Factor list box and click the Delete button.

10. Click the OK button to accept all of the changes made in the Static Pushover Case Data dialog box. Clicking the Cancel button means that none of the changes will be accepted. Note that you must also click OK in the Define Static Pushover Cases dialog box in order to accept the changes.

See Also

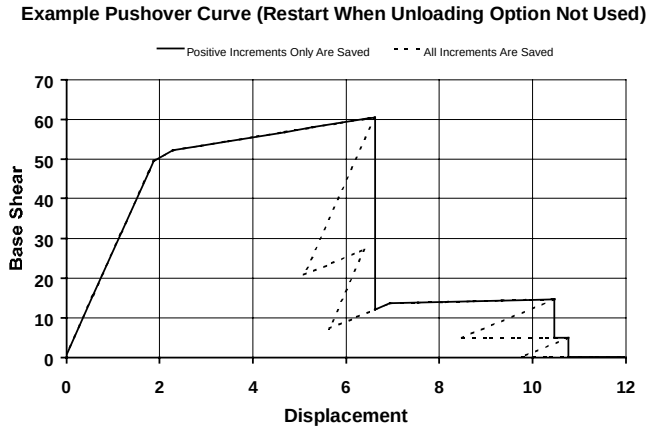
- Nonlinear Static Pushover Analysis
- Define Frame Hinge Properties
- Frame Hinge Property Data
- Frame Hinge Property Data For XXX
- Frame Hinge Interaction Surface
- Interaction Surface Definition
- Define Static Pushover Cases
- Assign Frame Hinges (Pushover)
- Pushover Curve
- Override Axis Labels And Range
- Override Structural Behavior Type

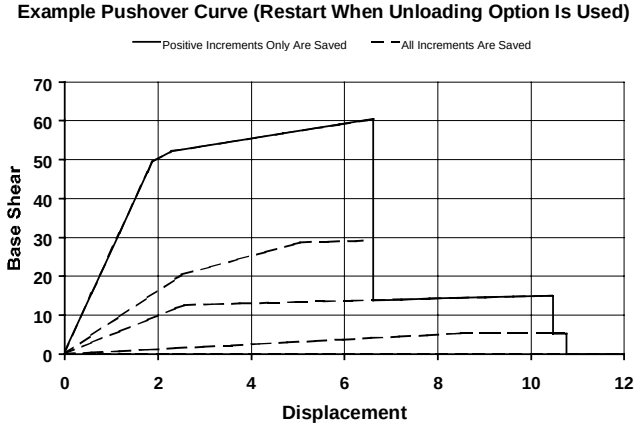
Push To Load Level Defined By Pattern

The Push To Load Level Defined By Pattern option button is used to perform a force-controlled analysis. The pushover typically proceeds to the full load value defined by the sum of all loads included in the Load Pattern box (unless it fails to converge at a lower force value). This option is useful for applying gravity load to the structure.

Save Positive Increments Only

Two simple example pushover curves are presented to illustrate the effect of the Save Positive Increments Only option. In the examples, the solid line represents the pushover curve if the Save Positive Increments Only check box is checked (the default) and the dashed line represents the pushover curve if the Save Positive Increments Only check box is *not* checked. The first figure shows the pushover curve when the Restart When Unloading option is *not* used, and the second shows the pushover curve when the Restart When Unloading option is used.

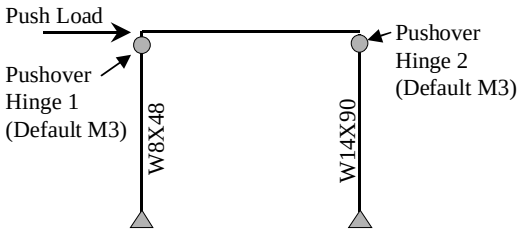




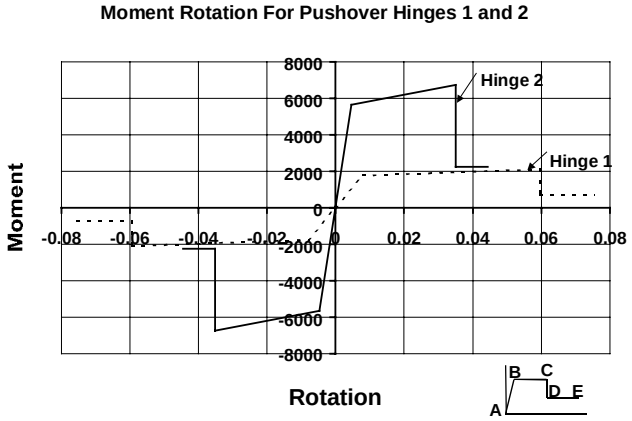
Restart When Unloading

The Restart When Unloading option uses a slightly different solution technique to obtain the pushover curve. Whenever a hinge reaches point C or E on its force displacement curve it must begin to drop load. When the Restart When Unloading option is used the program will start the pushover process over again each time a hinge unloads. The process is demonstrated by a simple example.

Consider the frame with two pushover hinges shown below.

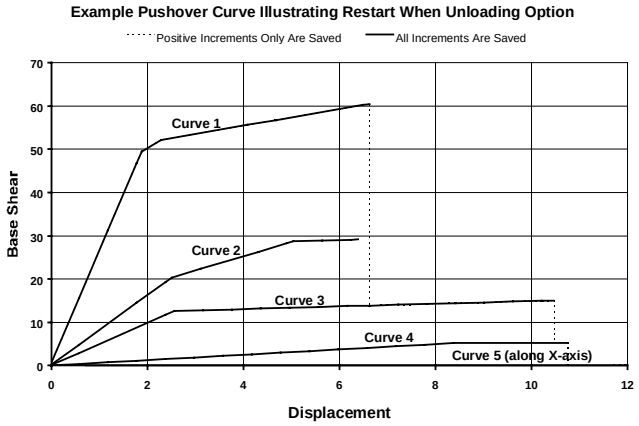


The force deflection characteristics of the two hinges are illustrated below.



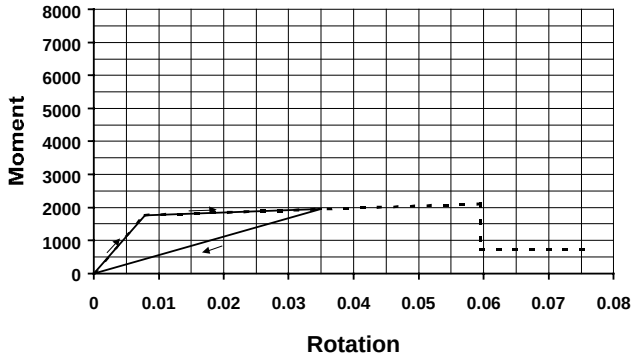
When the pushover analysis is run it turns out that pushover hinge 2 passes point E (i.e., unloads twice) before pushover hinge 1 reaches point C and begins to do any unloading.

The pushover curve using the Restart When Unloading option is shown in the figure below. The pushover curve is made up of five separate curves. Curve 1 ends when pushover hinge 2 reaches point C and the structure must drop load. Curve 2 ends when pushover hinge 2 reaches point E and the structure must drop additional load. Curve 3 ends when pushover hinge 1 reaches point C and Curve 4 ends when pushover hinge 2 reaches point E. Curve 5 ends when the displacement target is reached. Note that since a mechanism has occurred, curve 5 is just a flat line along the X-axis.

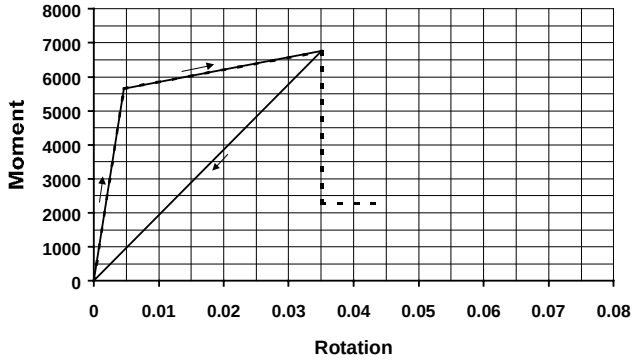


The following ten figures show the loading and unloading paths taken by pushover hinge 1 and pushover hinge 2 for each of the five components of the pushover curve.

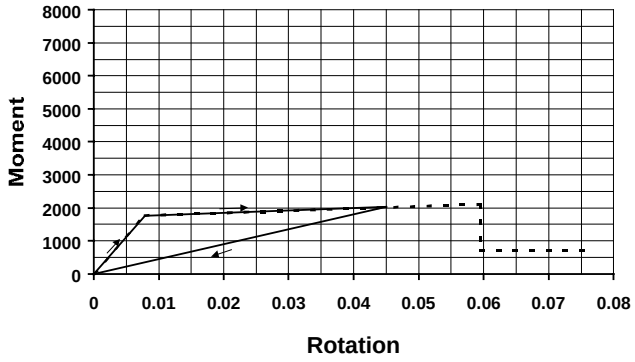
Moment Rotation For Hinge 1 Used To Create Curve 1



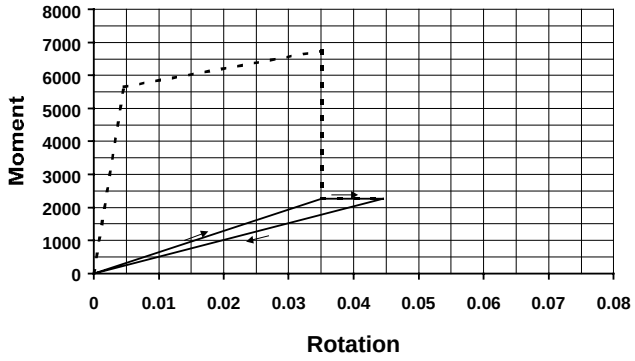
Moment Rotation For Hinge 2 Used To Create Curve 1



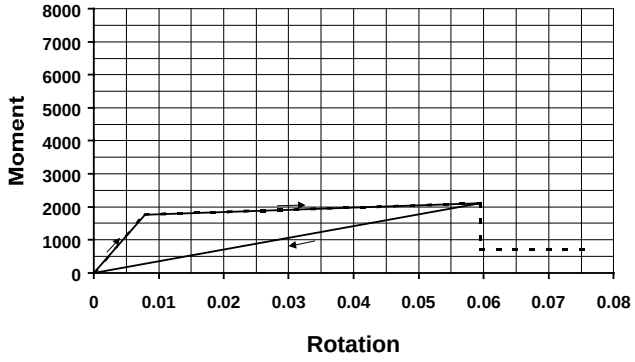
Moment Rotation For Hinge 1 Used To Create Curve 2



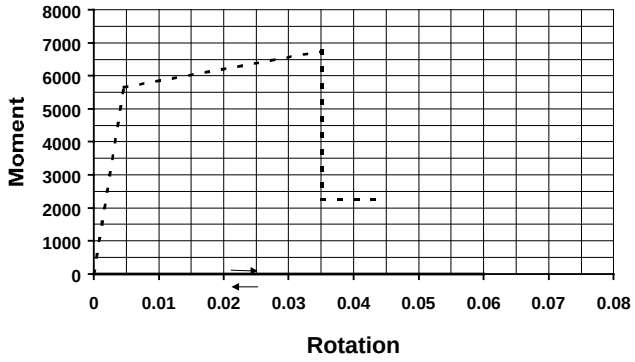
Moment Rotation For Hinge 2 Used To Create Curve 2



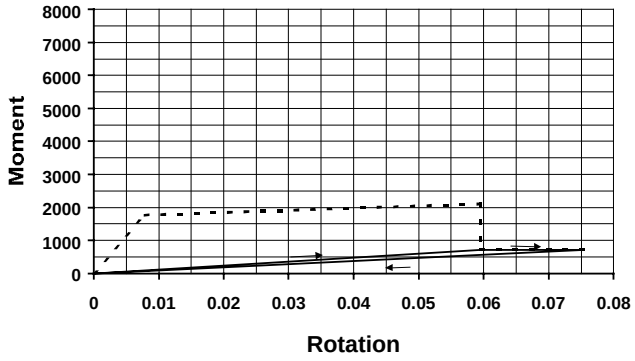
Moment Rotation For Hinge 1 Used To Create Curve 3



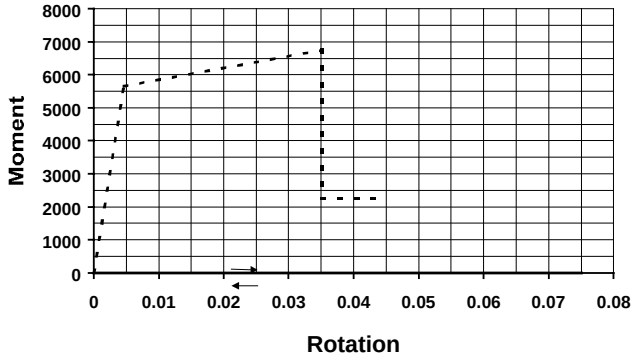
Moment Rotation For Hinge 2 Used To Create Curve 3



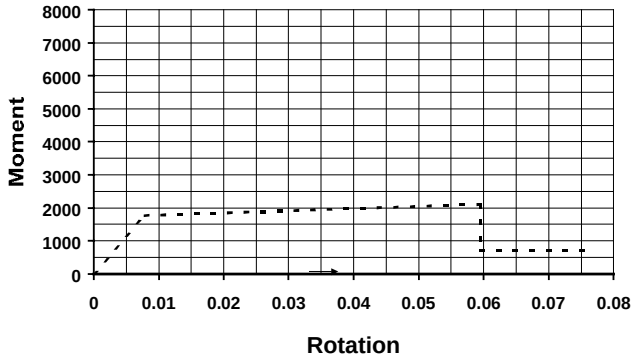
Moment Rotation For Hinge 1 Used To Create Curve 4

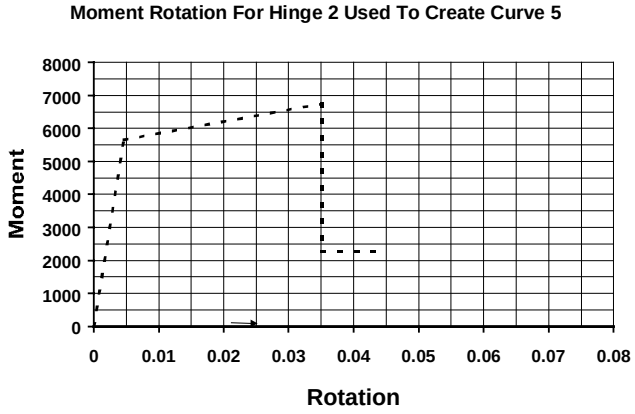


Moment Rotation For Hinge 2 Used To Create Curve 4



Moment Rotation For Hinge 1 Used To Create Curve 5





Push To Displacement Of

The Push To Displacement Of option button is used to perform a displacement-controlled analysis. The pushover typically proceeds to the specified displacement in the specified control direction at the specified control joint (unless it fails to converge at a lower displacement value). The specified displacement, specified control direction and specified control joint are all given default values by SAP2000; you can easily replace these default values. The default value for the specified displacement is 0.04 times the Z coordinate at the top of the structure. Note that if the base of the structure has a Z coordinate greater than zero, the default displacement may be quite large. The default value for the control joint is a joint located at the top of the structure. The default control direction is U1; other available directions are U2, U3, R1, R2, and R3. Note that the control directions are in local joint coordinates. If you have not rotated the local joint coordinate system, then the local joint coordinate system U1, U2, U3, R1, R2 and R3 corresponds directly to the global joint coordinate system UX, UY, UZ, RX, RY and RZ respectively (i.e., U1=UX, U2=UY, U3=UZ, R1=RX, R2=RY and R3=RZ). The Push To Displacement Of option is useful for applying lateral load to the structure.

Minimum and Maximum Saved Steps

The Minimum Saved Steps and Maximum Saved Steps provide control over the number of points actually saved in the pushover analysis. The default values are adequate in most cases. If the minimum number of steps saved is too small, you may not have enough points to adequately represent the pushover curve. If the minimum and maximum number of saved steps is too large, then the analysis may consume a considerable amount of disk space, and it may take an excessive amount of time to display results.

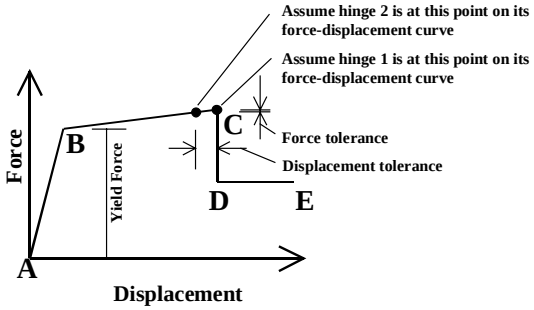
The program automatically determines the spacing of pushover steps to be saved as follows. The maximum step length is equal to total force goal or total displacement goal divided by the specified Minimum Saved Steps. The program starts by saving steps at this increment. If a significant event occurs at a step length less than this increment, then the program will save the step too and pick up with the maximum increment from there. For example, suppose the Minimum Saved Steps and Maximum Saved Steps are set at 20 and 30 respectively, and the pushover is to be to a displacement of 10 inches. The maximum increment of saved steps will be 10 / 20 = 0.5 inches. Thus, data is saved at 0.5, 1, 1.5, 2, 2.5 inches. Suppose that a significant event occurs at 2.7 inches. Then data is also saved at 2.7 inches, and continues on from there being saved at 3.2, 3.7, 4.2, 4.7, 5.2, 5.7, 6.2, 6.7, 7.2, 7.7, 8.2, 8.7, 9.2, 9.7 and 10.0 inches.

The Maximum Saved Steps controls the number of significant events for which data will be saved. The program will always reach the force or displacement goal within the specified number of maximum saved steps, however, in doing so it could have to skip saving steps at later events. For example, suppose the Minimum Saved Steps is set to 20, the Maximum Saved Steps is set to 21, and the pushover is to be to a displacement of 10 inches. The maximum increment of saved steps will be 10 / 20 = 0.5 inches. Thus, data is saved at 0.5, 1, 1.5, 2, 2.5 inches. Suppose that a significant event occurs at 2.7 inches. Then data is also saved at 2.7 inches, and continues on from there being saved at 3.2 and 3.7 inches. Suppose another significant event occurs at 3.9 inches. The program will not save the data at 3.9 inches because if it did it would not be able to limit the maximum increment to 0.5 inches and still get through the full pushover in no more than 21 steps. Note that if a second significant event occurred at 4.1 inches rather than 3.9 inches, then the program would be able to save the step and still meet the

specified criteria for maximum increment and maximum number of steps.

Event Force And Deformation Tolerances

The Event Force Tolerance and the Event Deformation Tolerance are ratios which are used to determine when an event actaully occurs for a hinge. Consider the figure which shows the location of two hinges on their force-displacement plots. Hinge 1 has reached an event location. For hinge 2, if both the Event Force Tolerance and the Event Displacement Tolerance are met, then the hinge is within event tolerance and it too will be treated as part of the event. In the figure, if the Force Tolerance divided by the Yield Force is less than the Event Force Tolerance specified in the Static Pushover Case Data, and the Displacement Tolerance divided by the horizontal distance from B to C is less than the Displacement Event Tolerance specified in the Static Pushover Case Data, then hinge 2 will be treated as part of the event. When determining the Force Tolerance Ratio, the denominator is always the yield force. When determining the Displacement Tolerance Ratio, the denominator is the horizontal length of the portion of the force-displacement curve that the hinge is currently on. In the figure, hinge 2 is on the B-C portion of the curve, thus we used the B-C horizontal length in the denominator of the Displacement Tolerance Ratio.



Maximum Failed Steps

The Maximum Failed Steps is used, if necessary, to declare failure (i.e., non-convergence) in a run before it reaches the specified force or displacement goal. The program may be unable to converge on a step when catastrophic failure occurs in the structure. There may also be instances where it is unable to converge on a step due to numerical sensitivity in the solution. The Maximum Failed Steps is a cumulative counter through the entire analysis. If the Maximum Failed Steps is reached, the analysis stops.

Load Pattern

Load Patterns are used to describe the distribution of force on the structure. There are four different methods of describing the distribution of load on the structure for a pushover load case:

1. If acc dir X, acc dir Y or acc dir Z is selected, then a uniform acceleration is applied in the appropriate direction, i.e., a lateral force is applied at each node that is proportional to the mass tributary to that node.
2. If mode shapes have been requested for the analysis, then a MODE load pattern is available. This load pattern applies a lateral force that is proportional to the product of a specified mode shape times the mass tributary to that node. You may specify the mode shape to be used when the Add button or the Modify button is clicked.
3. You may select a previously defined static load pattern.
4. You may combine any of the methods described in 1, 2 and 3 above.

Define Frame Hinge Properties

1. On the Define menu select Hinge Properties…. This displays the Define Frame Hinge Properties dialog box. A list of hinge properties is displayed in the Hinge Props dialog box. By default the list of properties in the Hinge Props list box includes the built-in default hinge properties and the user-defined hinge properties . It does not include the currently generated hinge properties that the program generates automatically at each hinge location. To view these generated properties check the Show Generated Props box, which will make these properties visible in the Hinge Props list box. You can then highlight one of the generated hinge properties, click the Modify/Show Properties button and view the properties for the hinge.
2. To define a new hinge property, click the Define New Property button and fill in the resulting Frame Hinge Property Data dialog box. Note that if you highlight an existing hinge property in the Hinge Props list box and then click the Define New Property button, the initial properties for the new hinge will be the same as those for the highlighted hinge. You can use this feature to easily "copy" hinge properties and then modify them as needed.
3. To modify or show (i.e. view) an existing frame hinge property, highlight the hinge property name in the Hinge Props list box, click the Modify/Show Property button and modify or view the resulting Frame Hinge Property Data dialog box. Note that you can view generated hinge properties, but you can not modify them.
4. To delete a hinge property, highlight the pushover case name in the Hinge Props list box and then click the Delete Property button. Note that if the hinge property is still being used anywhere in the model, the Delete Property button will be grayed out and inactive. A hinge property can not be deleted until it has been removed from all frame elements.

See Also

- Nonlinear Static Pushover Analysis
- Frame Hinge Property Data
- Frame Hinge Property Data For XXX
- Frame Hinge Interaction Surface
- Interaction Surface Definition
- Define Static Pushover Cases
- Static Pushover Case Data
- Assign Frame Hinges (Pushover)
- Pushover Curve
- Override Axis Labels And Range
- Override Structural Behavior Type

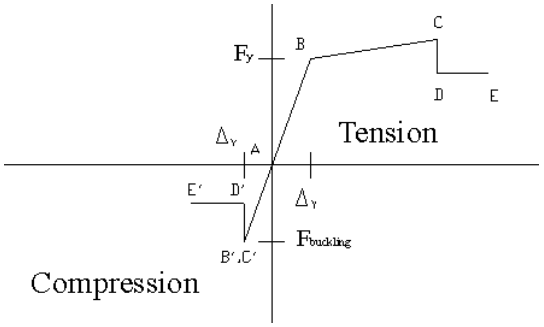
Default Hinge Properties

The built-in default hinge properties are typically based on FEMA-273 and/or ATC-40 criteria.

The default hinge properties currently available are:

- Steel Axial Hinge
- Steel Moment Hinge and Steel P-M-M Hinge
- Steel Shear Hinge
- Concrete Axial Hinge
- Concrete Moment Hinge and Steel P-M-M Hinge
- Concrete Shear Hinge

Steel Axial Hinge



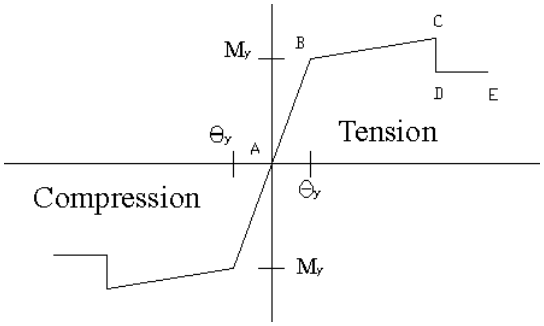
Hinge Features:

- Slope between points B and C is taken as 3 % strain hardening
- Hinge length assumption for ∆y is the length of the member
- Initial compression slope is taken to be same as the initial tension slope
- Points C, D and E based on FEMA 273 Table 5.8, Braces in Tension
- Points C', D' and E' based on FEMA 273 Table 5.8, Braces in Compression, Item C

See Also

Default Hinge Properties

Steel Moment Hinge and Steel P-M-M Hinge



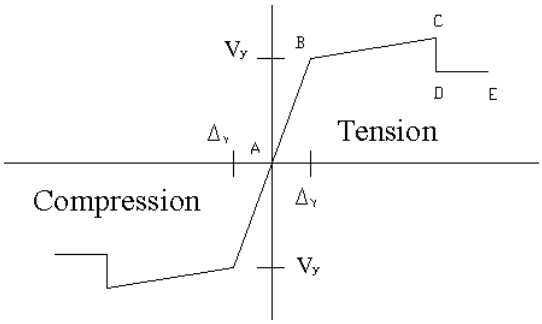
Hinge Features:

- Slope between points B and C is taken as 3 % strain hardening
- Θy based on FEMA 273, equation 5-1 and 5-2
- Points C, D and E based on FEMA 273 Table 5.4, for b / 2tf < 52 / (Fyc) ^ 1/2
- P-M-M curve taken to be same as the Moment curve in conjunction with the definition of Axial–Moment interaction curves.

See Also

Default Hinge Properties

Steel Shear Hinge



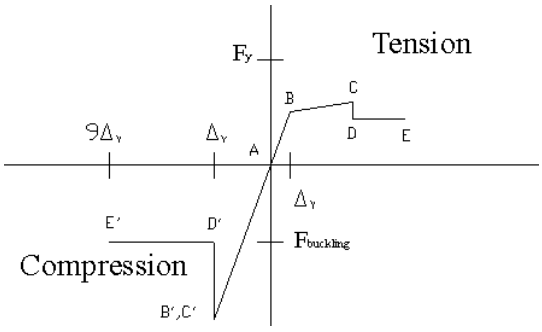
Hinge Features:

- Slope between points B and C is taken as 3 % strain hardening
- Δy = 0.01 radians per Note 3 in FEMA 273 Table 5.8
- Points C, D and E based on FEMA 273 Table 5.8, Link Beam, Item a

See Also

Default Hinge Properties

Concrete Axial Hinge



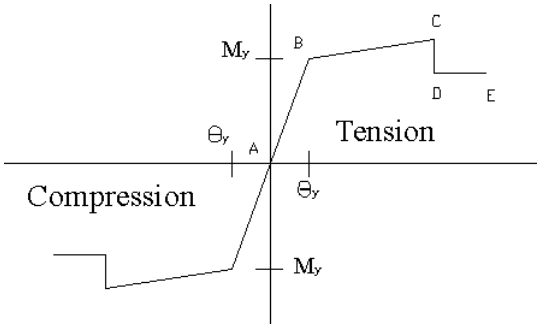
Hinge Features:

- Py = As fy
- Pc = 0.85 Ac f'c
- Slope between points B and C is taken as 10 % total strain hardening for steel
- Hinge length assumption for Δy is based on the full length
- Point B, C, D and E based on FEMA 273 Table 5.8, Braces in Tension
- Point B' = Pc
- Point E' taken as 9Δy

See Also

Default Hinge Properties

Concrete Moment Hinge and Concrete P-M-M Hinge



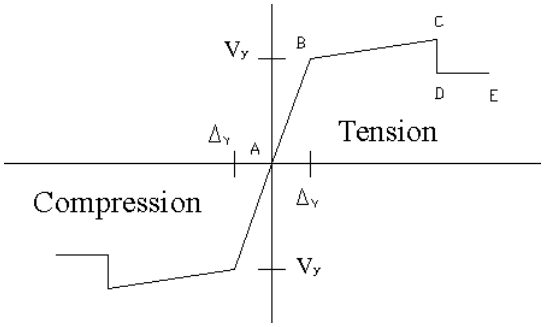
Hinge Features:

- Slope between points B and C is taken as 10 % total strain hardening for steel
- Θy = 0, since it is not needed
- Points C, D and E based on ATC-40, Table 9.6 . The four conforming transverse reinforcing rows are averaged
- My based on reinforcement provided, otherwise based on minimum allowable reinforcement
- P-M-M curve taken to be same as the Moment curve in conjunction with the definition of Axial–Moment interaction curves.

See Also

Default Hinge Properties

Concrete Shear Hinge



Hinge Features:

- Slope between points B and C is taken as 10 % total strain hardening for steel
- Vy = 2 As (f'c) ^ ½ + fy Asv d
- Points C, D and E based on ATC-40 Table 9.12, Item 2, average of the two rows labeled “Conventional longitudinal reinforcement” and “Conforming transverse reinforcement”

See Also

Default Hinge Properties

Types Of Hinge Properties (Pushover)

There are three types of hinge properties in SAP2000. They are default hinge properties, user-defined hinge properties and generated hinge properties. Only default hinge properties and user-defined hinge properties can be assigned to frame elements. When these hinge properties (default and user-defined) are assigned to a frame element, the program automatically creates a new generated hinge property for each and every hinge.

The built-in default hinge properties for steel members are generally based on Tables 5.4 and 5.8 in FEMA-273. The built-in default hinge properties for concrete members are generally based on Tables 9.6, 9.7 and 9.12 in ATC-40. **You should review any generated properties for their applicability to your specific project.**

Default hinge properties can not be modified. They also can not be viewed because the default properties are section dependent. The default properties can not be fully defined by the program until the section that they apply to is identified. Thus, to see the effect of the default properties, the default property should be assigned to a frame element, and then the resulting generated hinge property should be viewed.

User-defined hinge properties can be either be based on default properties or they can be fully user-defined. When user-defined properties are based on default properties, the hinge properties can not be viewed because, again, the default properties are section dependent. When user-defined properties are not based on a default properties, then the properties can be viewed and modified.

The generated hinge properties are used in the analysis. They can be viewed, but they can not be modified. Generated hinge properties have an automatic naming convention of LabelH#, where Label is the frame element label, H stands for hinge, and # represents the hinge number. The program starts with hinge number 1 and increments the hinge number by one for each consecutive hinge applied to the frame element. For example if a frame element label is F23, the generated hinge property name for the second hinge applied to the frame element is F23H2.

The main reason for the differentiation between defined properties (in this context, defined means both default and

user-defined) and generated properties is that typically the hinge properties are section dependent. Thus it is necessary to define a different set of hinge properties for each different frame section type in the model. This could potentially mean that you would need to define a very large number of hinge properties. To simplify this process, the concept of default properties is used in SAP2000. When default properties are used, the program combines its built-in default criteria with the defined section properties for each element to generate the final hinge properties. The net effect of this is that you do significantly less work defining the hinge properties because you don't have to define each and every hinge.

Frame Hinge Property Data

1. On the Define menu select Hinge Properties.... This displays the Define Frame Hinge Properties dialog box. Click the Define New Property button, or select a hinge property from the Hinge Props list box and click the Modify/Show Property button to display the Frame Hinge Property Data dialog box. Note if you select an existing user-defined hinge you can view or modify its properties, but if you select a generated hinge you can only view its properties.
2. To change the name of a new or existing user-defined hinge property, type the new name in the Property Name edit box.
3. To view the properties of an existing user-defined or generated hinge, click the Modify/Show button for an active hinge type. In this context, an active hinge type is one that has the check box to the left of it checked. Note that if the associated Default check box is checked, then the Modify/Show button is not active, and the properties can not be viewed or modified.
4. To define properties for a new hinge, or to add to an existing defined hinge property definition do the following:
   - In the Hinge Type column of the Hinge Properties area, check the check box to the left of the hinge type description to activate that hinge type for the current hinge property. Note that a hinge property can consist of more than one hinge type. Also note that for each hinge type, either the program can generate default properties, or you can define the properties.
   - Check the box in the Default column of the Hinge Properties area to have the program automatically generate the hinge properties for a particular hinge type. The hinge properties generated by the program are typically based on FEMA-273 and/or ATC-40 requirements. Note that if default properties are used for a particular hinge type in a defined hinge property, then the Modify/Show button for that hinge type will be grayed out and inactive. To see the properties used in this case, you must first assign the defined hinge property to a frame element and then view the generated hinge property.
   - To input user-defined hinge properties (i.e. non-default properties), click the Modify/Show button for the appropriate hinge type and fill in the resulting Frame Hinge Property Data For XXX dialog box (XXX represents the hinge property name).

Tip:When the Default hinge property is used for a P-M2-M3 hinge type, the resulting hinge property must be assigned to a frame element with either STEEL or CONC material type. This allows the program to pick up certain design properties (yield stress, etc.) for the element. If the frame element material type is not STEEL or CONC, the program will display a message saying the hinge property assignment can not be made.

Tip:When the Default hinge property is used for a P-M2-M3 hinge type, if the material property for the associated frame element is STEEL, then the "Steel, FEMA 273 Equation 5-4" option is used, the expected yield strength is set to either 1.0FY if FY ≤ 36 ksi or 0.8FY if FY > 36 ksi, and the strain-hardening slope is set to 3% of the elastic slope . If the material property for the associated frame element is CONC, then the "Concrete, ACI 318-95 with phi = 1" option is used and the strain-hardening slope is set to 5% of the elastic slope .

See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Pushover Curve

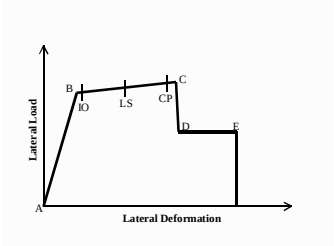
Override Axis Labels And Range
Override Structural Behavior Type

Hinge Types

The Axial P, Shear V2, Shear V3, Torsion T, Moment M2 and Moment M3 frame hinge types are all uncoupled. They can exist in the same frame hinge property, but there is no interaction between them. The Interacting P-M2-M3 frame hinge type is a coupled hinge property.

Elastic and Strain-Hardening Slopes

The elastic slope is the slope of line AB in the element load deformation curve shown in the figure. The strain-hardening slope is the slope of line BC.



Frame Hinge Property Data for XXX

1. On the Define menu select Hinge Properties.... This displays the Define Frame Hinge Properties dialog box. Click the Define New Property button, or select a defined hinge property from the Hinge Props list box and click the Modify/Show Property button to display the Frame Hinge Property Data dialog box. Click the Modify/Show button for a hinge type whose Hinge Type check box is checked to display the Frame Hinge Property Data For XXX dialog box. Note that in the dialog box the XXX will be replaced with the hinge property name.
2. If the Hinge Type is Axial P, Shear V2, or Shear V3, then the force-displacement criteria and the acceptance criteria may be input either as force versus displacement, or as stress versus strain . Thus for Axial P, Shear V2, or Shear V3 hinge types:
   - In the Type area, select either the Force - Displacement option or the Stress - Strain option. Note the Type Area is not visible for moment and torsion hinges. These types of hinges can only be specified using Force – Displacement criteria.
   - If the Stress- Strain option is chosen, then fill in the hinge length in the Hinge Length edit box. The hinge length may be input as either an absolute length or a relative length. Check the Relative Length box to input the hinge length as a relative length. When rigid end offsets are defined, the relative length is based on the clear length of the element between the rigid end offsets.
3. Fill in the force-displacement criteria in the Force-Displacement area. Note that the force and displacement are normalized by the yield force and the yield displacement respectively.
   - For Axial P hinge properties, when the force-displacement input type is chosen, the force is input as P/PY and the displacement is input as $\Delta/\Delta Y$. When the stress-strain input type is chosen, the force (stress) is input as $\sigma/\sigma Y$ and the displacement (strain) is input as $\varepsilon/\varepsilon Y$.
   - For Shear V2 and Shear V3 hinge properties, when the force-displacement input type is chosen, the force is input as V/VY and the displacement is input as $\Delta/\Delta Y$. When the stress-strain input type is chosen the force (stress) is input as $\tau/\tau Y$ and the displacement (strain) is input as $\varepsilon/\varepsilon Y$.
   - For Torsion T hinge properties, the force is input as T/TY and the displacement is input as $\theta/\theta Y$.
   - For Moment M2 and Moment M3 hinge properties, the force is input as M/MY and the displacement is input as $\theta/\theta Y$.
   - For P-M2-M3 hinge properties the force is input as M3/M3Y and the displacement is input as $\theta 3/\theta 3Y$. The program internally creates M2/M2Y versus $\theta 2/\theta 2Y$ and P/PY versus $\Delta/\Delta Y$ plots that are compatible with the

input M3/M3Y versus θ3/θ3Y plot.

4. Fill in the acceptance criteria in the Acceptance Criteria area. The acceptance criteria are displacements that have been normalized by the yield displacement.
   - For Axial P, Shear V2 and Shear V3 hinge properties, when the force-displacement input type is chosen, the acceptance criteria is input as Δ/ΔY. When the stress-strain input type is chosen the acceptance criteria is input as ε/εY.
   - For Torsion T, Moment M2 and Moment M3 hinge properties, the acceptance criteria is input as θ/θY.
   - For P-M2-M3 hinge properties the acceptance criteria is input as θ3/θ3Y. The program internally creates θ2/θ2Y and Δ/ΔY criteria that are compatible with the input θ3/θ3Y criteria.
5. By default the program will automatically calculate the yield force, moment or stress and the yield displacement, rotation or strain for the Axial P, Shear V2, Shear V3, Torsion T, Moment M2 and Moment M3 hinge types. To overwrite these default values uncheck the appropriate Calculate Yield check box in the Scaling area and type in your own value in the appropriate Yield edit box.
6. For P-M2-M3 hinge types, click the Interaction Surface button to define, modify or view the P-M2-M3 interaction surface. This will display the Frame Hinge Interaction Surface dialog box.

See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Pushover Curve
Override Axis Labels And Range
Override Structural Behavior Type

Stress Versus Strain

When the force-displacement criteria and the acceptance criteria are input as stress versus strain, the program internally converts it to force versus displacement. For axial hinges the force is calculated as stress times the cross-sectional area. For shear hinges the force is calculated as stress times the shear area. For both axial and shear hinges, the displacement is calculated as strain times the hinge length.

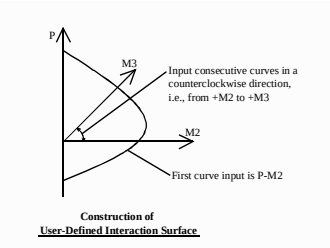
Frame Hinge Interaction Surface

1. On the Define menu select Hinge Properties…. This displays the Define Frame Hinge Properties dialog box. Click the Define New Property button, or select a defined hinge property from the Hinge Props list box and click the Modify/Show Property button to display the Frame Hinge Property Data dialog box. Click the Modify/Show button for a P-M2-M3 hinge type whose Hinge Type check box is checked to display the Frame Hinge Property Data For XXX dialog box. (Note that in the dialog box the XXX will be replaced with the hinge property name). Click the Interaction Surface button to display the Frame Hinge Interaction Surface dialog box.
2. Select one of the following options.
   - Steel, AISC-LRFD Equations H1-1a and H1-1b with phi = 1.
   - Steel, FEMA 273 Equation 5-4.
   - Concrete, ACI 318-95 with phi = 1 .
   - User Definition.

   The default option is User Definition.
3. If one of the steel options is chosen, type in the expected yield strength in the appropriate Expected Yield

Strength edit box.

4. If the User Definition option is chosen then:
   - Checking the Doubly Symmetric About M2 and M3 check box means the interaction surface is input in the (+ M2, +M3) quadrant for all values (i.e. both positive and negative) of P. Thus only 90 degrees of the full 360-degree interaction surface is input. Symmetry is then used by the program to define the interaction surface in the (- M2, +M3), (- M2, -M3), and (+ M2, -M3) quadrants. Unchecking the Doubly Symmetric About M2 and M3 check box means that you must input all 360 degrees of the interaction surface.
   - The Number of Curves edit box indicates the number of equally spaced P-M curves that will be used to define the interaction surface. If the Doubly Symmetric About M2 and M3 box is checked, then the curves are equally spaced between 0 and 90 degrees. If the Doubly Symmetric About M2 and M3 box is *not* checked, then the curves are equally spaced between 0 and 360 degrees.



Construction of User-Defined Interaction Surface

   - Defaults for the User Definition option are the Doubly Symmetric About M2 and M3 box checked, and 5 curves.
   - Click the Define/Show button to define or view the interaction surface curves. This opens the Interaction Surface Definition dialog box.

See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Pushover Curve
Override Axis Labels And Range
Override Structural Behavior Type

Concrete Interaction Surface

The methodology used for developing the interaction surface using the "Concrete, ACI 318-95 with phi = 1" option is the same as that documented in the SAP2000 Concrete Design manual except:

1. The phi factor is set equal to 1.
2. The maximum compressive axial load is not limited to Pmax.
3. For the pushover interaction surface, axial tension is positive, and axial compression is negative. This is an **opposite sign convention** from that used in concrete design.

Refer to ACI 318-95 for additional information.

Interaction Surface Definition

1. On the Define menu select Hinge Properties.... This displays the Define Frame Hinge Properties dialog box.

Click the Define New Property button, or select a defined hinge property from the Hinge Props list box and click the Modify/Show Property button to display the Frame Hinge Property Data dialog box. Click the Modify/Show button for a P-M2-M3 hinge type whose Hinge Type check box is checked to display the Frame Hinge Property Data For XXX dialog box. (Note that in the dialog box the XXX will be replaced with the hinge property name). Click the Interaction Surface button to display the Frame Hinge Interaction Surface dialog box. With the User Definition option selected, click the Define/Show button to display the Interaction Surface Definition dialog box.

2. Fill in the spreadsheet to indicate the various P-M interaction curves that together define the interaction surface. Note that there will be the same number of M columns as indicated in the Number of Curves edit box in the Frame Hinge Interaction Surface dialog box. You can view the additional M columns by using the scroll bars in the spreadsheet area. The angle associated with each M column is indicated at the top of the spreadsheet. The moment values in each M column should always be input as positive values. For the axial load values, tension should always be input positive and compression should always be input negative.
3. When the cursor in the spreadsheet is located in the P column, the plot at the right-hand side of the dialog box shows the P versus M Curve 1 diagram. When the cursor in the spreadsheet is located in an M column, the plot at the right of the dialog box shows P versus the M Curve associated with the cursor location.
4. Enter a scale factor in the Scale Factor For P edit box. The default for this value is 1.
5. Enter a scale factor in the Scale Factor For M edit box. The default for this value is 1. This scale factor applies to all of the M columns.
6. Click the OK button to accept all of the changes made in the Interaction Surface Definition dialog box. Clicking the Cancel button means that none of the changes will be accepted.

See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Pushover Curve
Override Axis Labels And Range
Override Structural Behavior Type

Spreadsheet

The spreadsheet in this dialog box has editing functions available to make input easier. You can use the standard Windows cut, copy and paste functions from the Edit menu in the dialog box, or use the standard Windows shortcut keys for these commands, Ctrl+X, Ctrl+C and Ctrl+V, respectively. You can also insert and delete rows using Edit menu commands. The Undo command in the Edit menu will undo the last action only. This spreadsheet does not have multilevel undo capabilities. Scroll bars are provided to show parts of the spreadsheet that are not currently visible.

Define Group Names

Defining Groups is a powerful tool in SAP2000. It helps in selecting elements, displaying and printing results as well as helping with design.

1 On the Define menu, click Groups… This will display the Define Groups dialog box with default Group ALL listed in the Groups list box.

2 To add a new Group, type in the new name in the Groups edit box and then click the Add new Group Name button.

3 To Change or Delete a group name, click on the Group name in the list box to select it and then click on the Change Group Name or Delete Group Name button.

3 Click OK.

Also see:

Assign Group Names

Define Static Load Cases

1 On the Define menu, click Static Load Cases… This will display the Static Load Case Names dialog box with a default Load Case, LOAD1 listed in the Loads list box.

2 To add a new Load Case name

- Type in the new name in the Load edit box
- Choose the load type to be Dead, Live, Quake, Wind, Snow or Other from the Type drop down list
- Type in a self-weight multiplier in the edit box and
- Click Add new Load button.

3 To change a Load Case name click on the name in the list box and then type over the changes in the edit boxes or choose a different Load Type from the drop down list and click Change Load button.

4 To delete a Load Case name click on the name in the list box and click Delete Load button.

5 Click OK.

Tip: Load Case names defined here are used in Load Combinations. Load Case Types are used by SAP2000 to develop automatic load combinations for Design/Stress Check of the structure.

Bridge Analysis

The bridge analysis can be used to compute influence lines for traffic on bridge structures and to analyze these structures for the response due to vehicle live loads. The vehicle live loads can be combined with static and dynamic loads and envelopes of the response can be computed.

Displacements, spring forces and Frame-element internal forces can be determined due to the influence of Vehicle live loads. Other element types (Shell, Plane, Asolid, Solid and NLLink) may be used; they contribute to the stiffness of the structure, but they are not analyzed for the effects of Vehicle loads.

You may select Vehicle live loads from a set of standard highway and railway Vehicles, or you may create your own Vehicle live loads.

What do you want to do ?

Define Lanes

Assign FRAME Lanes

Define Vehicles

Define Vehicles Classes

Define Bridge Response

Define Moving Loads

Define Joint Influence Lines

Display Frame Influence Lines

Define Lanes

This information is needed to define the traffic Lanes that are required for bridge moving-load analysis. Frame elements are used to represent the traffic Lanes.

1 On the Define menu, click Moving Load Cases … Lanes … This will display the Define Bridge Lanes dialog box.

2 Click Add new Lane button. This will display the Lane Data dialog box.

In the Lane Data dialog box

- Enter the Lane Name or accept the default name.

If you want to define the lanes explicitly:

- Enter the Label of the Frame representing the lane in the edit box.
- Enter the distance of the Lane (in the current length units) from the Frame representing the Lane in the Eccentricity edit box.
- Press the Add button to add the new lane.
- Click OK when you are finished adding frame members to the lane.

If you want to define the lanes through the graphic interface by Assigning Lanes:

- Click OK to just add the lane name to the list of lanes.

HINT: You can use the Insert button to insert a new Lane above the selected Lane.

3 The new Lane name will be added to the Lanes list box.

4 To Delete or Modify/Show a Lane, select it from the Lanes list box and then click on the Modify or Delete Lanes button.

5 Click OK.

Note: The frame members denoting the Lane should be nearly contiguous and progress in a consistent longitudinal direction. In the same way that a real lean on a road would be defined.

Also see:

Assign FRAME Lanes

Define Vehicles

This information defines the Vehicle loads that are required for bridge moving-load analysis.

On the Define menu, click Moving Load Cases ... Vehicles ... This will display the Define Vehicles dialog box.

To Add A New Standard Vehicle

1 Click Add new Standard Vehicle from the drop down list box. This will display the Standard Vehicle Data dialog box.

In the Standard Vehicle Data dialog box

- Select the Vehicle type from the drop down list.
- Depending on the selected vehicle you may be required to enter a Scale Factor or Dynamic Allowance in the appropriate edit box.
- Click OK.

2 The new Vehicle name will be added to the Vehicles list box.

3 To Delete or Modify/Show a Vehicle, select it from the Vehicles list box and then click on the Modify or Delete Lanes button.

4 Click OK.

To Add A New General Vehicle

1 Click Add General Vehicle from the drop down list box. This will display the General Vehicle Data dialog box.

In the General Vehicle Data dialog box

- Enter the Vehicle Name or accept the default name.
- Select what the Vehicle will be used for in the Usage area. More than one usage can be selected for a Vehicle.
- Enter uniform and first axle load values in the Leading and Trailing Loads area.
- Enter Floating Axial Load Values by selecting either a Single Valued Axle or a Double Valued Axle with separate values for Lane Moments.

- Enter Intermediate Loads.
  - Enter Uniform load between axles in the edit box.
  - Enter an Axle load in the edit box.
  - Enter a Minimum and Maximum Distance between the current and previous axle.
  - Press the Add button to add the axle information.
- Click OK.

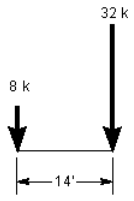
Note: A value of Zero for Max Distance indicates an infinite distance. Only one intermediate axle may have a value of Max Distance > Min Distance. The remaining intermediate axles must have a value of Max Distance = Min Distance.

2 The new Vehicle name will be added to the Vehicles list box.

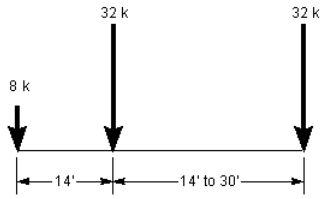
3 To Delete or Modify/Show a Vehicle, select it from the Vehicles list box and then click on the Modify or Delete Lanes button.

4 Click OK.

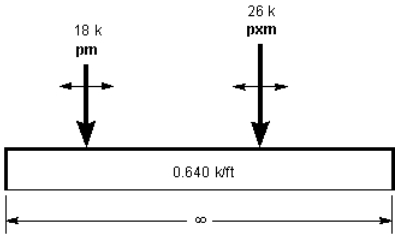
Standard Vehicles



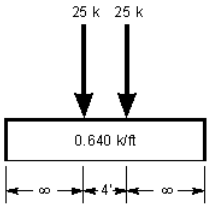
H20-44 Truck Load



HS20-44 Truck Load

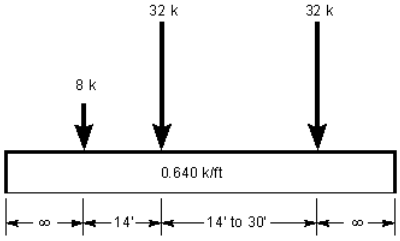


H20-44L and HS20-44L Lane Loads

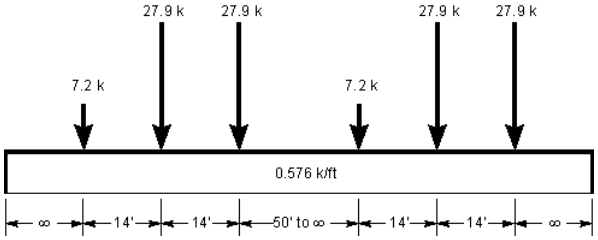


HL-93K Tandem and Lane Load

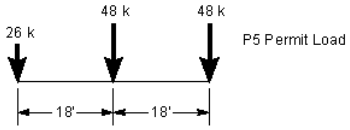
Note: All point loads will be increased by the dynamic load allowance, **im**, expressed as a percentage

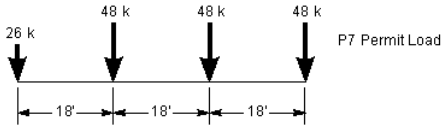


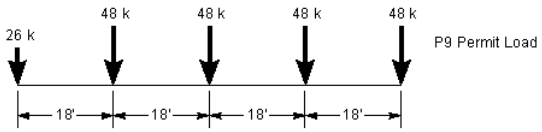
HL-93K Truck and Lane Load

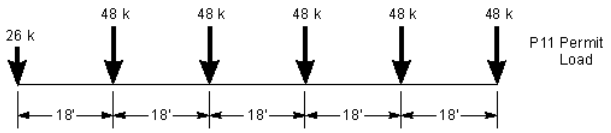


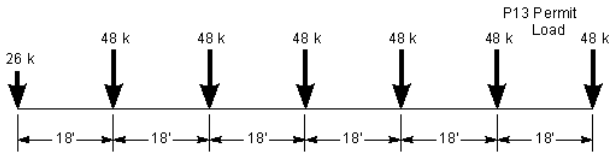
HL-93S Truck and Lane Load for Negative Moment and Reactions at Interior Piers

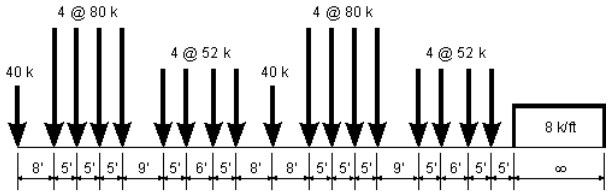

26 k
48 k
48 k
P5 Permit Load
18'
18'


26 k
48 k
48 k
48 k
P7 Permit Load
18'
18'
18'

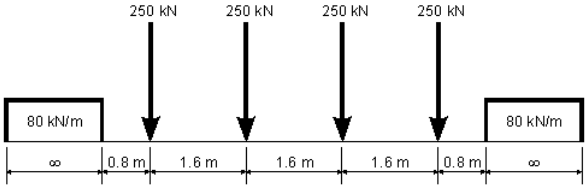

26 k
48 k
48 k
48 k
48 k
P9 Permit Load
18'
18'
18'
18'


26 k
48 k
48 k
48 k
48 k
48 k
P11 Permit Load
18'
18'
18'
18'
18'

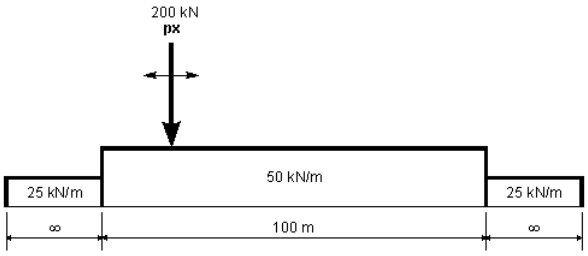

26 k
48 k
48 k
48 k
48 k
48 k
48 k
P13 Permit Load
18'
18'
18'
18'
18'
18'



Cooper E 80 Train Load

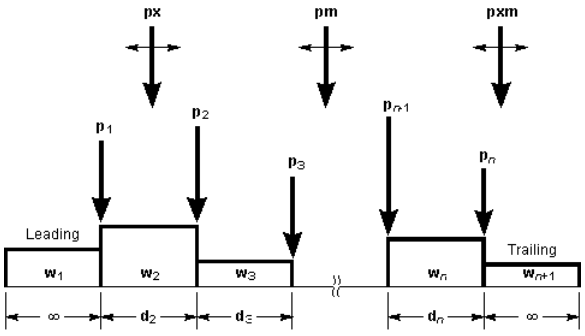


UIC80 Train Load



RL Train Load

General Vehicle



Notes:

(1) All loads are point loads or uniform line loads acting on the Lane center line

(2) Any of the point loads or uniform line loads may be zero

(3) The number of axles, *n*, may be zero or more

(4) One of the inter-axle spacings, $\mathbf{d}_2$ through $\mathbf{d}_n$, may vary over a specified range

(5) The locations of loads **px**, **pm**, and **pxm** are arbitrary

Define Vehicle Classes

This information is needed to define the Vehicle Classes or groups that are used to perform the bridge moving-load analysis. The order in which the Vehicles are assigned to a Vehicle class are not important. The Lanes assigned

to the Vehicle class will be checked for each Vehicle in that Vehicle Class.

1 On the Define menu, click Moving Load Cases … Vehicle Classes … This will display the Define Vehicle Classes dialog box.

2 Click Add Class button. This will display the Vehicle Class Data dialog box.

In the Vehicle Class Data dialog box

- Enter the Vehicle Class Name or accept the default name.
- Select a Vehicle Name from the drop down list box.
- Enter a Scale Factor for the Vehicle in the edit box.
- Press the Add button to add the Vehicle.
- Click OK when you are finished adding Vehicles to the Vehicle Class.

3 The new Vehicle Class name will be added to the Classes list box.

4 To Delete or Modify/Show a Vehicle Class, select it from the Classes list box and then click on the Modify or Delete Lanes button.

5 Click OK.

Define Bridge Response

This information is needed to allow you to selectively control what information is calculated for joints and frame elements in the computationally intensive moving-load analysis that is performed.

1 On the Define menu, click Moving Load Cases … Bridge Responses … This will display the Bridge Response Requests dialog box.

2 Check each of the response categories for which you want analysis done

- From the Select Group drop down list choose the group for which you want the responses computed. The default is ALL.

3 Under the Method of Calculation select Exact or Refinement Level and enter a value in the text input box. The value entered can be any positive integer. This feature is provided to give you a fast analysis option for preliminary analysis. The larger the integer the greater the level of refinement. The recommended Refinment Levels integers are 1 to 4.

4 Check Calculate Correspondence Values for Frames if you want to use the Max/Min Correspondence in your design of frame sections when using moving loads. This is a very time-intensive operation so you are advised to only use it when it is required.

5 Click OK.

Define Moving Loads

This information defines the Moving Load cases that determine the response to the Vehicles in the Vehicle Classes moving along the traffic Lanes.

1 On the Define menu, click Moving Load Cases … Moving Load … This will display the Define Moving Load Cases dialog box.

2 Click Add Load button. This will display the Moving Load Cases Data dialog box.

In the Moving Load Cases Data dialog box.

- Enter the Moving Load Case Name or accept the default name.
- Enter a Scale Factor for each of the Number of Lanes combinations selected from the drop down list box. This number is limited by the number of lanes defined in the Lane Data dialog box.
- Press the Add Assign button to assign a Vehicle class to the Moving Load Case.
  - In the Moving Load Case Assignment Data dialog box .
  - Select the Vehicle Class.
  - Enter the Vehicle Class Load Scale Factor.
  - Enter the Minimum and Maximum Number of Loaded Lanes for the selected Vehicle Class.

HINT: A value of zero for Minimum and Maximum Number of Loaded Lanes means include all the lanes .

- In the Assignment Lanes area Add the Lanes that the Vehicle Class can be assigned to.
- Click OK.
- To Modify/Show a Vehicle Class – Lane Assignment, select it from the Assignment Number drop down list box and press the Modify/Show Assign button.

- Add as many Vehicle Class Lane Assignments as required.
- Click OK when finished.

3 The new Moving Load Case name will be added to the Moving Loads list box.

4 To Delete or Modify/Show a Moving Load, select it from the Moving Loads list box and then click on the Modify or Delete Load button.

5 Click OK.

Define Joint Patterns

1 On the Define menu, click Joint Patterns… This will display the Pattern Names dialog box with a default pattern, DEFAULT listed in the Patterns list box.

2 To add a new pattern name type in the new name in the Patterns edit box and click Add new Pattern Name.

3 To change a pattern name click on the name in the list box and then type over the changes in the edit box and click Change Pattern Name.

4 To delete a pattern name click on the name in the list box and click Delete Pattern Name.

5 Click OK.

Tip: Pattern names defined here are used in Assign Joint Patterns .

Assign Dynamic Loads

Dynamic Loads (as ground motions) may be input as Response Spectra or Time Histories. Time Histories may also be applied as Joint Time Histories.

What do you want to do?

Define Time History Functions

Define Time History Cases

Define Response Spectrum Functions

Define Response Spectrum Cases

Define Time History Functions

1 On the Define menu, click Time History Functions… This will display the Time History Functions dialog box with a built-in RAMP function listed in the Functions list box.

To Define Function by specifying Time and Value:

- Click Add new Function button. This will display the Function Definition dialog box.
- Enter a pair of Time and Value (acceleration, displacement or velocity) in the Define Function edit box and then click Add. This will add the pair to the list box.
- Repeat to enter other pairs defining the function.
- Clicking on a value in the list box selects it and then it is possible to either modify or delete it.
- Click OK.

To Define Function from File:

- Click Add Function from File button. This will display the Time History Function Definition dialog box.
- Enter the function name in the Function Name edit box or accept the default name.
- Click on the Open File button to select a file.
- Choose the file name from the Pick Function Data File selection box.
- Click Open.
- Enter the number of points defined per line in the file.
- Click to check if the function is defined at equal intervals or as pairs of time and function values.
- If the function is defined at equal intervals then enter the time step in the edit box.
- Click OK.

2 The new Function Name will be added to the Functions list box.

3 To Delete or Modify/Show Function, select the function name from the list box and then click on the Show or Delete Function button.

4 Click OK.

Define Time History Cases

1 On the Define menu, click Time History… This will display the Time History Cases dialog box.

2 Click Add new History button. This will display the Time History Case Data dialog box.

In the Time History Case Data dialog box

- Enter the Time History Case Name or accept the default name.
- In the Options area:
  - Choose Analysis Type to be Linear , Periodic or NonLinear from the drop down box.
  - Click Modify/Show button to specify Modal Damping.
  - Enter the Number of Output Time Steps in the edit box.
  - Enter the Output Time Step size in the edit box.
  - If desired, choose the name of the Time History from which this Time History should be started by clicking on the Start From drop down list.
  - If you want an Envelope of the results to be recovered for reporting in the *filename*.OUT file or for use in design, then select the Envelopes check box.
- In the Load Assignments area:
  - Choose the Load Case from the Load drop down box (acc dir 1 , acc dir 2 , acc dir 3 and LOAD1 are listed as default cases)
  - Choose the Function Name from the Function drop down box (RAMP function is listed as default function)
  - Enter the Scale Factor, Arrival Time and Angle of excitation direction in the edit boxes. The Angle is only available for base accelerations and is turned off for Load Cases.
  - Click Add. This will add a Load Assignment to the list box below.
  - To change or delete the definition of a Load Assignment, select the assignment from the list box and then click on the Modify or Delete button.
  - Repeat to add more load assignments to make up a Time History Case.
- Click OK.

3 The new Time History Case name will be added to the History Cases list box.

4 To Delete or Modify/Show History, select the history case from the list box and then click on the Modify or Delete History button.

5 Click OK.

Note: You will not be able to get Time History results in the *filename*.OUT file or do a fast design using envelope results unless you have the Envelopes option checked.

Define Response Spectrum Functions

1 On the Define menu, click Response Spectrum Functions… This will display the Response Spectrum Functions dialog box with built in UNIT, UBC94S1 , UBC94S2 and UBC94S3 functions listed in the Functions list box.

To Define Function by specifying Time and Acceleration:

- Click Add new Function button..
- Enter the function name in the Function Name edit box or accept the default name.
- Enter a pair of Period and Acceleration values in the Define Function edit boxes and then click Add. This will add the pair to the list box.
- Repeat to enter other pairs defining the function.
- Clicking on a value in the list box selects it and then it is possible to either modify or delete it.
- Click OK.

To Define Function from File:

- Click Add Function from File button..
- Enter the function name in the Function Name edit box or accept the default name.
- Click on Open File button to select a file.
- Choose the file name from the Pick Function Data File selection box.
- Click Open.
- Enter the number of points defined per line in the file.
- Click to check if the function is defined at equal intervals or as pairs of Period and Acceleration values.
- If the function is defined at equal intervals then enter the Period step in the edit box.
- Click OK.

2 The new Function Name will be added to the Functions list box.

3 To Delete or Modify/Show Function, select it from the Functions list box and then click on the Modify or Delete Function button.

4 Click OK.

Define Response Spectrum Cases

1 On the Define menu, click Response Spectrum Cases… This will display the Response Spectra dialog box.

2 Click Add new Spectra button. This will display the Response Spectrum Data dialog box.

In the Response Spectrum Data dialog box

- Enter the Response Spectrum Name or accept the default name.
- Enter the excitation angle in the edit box.
- Choose the Modal Combination Technique by clicking on either **CQC, SRSS, ABS or GMC.**
- Enter the Damping ratio in the edit box. This ratio is used for modal combination.
- If GMC is chosen then enter characteristic frequencies F1 and F2 as defined in ASCE 4 for GMC.
- Choose the Directional Combination Technique by clicking on either SRSS or ABS.
- In the Input Response Spectra area:
  - Select the Function Names for U1, U2 and U3 directions from the Function drop down boxes. Unit function is the default built in function.
  - Enter the Scale Factor for each function in the Scale Factor edit box. The default value is 1.0
- Click OK.

3 The new Spectrum name will be added to the Spectra list box.

4 To Delete or Modify/Show Spectra, select it from the Spectra list box and then click on the Modify or Delete Spectra button.

5 Click OK.

Define Load Combinations

1 On the Define menu, click Load Combinations… This will display the Load Combinations dialog box.

2 To add a new Load Combination, click Add new Combo button. This will display the Load Combination Data dialog box.

In the Load Combination Data dialog box

- Type in the combination name in the Load Combination Name edit box.
- Select the Load Combination Type: ADD, ENVE, ABS, SRSS .
- Type in the title (description of load combo) in the Title edit box.
- Select an Analysis Case name from the Case Name drop down list box and type in the multiplier in the Scale Factor edit box and click ADD. This will add the Analysis Case and the multiplier to the list box.
- Similarly select other Analysis Cases and add them to the list box to complete the Load Combination definition.
- It is possible to modify or delete an Analysis Case from the Combination definition by selecting the Analysis Case from the list box and clicking the Modify or Delete button.
- Select whether the combo should be used as a part of the Steel or Concrete design loading combinations.
- Click OK. This will add the Load Combination to the Combinations list box.

3 To include the default design load combinations for the code specified in Concrete and Steel Preferences, click on the Add Default Design Combo Button. The program will add Steel design combos if the Steel Design is selected in the Design menu and Concrete design combos if the Concrete Design is selected in the Design menu. The Type assignment for the Static Load Case is used to determine the appropriate load factors.

4 To modify a Load Combination, click on the name in the Combinations list box to select it and then click Modify/Show Combo button. This will display the Load Combination Data dialog box. Make the appropriate

changes and then click OK.

5 To delete a Load Combination, click on the name in the Combinations list box to select it and then click Delete Combo button. This will delete the Load Combination.

6 Click OK.

Tip: Load Combinations defined here are used in Select Design Combos to define design loading combinations.

DRAW MENU

Draw Mode

Clicking on any of these buttons  from the **Side Toolbar** puts SAP2000 in the Draw mode.
The Draw mode allows for drawing of new members and editing one element or a joint at a time.
Draw Mode is also the default mode when a NEW model is started.
Select various Draw options by clicking as follows:

To Draw or Edit	Do this
Edit/Shape/Move Element	Click on  and click on a joint or Element. Use the handles to move or stretch.
An extra or Special Joint	Click on  and click on any point to add a Joint
A Frame Element (joint to joint)	Click on 
Quick draw Frame Elements	Click on 
A Quadrilateral Shell Element	Click on 
A Rectangular Shell Element	Click on 
Quick draw Shell Elements	Click on 

Draw Members

There are two ways to draw Elements:

Quick Draw	a single click on a grid segment (for FRAME) or an area bounded by four grids (for SHELL) will draw the element. The ESC key takes you out of the Draw Mode and puts you in the Select mode.
Joint to Joint Draw	a sequential drawing of elements by clicking on previously defined joints or points in space. Double click on a joint or pressing ENTER terminates the sequential draw and the ESC key takes you out of the Draw Mode and puts you in the Select mode.

What do you want to do?

 Draw a Quick FRAME Element

Draw a FRAME Element by clicking on 2 Joints

Draw a Quick SHELL Element

Draw a SHELL Element by clicking on 4 Joints

Draw an NLLINK Element

Also See

Snap Tools

Drawing Constraints

Reshape Element

Reshape Element option is used to edit element forms or shapes. With the help of this option it is possible to move FRAME or SHELL elements, stretch or shrink FRAME elements and reshape SHELL elements. To move or reshape an element do the following:

Add in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to reshape the member.
3. Click on  to switch to Reshape Element mode.
4. Click once on a FRAME or a SHELL element to select it. This will display the member end handles.
5. Grabbing one of the handles of the element and moving it will stretch, shrink, rotate or reshape the element.
6. Grabbing the element (any point away from the handles) and dropping it at another location will allow you to move the element.

Add in 3-D

In 3-D, the element editing works the same way as 2-D except that the handle may only be dropped at another pre-defined JOINT or Grid intersection.

Note: Only one Element at a time may be moved or reshaped through this option.

Also See

Snap Tools

Drawing Constraints

Add Special Joints

In building a model in SAP2000 it is not necessary to pre-define Joints. The Joints are automatically added to the ends or corners of the elements. Special Joints are those Joints which are added by the user. These Joints may be necessary to add in rare cases such as at one end of an NLLink Element - an end where there is no other element present and hence no automatically generated Joint. To add a Special Joint do the following:

Add in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to add a Joint.
3. Click on  to switch to Add Special Joint mode.
4. Click on a Grid intersection or any other point in that plane to add a Joint.

Add in 3-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off

the Grids.

2 Click on  to switch to Add Special Joint mode.

3 In the 3-D view, click on a Grid intersection to add a Joint.

Note: The Joints may only be added at grid intersections in 3-D view.
Right Click on a Joint will open Joint Information edit box. You may edit the Joint location in this edit box to locate the Joint precisely in the desired location.

Draw a Frame Element from Joint to Joint

Draw in 2-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.

2 Select 2-D Plane in which you want to draw the member.

3 Click on  to switch to drawing a FRAME element between 2 points.

4 Click on a Grid intersection, a previously defined Joint or any point in the plane. Click again on another point to add a single FRAME element. Every subsequent click will add another FRAME element unless a double click on the same joint is performed or the ENTER key is pressed. Pressing the ESC key will also terminate the sequential draw and take you out of the Draw mode.

Draw in 3-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.

2 Click on  to switch to drawing a FRAME element between 2 points.

3 In the 3-D view click on a Grid intersection or a previously defined Joint (a small red ball appears to confirm that the joint has been selected).

4 Click on another Grid intersection or a Joint to add a FRAME element. Every subsequent click on a selected point will add another FRAME element unless a double click on the same joint is performed or the ENTER key is pressed. Pressing the ESC key will also terminate the sequential draw and take you out of the Draw mode.

Note: It is not possible to draw a FRAME element between any two arbitrary points in space in the 3-D view.

Draw a Quick Frame Element

Draw in 2-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.

2 Select 2-D Plane in which you want to draw the member.

3 Click on  to choose the quick draw mode.

4 Click on a grid segment to draw a quick single FRAME element.
Clicking in a space bounded by 4 grid lines will add a Cross Brace.

Draw in 3-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.

2 In the 3-D view click on a grid segment. This will add a single element between two grid intersections.

Note: It is not possible to Quick Draw a Cross Brace in 3-D.

Draw a Shell Element between 4 Joints

Draw in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to draw the member.
3. Click on  to switch to drawing a SHELL element between 4 Joints.
4. Click on a Grid intersection, a previously defined Joint or any point in that plane.
5. Click again on 3 other points either clockwise or counterclockwise to add a SHELL element.

Note: If you want to add a triangular element the fourth point clicked should be the same as the first.

Draw in 3-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Click on  to switch to drawing a SHELL element between 4 Joints.
3. In the 3-D view click on a Grid intersection or a previously defined Joint (a small red ball appears to confirm that the point has been selected).
4. Click on 3 other similar points either clockwise or counterclockwise to add a SHELL element .

Note: It is not possible to draw a SHELL element between any 4 arbitrary points in space in the 3-D view.

Draw a Rectangular Shell Element

Draw in 2-D Only

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to draw the member.
3. Click on  to switch to drawing a SHELL element by defining the two opposite corners of the rectangular shape.
4. Click on a Grid intersection, a previously defined Joint or any point in that plane.
5. Click again on the opposite corner of the shape to finish defining the element.

Draw a Quick Shell Element

Draw in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to draw the member.
3. Click on  to choose the quick draw mode.
4. Click in a grid space, bounded by 4 grid lines, to draw a quick single SHELL element.

Draw in 3-D

Note: It is not possible to draw a Quick SHELL Element in 3-D view.

Draw an NLLink Element

Draw in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.
2. Select 2-D Plane in which you want to draw the member.
3. On the Draw menu, click Draw NLLink Element. This puts you in the non-linear link drawing mode.
4. Click on a previously defined Joint in that plane. Click again on another Joint to add a single NLLink element. Click again on two Joints to add another NLLink element. Double clicking on the same Joint will add a zero length link element. ESC key will take you out of the NLLink drawing mode.

Draw in 3-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. On the Draw menu, click Draw NLLink Element. This puts you in the non-linear link drawing mode..
3. In the 3-D view, click on a previously defined Joint (a small red ball appears to confirm that the joint has been selected). Click again on another Joint to add a single NLLink element. Click again on two Joints to add another NLLink element. Double clicking on the same Joint will add a zero length link element. ESC key will take you out of the NLLink drawing mode.

Note: It is not possible to draw an NLLink element between any two arbitrary points in space. The link element may only be drawn between previously defined or generated Joints.

Edit Grids

After defining your coordinate system you can edit the location of the individual grids in the active coordinate system.

1. On the Draw menu, click Edit Grid or double click on a grid line in the graphics screen.
2. In the Modify Grids dialog box, Select Direction of the grids you want to edit.
3. In the location area:

To Move a Grid

- Select the grid location from the list.
- Edit the grid's location in the text edit box.
- Click the Move Grid Line button.

To Add a new Grid

- Enter the grid's location in the text edit box.
- Click the Add Grid Line button.

To Delete a Grid

- Select the grid location from the list.
- Click the Delete Grid Line button.

4. The Delete All button will delete all the grids in the coordinate system.
5. Check the Lock Grid Lines if you want to lock the grid lines from being moved using the method discussed in the note below.
6. Check the Snap to Grid lines if you want the feature on.
7. Check Glue Joints to Grid Lines if you want the joints to move with the grids. This is a very powerful way to quickly edit the structure without having to redefine joint locations.
8. Click OK when finished.

Note: From the display screen with the Reshape Element option set, you can add a new grid line by holding the CTRL-key down and clicking on an existing grid line and dragging away a duplicate.

Also See

Setup Coordinate System

Snap Tools

The snap tools are essentially a fast and accurate way to draw and edit elements. Snap tools find the closest snap location to your pointer as you move it over your model. The snap tools can be turned on and off as you draw, so you can snap to different locations for every point. More than one snap tool can also be set at the same time giving you a

choice of snap locations. The closest snap location is indicated by a red dot and a text field that identifies which snap location was found. The area that is searched for snap points is set by the Screen Snap to Tolerance located in the Preferences dialog box under the Options menu.

The Snap Tools include:

- Snap to Joints and Grid Points
- Snap to Midpoints and Ends
- Snap to Element Intersections
- Snap to Perpendicular
- Snap to Lines and Edges

It is sometimes helpful to use the snap tools to quickly draw an object then use the edit tools to modify the object.

Note: The snap options are evaluated in the above order. So if a location has both an end point and an intersection, the end point will be found first because it is evaluated first.

Also See

Drawing Constraints

Snap to Joints and Grid Points

The Snap to Joints and Grid Points tool finds and snaps to the Joint or intersection of Grid Lines closest to the mouse pointer.

1. Select the Snap to Joints and Grid Points tool, by pressing the button on the side toolbar or selecting it from the Snap to option under the Draw menu.
2. Move the pointer in the graphics window.
3. When a snap location is found close to the pointer, a red dot and a text field describing the snap location identify it.
4. When the desired snap location is found press the left mouse button to accept it.
5. Modify the Snap options as required and continue entering elements.

Note: Snap tools can also be used when reshaping an element.

Also See

Snap Tools

Snap to Midpoints and Ends

Snap to Element Intersections

Snap to Perpendicular

Snap to Lines and Edges

Drawing Constraints

Snap to Midpoints and Ends

The Snap to Midpoints and Ends tool finds and snaps to the closest midpoint or end of frames and shells. It will also snap to the end points of NLLink elements.

1. Select the Snap to Midpoints and Ends tool, by pressing the button on the side toolbar or selecting it from the Snap to option under the Draw menu.
2. Move the pointer in the graphics window.
3. When a snap location is found close to the pointer, a red dot and a text field describing the snap location identify it.
4. When the desired snap location is found press the left mouse button to accept it.

5 Modify the Snap options as required and continue entering elements.

Note: **Snap tools can also be used when reshaping an element.**

Also See

Snap Tools
Snap to Joints and Grid Points
Snap to Element Intersections
Snap to Perpendicular
Snap to Lines and Edges
Drawing Constraints

Snap to Intersections

The Snap To Intersections tool finds and snaps to the intersection of two frame elements and a frame element with a shell element, regardless of whether or not there is a joint at the intersection location.

1 Select the Snap to Intersections tool, by pressing the  button on the side toolbar or selecting it from the Snap To option under the Draw menu.
2 Move the pointer in the graphics window.
3 When a snap location is found close to the pointer, a red dot and a text field describing the snap location identify it.
4 When the desired snap location is found press the left mouse button to accept it.
5 Modify the Snap options as required and continue entering elements.

Note: **Snap tools can also be used when reshaping an element.**

Also See

Snap Tools
Snap to Joints and Grid Points
Snap to Midpoints and Ends
Snap to Perpendicular
Snap to Lines and Edges
Drawing Constraints

Snap to Perpendicular

The Snap To Perpendicular tool finds and snaps to the intersection point of a line drawn from the last entered point perpendicular to the frame elements or shell edge closest to the mouse pointer. This is a helpful way to make sure that lines are perpendicular to each other.

1 Select the Snap To Perpendicular tool, by pressing the  button on the side toolbar or selecting it from the Snap To option under the Draw menu.
2 Move the pointer in the graphics window.
3 When a snap location is found close to the pointer, a red dot and a text field describing the snap location identify it.
4 When the desired snap location is found press the left mouse button to accept it.
5 Modify the Snap options as required and continue entering elements.

Note: **Snap tools can also be used when reshaping an element.**

Also See

Snap Tools
Snap to Joints and Grid Points
Snap to Midpoints and Ends
Snap to Element Intersections
Snap to Lines and Edges
Drawing Constraints

Snap to Lines and Edges

The Snap to Lines and Edges tool finds and snaps or “hugs” the closest frame element, grid line or edge of the closest shell element. Even though this does not provide the same level of accuracy as the other snap options, it is a good way to make sure that the element being drawn is located on the element being “hugged”.

1. Select the Snap to Lines and Edges tool, by pressing the button on the side toolbar or selecting it from the Snap To option under the Draw menu.
2. Move the pointer in the graphics window.
3. When a snap location is found close to the pointer, a red dot and a text field describing the snap location identify it.
4. When the desired snap location is found press the left mouse button to accept it.
5. Modify the Snap options as required and continue entering elements.

Note: **Snap tools can also be used when reshaping an element.**

Also See

Snap Tools
Snap to Joints and Grid Points
Snap to Midpoints and Ends
Snap to Element Intersections
Snap to Perpendicular
Drawing Constraints

Drawing Constraints

Drawing Constraints provide the capability to constrain one of the axes when drawing or reshaping an element. In this manner, one can quickly draw a frame element parallel to one of the global axes. The constraint tools can be reached from the Draw menu or by pressing the X, Y or Z keys on your keyboard while drawing an element.

Drawing constraints Include:

- Constrain X: Locks the X component of the next point to that of the previous one
- Constrain Y: Locks the Y component of the next point to that of the previous one
- Constrain Z: Locks the Z component of the next point to that of the previous one
- Press the space bar or select None from the Draw Constraints menu if you want to remove the drawing constraint option

There are three steps to using the constraint tools:

1. Locate the first point.
2. Press one of the constraint keys - X, Y or Z or select the Constraint Drawing To… option from the Draw menu.
3. Locate the next point. SAP2000 only picks up the unconstrained component of the selected point.

Note: Snaps can be used in conjunction with constraints, but are not required. Only the unconstrained component of the selected snap point is used when a constraint has been selected. Snap tools can also be used when reshaping an element.

Also See

 Snap Tools

Initialize a New Labeling Scheme

The labels in SAP2000 are alphanumeric. By default the program automatically assigns a numeric numbering scheme to the Joints and Elements. However, it is possible assign an alphanumeric labeling scheme by giving an alpha prefix and a starting numeric sequence. All Joints and Elements, added after the scheme is initialized, will be affected by the scheme.

To Initialize a New Labeling Scheme:

1. On the Draw menu, click New Labels... This will display the New Labels dialog box.
2. In the New Labels dialog box enter an alphanumeric prefix, the starting numeric number for the scheme to be added to the prefix and a numeric increment for the numeric sequence for Joints, Frames and Shells.
3. Click OK.

SELECT MENU

Selection Procedures

Multiple selection of Objects (i.e. Elements or Joints) may be made in any session except while in the DRAW mode. In the DRAW mode only one Joint or Element  may be selected for editing purposes. Clicking on an object selects it. Clicking again on a selected object will deselect it.

You may Also

 Select Objects by drawing a window around them
 Select Objects by drawing a line intersecting them
 Select Objects lying in XY, XZ and YZ planes
 Select Objects by pre-assigned Group names
 Select Objects by Types of Frame Sections
 Select Objects by Types of Shell Sections
 Select Objects by NLLink Properties
 Select Objects by Types of Constraints
 Select Objects by Labels
 Select All Objects

Note: Deselect Options are the same as Select Options and are available through the Select menu. When Deselect options are activated they will deselect the previously selected objects.

Hints: To bring back the previous selection click  from the side toolbar or on the Select menu, click Get Previous Selection.

To clear the entire selection click  from the side toolbar or on the Select menu, click Clear Selection.

Select Objects by Window

1. Click  from the side toolbar or on the Select menu, click Select and then Pointer/Window from the sub-menu. This will switch you into the selection mode.
2. Select Objects by either clicking on them or drawing a window around them. Only those elements completely within the window are selected.

HINT: It is sometimes easier to select a single Joint by drawing a window around it.

Select Objects by Intersecting Line

1. Click  from the side toolbar or on the Select menu, click Select and then Intersecting Line from the sub-menu. This will switch you into the selection mode.
2. Draw a line by clicking at one end and then holding down the left mouse button drag the pointer to the other end of the line. All Objects crossing this line will be selected.

HINT: It is sometimes easier to select a single or a few Elements by drawing a line across.

Select Objects by 2D Planes

1. On the Select menu, click Select and then XY Plane or XZ Plane or YZ Plane from the sub-menu. This will switch you into the selection mode.
2. Pick any point on the plane desired and all Elements on that Plane will be selected.

HINT: It is a useful selection for editing purposes e.g. adding or deleting a story may very easily be accomplished by this selection option.

Select Objects by Groups

1. On the Select menu, click Select and then Groups from the sub-menu. This will open the Select Groups list box. Click to select one or more of the previously defined groups.
2. Click OK. All Elements belonging to those groups will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output based upon group selections.

Select Objects by Frame Sections

1. On the Select menu, click Select and then Frame Sections**...** from the sub-menu. This will open the Select Sections list box. Click to select one or more of the previously defined sections.
2. Click OK. All Frame Elements belonging to the selected section types will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output Frame Elements based upon selected sections.

Select Objects by Shell Sections

1. On the Select menu, click Select and then Shell Sections**...** from the sub-menu. This will open the Select Sections list box. Click to select one or more of the previously defined sections.
2. Click OK. All Shell Elements belonging to the selected Sections will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output Shell Elements based upon selected sections.

Select Objects by NLLink Properties

1. On the Select menu, click Select and then NLLink Properties**...** from the sub-menu. This will open the Select Properties list box. Click to select one or more of the previously defined nonlinear link properties.
2. Click OK. All nonlinear link Elements belonging to the selected properties will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output nonlinear link Elements based upon selected properties.

Select Objects by Constraints

1. On the Select menu, click Select and then Constraints**...** from the sub-menu. This will open the Select Constraints list box. Click to select one or more of the previously defined Constraints.
2. Click OK. All Joints belonging to the selected Constraints will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output support Joints or Joints with other type of Constraints.

Select Objects by Labels

1. On the Select menu, click Select and then Labels from the sub-menu This will open the Select by Labels dialog box.
   - Select the Element Type from the drop down list box.
   - Enter Start and End labels.
   - Enter Increment between labels.
   - Click OK to select the elements using the specified criteria.
2. The total number of Elements selected is displayed on the status bar.

HINT: It is a useful technique to find specific elements in a large model.

Select All Objects

1. Click  from the side toolbar or on the Select menu, click Select and then All from the sub-menu. This will select all of the objects in the model.
2. The total number of all the Joints and Elements selected is displayed on the status bar.

HINT: It is a useful technique to find out the total number of Joints or Elements in the model.

ASSIGN MENU

Assign Options

Clicking on any of these buttons  from the Toolbar allows you to Assign properties, loads and restraints/constraints to your selection. Members or Joints have to be selected first in order to make an assignment. Select various Assign options by clicking as follows:

To Assign	Do this
Supports or Restraints to Joints	Click on 
Sections to Frame Elements	Click on 
Thickness to Shell Elements	Click on 
Loads on Joints	Click on 
Span and Point Loads on Frame Elements	Click on 
Uniform Loads on Shell Elements	Click on 

Assign Section Properties

FRAME Elements

1. Select one or more Frame Elements to which you want to Assign the same Section properties.
2. On the Assign menu, click Frame and then Sections… in the sub-menu.
3. In the Frame Sections display box:
   - Select a previously defined Section
   - Click the OK button

Note: If you have not previously defined frame sections, you may define them by importing new sections from a Section property file or adding a user defined section from the “Frame Sections” display box.

SHELL Elements

1. Select one or more Shell Elements to which you want to Assign Section properties.
2. On the Assign menu, click Shell and then Sections… in the sub-menu.
3. In the Shell Sections display box:
   - Select a previously defined Section

- Click the OK button

Note: If you have not previously defined shell sections, you may add new sections by clicking on Add New Section.

Assign Joint Constraints

Constraints are applied to selected joints.

1. Select the Joints to which you want to apply Constraints.
2. On the Assign menu, click Joint and then Constraints… in the sub-menu. This will display the Constraints dialog box.
3. In the Constraints dialog box:
   - Add a new Constraint by clicking on the Add drop down list and selecting the appropriate type of constraint. Type in the new name or accept the default name, choose the axis (X, Y or Z) or Auto to define the direction of the constraint and then click OK.
4. Click OK.

How do I ?

Add some Joints to an existing Constraint

Delete or Remove some Joints from an existing Constraint

Add Joints to an existing Constraint

1. Select the Joints you want to add to an existing Constraint.
2. On the Assign menu, click Joint and then Constraints… in the sub-menu. This will display the Constraints dialog box.
3. In the Constraints dialog box:
   - Choose the existing Constraint from the Constraints list box
   - Click OK
4. The Joints will be added to the existing Constraint.

Delete or Remove Joints from an existing Constraint

1. Select the Joints you want to remove from an existing Constraint.
2. On the Assign menu, click Joint and then Constraints… in the sub-menu. This will display the Constraints dialog box.
3. To remove the joint from only one constraint:
   - In the Constraints dialog box:
     - Choose the existing Constraint from the Constraints list box
     - Click on Modify/Show Constraint
     - Choose Remove Constraint
     - Click OK
     - The Joints will be removed from the existing Constraint.
4. To remove the joint from all constraints:
   - In the Constraints dialog box:
     - Choose Null from the Constraints list box
     - Click OK
     - The Joints will be removed from all Constraint.

Generalized Constraints

Constraints are used to enforce certain types of rigid-body behavior, to connect together different parts of the model, and/or to impose certain types of symmetry conditions.

A constraint consists of a set of two of more **constrained joints**. The displacements of each pair of joints in the constraint are related by constraint equations. The types of behavior that can be enforced by constraints are, Rigid-body behavior, Equal-displacement behavior, Symmetry and anti-symmetry conditions.

Also See

- Rigid Body: fully rigid for all displacements
- Rigid Diaphragm: rigid for membrane behavior in a plane
- Rigid Plate: rigid for plate bending in a plane
- Rigid Rod: rigid for extension along an axis
- Rigid Beam: rigid for beam bending on an axis
- Equal-displacement behavior
- Partial Connection
- Local Constraint
- Symmetry About a Plane
- Anti-symmetry About a Plane
- Axisymmetry
- Cyclic symmetry
- Symmetry About a Point

Hint: The use of constraints reduces the number of equations in the system to be solved and will usually result in increased computational efficiency.

Body Constraint

A Body Constraint causes all of its constrained joints to move together as a three-dimensional rigid body. Effectively, all constrained joints are connected to each other by rigid links and cannot displace relative to each other.

This Constraint can be used to:

- Model rigid connections, such as where several beams and/or columns frame together
- Connect together different parts of the structural model that were defined using separate meshes
- Connect Frame elements that are acting as eccentric stiffeners to Shell elements

Joint Connectivity

Each Body Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space.

Local Coordinate System

Each Body Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. These correspond to the X, Y, and Z axes of the global coordinate system, respectively. The actual orientation of the local axes is not important since the constraint equations are independent of the coordinate system.

Diaphragm Constraint

A Diaphragm Constraint causes all of its constrained joints to move together as a planar diaphragm that is rigid against membrane deformation. Effectively, all constrained joints are connected to each other by links that are rigid in the plane, but do not affect out-of-plane (plate) deformation.

This Constraint can be used to:

- Model concrete floors (or concrete-filled decks) in building structures, which typically have very high in-

plane stiffness
- Model diaphragms in bridge superstructures

The use of the Diaphragm Constraint for building structures eliminates the numerical-accuracy problems created when the large in-plane stiffness of a floor diaphragm is modeled with membrane elements. It is also very useful in the lateral (horizontal) dynamic analysis of buildings, as it results in a significant reduction in the size of the eigenvalue problem to be solved.

Joint Connectivity

Each Diaphragm Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space, but for best results all joints should lie in the plane of the constraint. Otherwise, in-plane moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Local Coordinate System

Each Diaphragm Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 3 is always normal to the plane of the constraint. The program arbitrarily chooses the

orientation of axes 1 and 2 in the plane. The actual orientation of the planar axes is not important since only the normal direction affects the constraint equations.

Plate Constraint

A Plate Constraint causes all of its constrained joints to move together as a flat plate that is rigid against bending deformation. Effectively, all constrained joints are connected to each other by links that are

rigid for out-of-plane bending, but do not affect in-plane (membrane) deformation.

This Constraint can be used to:

- Connect structural-type elements (Frame and Shell) to solid-type elements (Plane and Solid); the rotation in the structural element can be converted to a pair of equal and opposite translations in the solid element by the Constraint
- Enforce the assumption that “plane sections remain plane” in detailed models of beam bending

Joint Connectivity

Each Plate Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space. Unlike the Diaphragm Constraint, equilibrium is not affected by whether or not all joints

lie in the plane of the Plate Constraint.

Local Coordinate System

Each Plate Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 3 is always normal to the plane of the constraint. The program arbitrarily chooses the orientation of axes 1 and 2 in the plane. The actual orientation of the planar axes is not important since only the normal direction affects the constraint equations.

Rod Constraint

A Rod Constraint causes all of its constrained joints to move together as a straight rod that is rigid against axial deformation. Effectively, all constrained joints maintain a fixed distance from each other in

the direction parallel to the axis of the rod, but translations normal to the axis and all rotations are unaffected.

This Constraint can be used to:

- Prevent axial deformation in Frame elements
- Model rigid truss-like links

An example of the use of the Rod Constraint is in the analysis of the two-dimensional frame shown in Figure 30 (page 128) of the User’s Manual. If the axial deformations in the beams are negligible, a single Rod Constraint could be defined containing the five joints. Instead of five equations, the program would use a single equation to define the X-displacement of the whole floor. However, it should be noted that this will result in the axial forces of the beams being output as zero, as the Constraint will cause

the ends of the beams to translate together in the X-direction. Interpretations of such results associated with the use of Constraints should be clearly understood.

Joint Connectivity

Each Rod Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space, but for best results all joints should lie on the axis of the constraint. Otherwise,

transverse (bending) moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may

not be in equilibrium.

Local Coordinate System

Each Rod Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 1 is always the axis of the constraint. The program arbitrarily chooses the orientation

of the transverse axes 2 and 3. The actual orientation of the transverse axes is not important since only the axial direction affects the constraint equations.

Beam Constraint

A Beam Constraint causes all of its constrained joints to move together as a straight beam that is rigid against bending deformation. Effectively, all constrained joints are connected to each other by links that are rigid for off-axis bending, but do not affect translation along or rotation about the axis.

This Constraint can be used to:

- Connect structural-type elements (Frame and Shell) to solid-type elements (Plane and Solid); the rotation in the structural element can be converted to a pair of equal and opposite translations in the solid element by the Constraint
- Prevent bending deformation in Frame elements

Joint Connectivity

Each Beam Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space, but for best results all joints should lie on the axis of the constraint. Otherwise,

torsional moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Local Coordinate System

Each Beam Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 1 is always the axis of the constraint. The program arbitrarily chooses the orientation

of the transverse axes 2 and 3. The actual orientation of the transverse axes is not important since only the axial direction affects the constraint equations.

Equal Constraint

An Equal Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the constraint local coordinate system. The other degrees of freedom are unaffected.

The Equal Constraint differs from the rigid-body types of Constraints in that there is *no coupling* between the rotations and the translations.

This Constraint can be used to:

- Model symmetry and anti-symmetry conditions with respect to a plane
- Partially connect together different parts of the structural model, such as at expansion joints and hinges

For fully connecting meshes, it is better to use the Body Constraint when the constrained joints are not in exactly the same location.

Joint Connectivity

Each Equal Constraint connects a set of two or more joints together. For a given Constraint, if *any* of the selected degrees of freedom are negative (i.e., opposite), only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space, but for best results all joints should share the same location in space if used for connecting meshes. Otherwise, moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Such restrained moments may also be generated when Equal Constraints are used for symmetry purposes. They are necessary to enforce the desired symmetry or anti-symmetry of the displacements when the applied loads are not correspondingly symmetric or anti-symmetric.

Local Coordinate System

Each Equal Constraint uses a fixed coordinate system, **csys**, that you specify. The default for **csys** is zero, indicating the global coordinate system. The axes of the fixed coordinate system are denoted X, Y, and Z.

Selected Degrees of Freedom

For each Equal Constraint you may specify a list, **cdofs**, of up to six degrees of freedom in coordinate system **csys** that are to be constrained. The degrees of freedom are indicated as UX, UY, UZ, RX, RY, and RZ. A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., -UX.

Partial Connection

When joints are being connected, all specified degrees of freedom are positive. For example, consider an idealized hinge connection of eight space-truss members. Only displacements are continuous across the hinge, not rotations. Each truss member is connected to a separate joint (node) at the connection. One Equal Constraint is defined for the eight constrained joints. The degrees of freedom that would be specified for this Constraint are: UX, UY, and UZ.

The eight joints should be coincident or the axes of the truss members should all intersect at the same point. Otherwise, moments may be generated that are unrealistically restrained by the Constraint.

Local Constraint

A Local Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the separate joint local coordinate systems.

The other degrees of freedom are unaffected.

The Local Constraint differs from the rigid-body types of Constraints in that there is *no coupling* between the rotations and the translations. The Local Constraint is the same as the Equal Constraint if all constrained joints have the same local coordinate system.

This Constraint can be used to:

- Model symmetry conditions with respect to a line or a point
- Model displacements constrained by mechanisms

The behavior of this Constraint is dependent upon the choice of the local coordinate systems of the constrained joints.

Joint Connectivity

Each Local Constraint connects a set of two or more joints together. If *any* of the selected degrees of freedom for a given Constraint are negative (i.e., opposite) only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space. If the joints do not share the same location in space, moments may be generated that are restrained by the Constraint. If this happens, the constraint forces printed in the output file may not be in equilibrium. These moments are necessary to enforce the desired symmetry of the displacements when the applied loads are not symmetric, or may represent the constraining action of a mechanism.

No Local Coordinate System

A Local Constraint does not have its own local coordinate system. The constraint equations are written in terms of constrained joint local coordinate systems, which may differ. The axes of these coordinate

systems are denoted 1, 2, and 3.

Selected Degrees of Freedom

For each Local Constraint you may specify a list, **ldofs**, of up to six degrees of freedom in the joint local coordinate systems that are to be constrained. The degrees of freedom are indicated as U1, U2, U3, R1, R2, and R3. A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., -U1.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts I and j) in a Local Constraint. These equations are expressed in terms of the translations u1, u2, and u3 and the rotationsr1, r2 and r3, all taken in joint local coordinate systems. The equations used depend upon the selected degrees of freedom and their signs. Some important cases are described next.

Symmetry About a Plane

For a structure that is symmetric about a plane, symmetric loading causes symmetric displacements as follows:

- Forces and displacements parallel to the plane of symmetry are equal
- Forces and displacements normal to the plane of symmetry are opposite
- Moments and rotations parallel to the plane of symmetry are opposite
- Moments and rotations normal to the plane of symmetry are equal

As an example, consider a structure that is symmetric with respect to a plane normal to the X axis and subjected to symmetric loading. A separate Equal Constraint must be defined for each pair of joints

that is symmetrically located with respect to the plane. The degrees of freedom that would be specified for these Constraints are: -UX, UY, UZ, RX, RY, and -RZ.

Any joints on the plane of symmetry should not be constrained, but instead have their UX, RY, and RZ degrees of freedom restrained.

Anti-symmetry About a Plane

For a structure that is symmetric about a plane, anti-symmetric loading causes anti-symmetric displacements. All degrees of freedom that are equal when symmetric are opposite when anti-symmetric, and all degrees of freedom that are opposite when symmetric are equal when anti-symmetric. Thus the specification of the anti-symmetric degrees of freedom simply uses the opposite signs from the symmetric case.

Consider the example above of a structure that is symmetric with respect to a plane normal to the X axis, but now subjected to anti-symmetric loading. A separate Equal Constraint must be defined for each pair

of joints that is symmetrically located with respect to the plane. The degrees of freedom that would be specified for these Constraints are: UX, -UY, -UZ, -RX, RY, and RZ. The signs of the constraint equations are corresponding changed from the symmetric case.

Axisymmetry

Axisymmetry is a type of symmetry about a line. It is best described in terms of a cylindrical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each

said to be axisymmetric about a line if they do not vary with angular position around the line, i.e., they are independent of the angular coordinate CA.

To enforce axisymmetry using the Local Constraint:

- Model any cylindrical sector of the structure using any axisymmetric mesh of joints and elements
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +CR, +CA, and +CZ, respectively

- For each axisymmetric set of joints (i.e., having the same coordinates CR and CZ, but different CA), define a Local Constraint using all six degrees of freedom: U1, U2, U3, R1, R2, and R3
- Restrain joints that lie on the line of symmetry so that, at most, only axial translations (U3) and rotations (R3) are permitted

Cyclic symmetry

Cyclic symmetry is another type of symmetry about a line. It is best described in terms of a cylindrical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements
are each said to be cyclically symmetric about a line if they vary with angular position in a repeated (periodic) fashion.

To enforce cyclic symmetry using the Local Constraint:

- Model any number of adjacent, representative, cylindrical sectors of the structure; denote the size of a single sector by the angle q
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +CR, +CA, and +CZ, respectively
- For each cyclically symmetric set of joints (i.e., having the same coordinates CR and CZ, but with coordinate CA differing by multiples of q, define a Local Constraint using all six degrees of freedom: U1, U2, U3, R1, R2, and R3.
- Restrain joints that lie on the line of symmetry so that, at most, only axial translations (U3) and rotations (R3) are permitted

For example, suppose a structure is composed of six identical 60^0 sectors, identically loaded. If two adjacent sectors were modeled, each Local Constraint would apply to a set of two joints, except that
three joints would be constrained on the symmetry planes at 0^0, 60^0, and $120^{0.}$

If a single sector is modeled, only joints on the symmetry planes need to be constrained.

Symmetry About a Point

Symmetry about a point is best described in terms of a spherical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be symmetric about a point if they do not vary with angular position about the point, i.e., they are independent of the angular coordinates SB and SA. Radial translation is the only displacement component that is permissible.

To enforce symmetry about a point using the Local Constraint:

- Model any spherical sector of the structure using any symmetric mesh of joints and elements
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +SB, +SA, and +SR, respectively
- For each symmetric set of joints (i.e., having the same coordinate SR, but different coordinates SB and SA), define a Local Constraint using only degree of freedom U3
- For all joints, restrain the degrees of freedom U1, U2, R1, R2, and R3
- Fully restrain any joints that lie at the point of symmetry

It is also possible to define a case for symmetry about a point that is similar to cyclic symmetry around a line, e.g., where each octant of the structure is identical.

Assign Joint Springs

Springs are added to selected joints.

1. Select the Joints to which you want to apply Springs.
2. On the Assign menu, click Joint and then Springs… in the sub-menu. This will display the Joint Springs dialog box.
3. In the Joint Springs dialog box:
   - Type in the Spring Stiffness values in the local directions for the three Translations and the three Rotations.
   - In the Options area select the appropriate option - Add to existing springs, Replace existing springs or Delete existing springs.
   - If required specify the upper half of the Coupled 6X6 Spring matrix, accessed by clicking the Advanced button.
4. Click OK.

Assign Joint Masses

Masses are added to selected joints.

1. Select the Joints to which you want to apply Masses.
2. On the Assign menu, click Joint and then Masses… in the sub-menu. This will display the Joint Masses dialog box.
3. In the Joint Masses dialog box:
   - Type in the Mass values in local Direction 1, Direction 2 and Direction 3.
   - Type in the Mass Moment of Inertia values in Rotation about 1, Rotation about 2 and Rotation about 3.
   - In the Options area select the appropriate option - Add to existing masses, Replace existing masses or Delete existing masses.
4. Click OK.

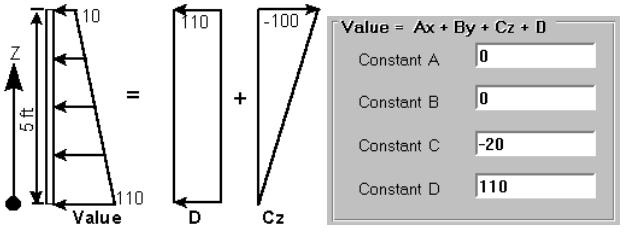
Assign Joint Restraints

Restraints are added to selected joints. 

1. Select the Joints to which you want to apply Restraints.
2. On the Assign menu, click Joint and then Restraints… in the sub-menu. This will display the Joint Restraints dialog box.
3. In the Joint Restraint dialog box:
   - Select from the Restraints in Local Directions, the joint restraint(s) desired, or
   - Select from Fast Restraints, the icon that represents the desired restraint condition.
4. Click OK.

Assign Joint Patterns

1. Select the Joints to which you want to assign the pattern.
2. On the Assign menu, click Joint Patterns… This will display the Pattern Data dialog box.
3. Click on the Pattern Name drop down box and choose a previously defined pattern name by clicking on it.
4. Type in the values in the edit boxes for Constants A, B, C and D, relevant to defining the Value of **Ax+By+Cz+D** (x, y and z are joint coordinates relative to the origin of the current coordinate system). It is this Value that will be assigned to the Joints. It can define the variation of Temperature for Frame Elements and variation of Pressure for Shell Elements. These Values are then subsequently multiplied by the Temperature values assigned from the Frame Static Loads or Pressure values assigned from the Shell Static Loads. For instance if you want to assign a pattern which represents the variation of soil pressure on a wall:



5. Options
   - To add computed Values from Constants to existing Values from previously assigned patterns, click on Add to existing values under Options.
   - To replace computed Values from Constants to existing Values from previously assigned patterns, click on Replace existing values under Options.
   - To delete previously assigned patterns, click on Delete existing values under Options.
6. Depending on the pattern you are trying to define select either::
   - Use all values if all values are desired
   - Zero Negative Values if negative variation is not desired.
   - Zero Positive Values if positive variation is not desired.
7. Click OK.

Note: Once you have assigned the pattern you will still need to assign a load to that pattern by selecting from the assign menu, Shell Static Loads... Pressure... .

Also See:

Define Joint Patterns

Assign Pressure Loads to Shells

Assign Local Axes

All Elements and Joints have default local axes. It is possible to change the definition of local axes by simply defining a rotation angle.

What do you want to do?

Assign/Change Local Axes for FRAME Elements

Assign/Change Local Axes for SHELL Elements

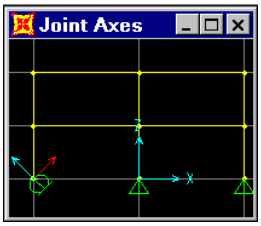
Assign/Change Local Axes for JOINTS

Assign Local Axes for Joints

1. Select one or more Joints to which you want to Assign the same Local Axes.
2. On the Assign menu, click Joint and then Local Axes… in the sub-menu.
3. In the Joint Local Axis display box:
   - Type in a value for rotation angles about Z, Y' and X" axes, in degrees (Y' and X" instead of Y and X are used as axes names to emphasize that once an angle about Z is specified the local X and Y will change as a result).
   - Check the Use default box if Global definition is to be used for the local axes.
   - Click the OK button

Note: The local axes arrows are only shown when the local axes for the JOINT are different than the Global.

EXAMPLE:



Assign Local Axes for Frame Elements

1. Select one or more Frame Elements to which you want to Assign the same Local Axes.
2. On the Assign menu, click Frame and then Local Axes… in the sub-menu.
3. In the Frame Local Axis display box:
   - Type in a value for Angle in Degrees. This is an angle by which local axis 2 of the Element will be rotated around local axis 1. Local axis 1 is along the length of the Element. By default local axis 2 is always in the 1-Z plane except if the Element is vertical and then it is parallel to the global X axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 1 is pointing towards you..
   - Check the Reverse start and end connectivity box if you want to flip the I and J ends of the members i.e. flip the local axis 1.

- Click the OK button

Note: By default global Z is taken as the 3D View Up Direction. You may change the UP direction from the Option pull down menu. If you change the UP direction, it will only affect the display of the model and does not affect the local axes definition in any way.

Assign Frame Releases

Releases are assigned to selected Frame Elements.

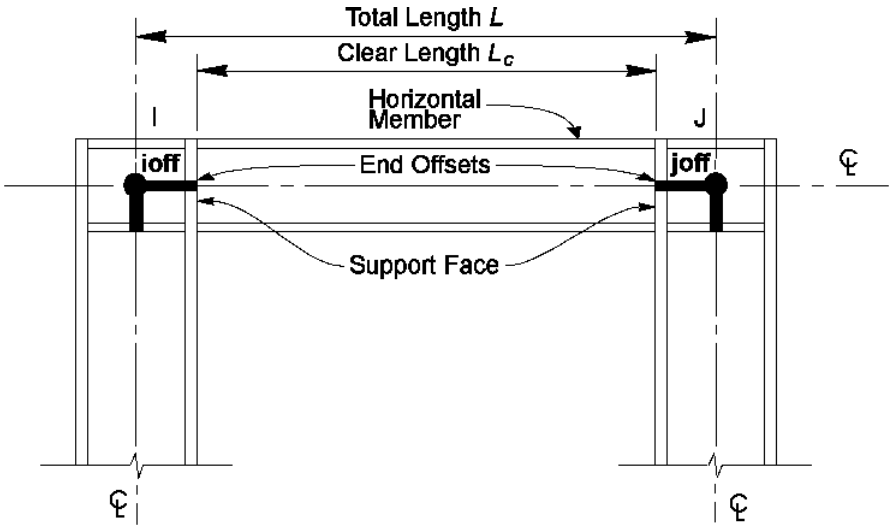
1. Select the FRAME Elements to which you want to apply the releases.
2. On the Assign menu, click Frame and then Releases… in the sub-menu. This will display the Frame Releases dialog box.
3. In the Frame Releases dialog box:
   - For each end of the FRAME Element check the type of release desired (Axial, Shear Force 2 (Major), Shear Force 3 (Minor), Torsion, Moment 22 (Minor) and Moment 33 (Major).
   - If no releases are desired then check the No Releases box.
4. Click OK.

Note: The releases specified always replace the existing releases.

Assign Frame End Offsets

Frame End Offsets are assigned to selected Frame Elements.

1. Select the FRAME Elements to which you want to apply the Frame End Offsets.
2. On the Assign menu, click Frame and then End Offsets … in the sub-menu. This will display the Frame End Offsets dialog box.
3. In the Frame End Offsets dialog box:
   - If you want to use User Specified End Offsets then check Define Lengths and type in the values for the Offsets at End I and End J.
   - If you want the program to calculate the End Offsets from the connectivity of the model then check Update lengths from current connectivity. The program will automatically calculate the End Offsets from the Depth (Major) and Width (Minor) specified for the Frame Element properties.
   - Specify a Rigid Zone Factor in the edit box. This is a factor used to define the percentage of the Zone specified through End Offsets to be taken as fully rigid. 0 means no rigid zone and 1 means that the entire zone is taken as rigid.
4. Click OK.



Assign Frame Output Segments

Frame Output Segments are assigned to selected Frame Elements.

1. Select the FRAME Elements to which you want to assign the Frame Output Segments.
2. On the Assign menu, click Frame and then Output Segments… in the sub-menu. This will display the Frame Output Segments dialog box.
3. In the Frame Output Segments dialog box:
   - In the Number of Segments edit box specify the number of segments at which the output is desired.
4. Click OK.

Note: The output segments are specified on the Clear Length of the Elements.

Tip: It is advisable to specify a different number of output segments for Columns than for Beams. (columns typically are designed using only the two end values of forces and moments whereas beams may be designed at the maximum values occurring somewhere in between the length because of the presence of point or distributed loads on the span of the beam).

Assign Frame Prestress

Assigning Frame Prestress loads is a way to model the prestress cable load on a frame member.

1. Select the FRAME Elements to which you want to assign a Prestress load.
2. On the Assign menu, click Frame and then Prestress… in the sub-menu. This will display the Frame Prestressing Patterns dialog box.
3. In the Frame Prestressing Patterns dialog box enter:
   - The Cable Tension in the text edit box
   - The Start, Middle and End cable locations.
   - Select from Add, Replace or Delete.
4. Click OK.

Note: You can assign more than one Prestress load to an individual frame member. Once you have assigned prestress loads to elements you will need to assign the frame prestress loads to a static load case.

Also see:

Assign Prestress to FRAME Static Load

Assign Frame Initial P-Delta Force

1. Select the FRAME Elements to which you want to assign a P-Delta load.
2. On the Assign menu, click Frame and then P-Delta … in the sub-menu. This will display the Frame Initial P-Delta Forces dialog box.
3. In the Frame Initial P-Delta Forces dialog box:
   - Select in which direction you want the load specified and enter the P-Delta axial force acting on the selected frame elements.
   - Select from Add, Replace or Delete.
4. Click OK.

Note: You can assign more than one P-Delta load to an individual frame member.

Assign Frame Lanes

1. Select the FRAME Elements you want assigned to an already defined lane.
2. On the Assign menu, click Frame and then Lane … in the sub-menu. This will display the Assign Lane dialog box.
3. In the Assign Lane dialog box:
   - Select the lane you want the frames assigned to from the drop down list box.
   - Enter an eccentricity for the lane relative to the frame member in the text edit box. The units for the eccentricity are the same as the current length units.
   - Press the Modify/Show Lane button to edit the order of the frame elements and their eccentricities.
4. Click OK.

Note: A frame member can be a part of more then one lane.

Define Lanes

Assign Frame Hinges (Pushover)

1. Select a frame member (or members). From the Assign menu select Frame and then select Hinges (Pushover). This displays the Frame Hinges (Pushover) dialog box. Note that if you have selected a single frame element, and that element already has hinges defined, those hinges will be displayed in the Frame Hinge Data area. If you have selected multiple frame elements, and each of those elements has exactly the same hinges defined, then those hinges will be displayed in the Frame Hinge Data area. If you have selected multiple frame elements, and each of those elements does not have exactly the same hinges defined, then the Frame Hinge Data area will be blank. (See the Tip below.)
2. Review, add to, or modify the data in the Frame Hinge Data area.
   - To add a hinge property to the selected frame element(s), select the Hinge Property from the drop down box, type in the location in the Relative Distance edit box and click the Add button.
   - To modify the location of a hinge on the selected frame element(s), select the hinge in the Hinge Property/Relative Location list box, edit the location in the Relative Distance edit box and click the Modify button.
   - To delete a hinge property from the selected frame element(s), select the hinge in the Hinge Property/Relative Location list box and click the Delete button.
3. Click the OK button to accept all of the changes made in the Frame Hinges (Pushover) dialog box. Clicking the Cancel button means that none of the changes will be accepted.

Tip: The Frame Hinges (Pushover) dialog box always works in a mode that replaces the existing hinges. When the OK button is clicked, the existing hinges on all selected elements will be replaced by those currently visible in the list box. Typically this does not cause any problem or confusion, **However, when multiple elements are selected, care must be taken.** If you select multiple frame elements, and those frame elements do not all have exactly the same hinges assigned to them, then no hinges will be listed in the list box. Thus, if you then click the OK button, with still no hinges defined in the list box, you will effectively be deleting all of the hinges on the selected frame elements. Similarly, if you add some hinges to the list box and then click the OK button, you will be replacing all of the existing hinges with those specified in the list box.

See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases

Static Pushover Case Data
Pushover Curve
Override Axis Labels And Range
Override Structural Behavior Type

Relative Distance

The relative distance is measured from the I-end of the frame element. When rigid end offsets are defined, the relative distance is based on the clear length of the element between the rigid end offsets.

Hinge Property

The hinge properties that are available in the Hinge Property drop down box on the Frame Hinges (Pushover) dialog box are the Automatic hinge properties, and the user defined hinge properties. Hinge properties that have been generated from the automatic hinge properties are not available in this drop down box.

Assign Prestress to Frame Static Load

Once the prestress has been assigned to the frame member the load from that prestress needs to be assigned to a Load Case for SAP2000 to be able to include it in the analysis.

1. Select the FRAME Elements for which you want the prestress load assigned to a defined Load Case.
2. On the Assign menu, click Frame Static Loads and then Prestress … in the sub-menu. This will display the Frame Prestress Loads dialog box.
   - Select the Load Case from the drop down list box.
   - Set a scale factor for the prestress load. This value is independent of scale factor for Load Combinations .
   - Select Add, Replace, Delete existing load.
3. Click OK

Also see:

Assign FRAME Prestress

Assign Local Axes for Shell Elements

1. Select one or more Shell Elements to which you want to Assign the same Local Axes.
2. On the Assign menu, click Shell and then Local Axes… in the sub-menu.
3. In the Shell Local Axis display box:
   - Type in a value for Angle in Degrees. This is an angle by which local axis 2 of the Element will be rotated around local axis 3. Local axis 3 is normal to the surface of the SHELL Element. By default local axis 2 is always in the 3-Z plane (and lies in the plane of the SHELL) except if the Element is horizontal and then it is parallel to the global X axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 3 is pointing towards you..
   - Check the Reverse direction of normal box if you want to flip the direction of local axis 3.
   - Click the OK button

Note: The positive direction of the local axis 3 is determined by the definition of the SHELL connectivity. See Sign Convention for more info.

Assign Local Axis For NLLinks

1. Select one or more NLLink Elements to which you want to Assign the same Local Axes.
2. On the Assign menu, click NLLink and then Local Axes… in the sub-menu.
3. In the NLLink Local Axis display box:
   - Type in a value for Angle in Degrees. This is an angle by which local axis 2 of the Element will be rotated around local axis 1. Local axis 1 is along the length of the Element. By default local axis 2 is always in the 1-Z plane except if the Element is vertical and then it is parallel to the global X axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 1 is pointing towards you..
   - Check the Reverse start and end connectivity box if you want to flip the I and J ends of the members i.e. flip the local axis 1.
   - Click the OK button

Note: By default global Z is taken as the 3D View Up Direction. You may change the UP direction from the Option pull down menu. If you change the UP direction, it will only affect the display of the model and does not affect the local axes definition in any way.

Assign Static Loads

Loads are applied to selected JOINTS, FRAME elements or SHELL elements.

What do you want to do?

Assign Loads or Displacements to JOINTS

Assign Gravity Loads to FRAMES

Assign Point and Uniform Loads to FRAMES

Assign Trapezoidal Loads to FRAMES

Assign Temperature Loads to FRAMES

Assign Gravity Loads to SHELLS

Assign Uniform Loads to SHELLS

Assign Pressure Loads to SHELLS

Assign Temperature Loads to SHELLS

Assign Loads or Displacements to Joints

1. Select one or more JOINTS to assign loads to.
2. On the Assign menu, click Joint Static Loads and then Forces… or Displacements… in the sub-menu.
3. In the Joint Forces or Ground Displacement dialog box:
   - Select the Load Case
   - Provide Forces and Moments or Translations and Rotations.
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the Joints selected.

Note: Displacements may only be applied to previously RESTRAINED Joints.

Tip: Click  to assign Joint forces.

Assign Gravity Loads to Frames

This loading method is a way to add the factored self weight of the members in as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the Static Load Cases .

1. Select one or more FRAMES to assign loads to.
2. On the Assign menu, click Frame Static Loads and then Gravity… in the sub-menu. This will display the Frame Gravity Loads dialog box.
3. In the Frame Gravity Loads dialog box:
   - Select the Load Case
   - Provide Gravity Multipliers
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Point and Uniform Loads to Frames

1. Select one or more FRAMES to assign loads to.
2. On the Assign menu, click Frame Static Loads and then Point and Uniform… in the sub-menu. This will display the Point and Uniform Span Loads dialog box.
3. In the Point and Uniform Span Loads dialog box:
   - Select Load Case
   - Choose Load Type and Direction.
     Note: The Gravity direction is taken to be in the negative Global Z direction. So to enter a load in the gravity direction you would enter a positive number if the direction is set to Gravity and a negative number if the direction is set to Global Z.
   - Provide Point Loads and Distances
   - Provide Uniform Load
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Tip: Click  to assign Point and Uniform loads to FRAMES.

Assign Trapezoidal Loads to Frames

1. Select one or more FRAMES to assign loads to.
2. On the Assign menu, click Frame Static Loads and then Trapezoidal… in the sub-menu. This will display the Trapezoidal Span Loads dialog box.
3. In the Trapezoidal Span Loads dialog box:
   - Select Load Case
   - Choose Load Type and Direction
   - Provide Loads and Distances
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Temperature Loads to Frames

1. Select one or more FRAMES to assign loads to.
2. On the Assign menu, click Frame Static Loads and then Temperature… in the sub-menu. This will display the Frame Temperature Loading dialog box.
3. In the Frame Temperature Loading dialog box:
   - Select Load Case
   - Select Type from Temperature, Temperature Gradient 2-2, Temperature Gradient 3-3
   - Select Temperature by Element or Joint Pattern
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Gravity Loads to Shells

This loading method is a way to add the factored self weight of the members in as a force in any of the global

directions. It is recommended that the actual self weight of the structure be included in the definition of the Static Load Cases .

1. Select one or more SHELLS to assign loads to.
2. On the Assign menu, click Shell Static Loads and then Gravity… in the sub-menu. This will display the Shell Gravity Loads dialog box.
3. In the Shell Gravity Loads dialog box:
   - Select the Load Case
   - Provide Gravity Multipliers
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Uniform Loads to Shells

1. Select one or more SHELLS to assign loads to.
2. On the Assign menu, click Shell Static Loads and then Uniform… in the sub-menu. This will display the Shell Uniform Loads dialog box.
3. In the Shell Uniform Loads dialog box:
   - Select Load Case
   - Provide Uniform Load value and Direction
     Note: The Gravity direction is taken to be in the negative Global Z direction. So to enter a load in the gravity direction you would enter a positive number if the direction is set to Gravity and a negative number if the direction is set to Global Z.
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Tip: Click  to assign Uniform loads to SHELLS.

Assign Pressure Loads to Shells

1. Select one or more SHELLS to assign loads to.
2. On the Assign menu, click Shell Static Loads and then Pressure… in the sub-menu. This will display the Shell Pressure Loads dialog box.
3. In the Shell Pressure Loads dialog box:
   - Select Load Case
   - Choose if pressure will be applied by Element or by Joint Pattern
   - If by Element, then specify pressure value
   - If by Joint Pattern , then choose the pattern and multiplier
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Temperature Loads to Shells

1. Select one or more SHELLS to assign loads to.
2. On the Assign menu, click Shell Static Loads and then Temperature… in the sub-menu. This will display the Shell Temperature Loading dialog box.
3. In the Shell Temperature Loading dialog box:
   - Select Load Case
   - Choose the type of Temperature Loading, i.e. Temperature or Gradient
   - Choose if temperature to be applied by Element or by Joint Pattern
   - If by Element, then specify temperature value
   - If by Joint Pattern, then choose pattern and multiplier
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Gravity Loads to NLLinks

This loading method is a way to add the factored self weight of the members in as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the Static Load Cases .

1. Select one or more NLLinks to assign loads to.
2. On the Assign menu, click NLLink Loads and then Gravity… in the sub-menu. This will display the NLLink Gravity Loads dialog box.
3. In the NLLink Gravity Loads dialog box:
   - Select the Load Case
   - Provide Gravity Multipliers
   - Select from Add, Replace, Delete
4. Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Group Names

Assigning Groups is a powerful tool in SAP2000. It helps in selecting elements, displaying and printing results as well as helping with design.

Group Name is assigned to selected Joints and Elements.

1. Select the Joints and Elements to which you want to assign a Group Name.
2. On the Assign menu, click Group Name… This will display the Assign Group dialog box.
3. In the Assign Group dialog box:
   - Click on the group name in the Groups list box. A default name ALL is always present in the list box.
   - It is possible to define a new group at this stage by typing in the new name in the Groups edit box and then clicking on the Add new Group Name button.
   - To Change or Delete a group name, click on the Group name in the list box to select it and then click on the Change Group Name or Delete Group Name button.
4. Click OK.

Tip: To add more Joints and/or Elements to an already assigned Group, first select by group name, select more Joints and/or Elements and then assign them to the Group. Group assignment always replaces the existing Elements in that group.

ANALYZE MENU

Analyzing a Model

1 Click on the Analyze menu and then Set Options... This will display the Analysis Options dialog box.

2 Click to check the appropriate degrees of freedom (U1, U2, U3, R1, R2, R3) to be available for either a 2-D or 3-D model in the Available DOF's area.

3 Alternatively, the available degrees of freedom may be automatically checked by clicking on the four Fast DOF's option i.e. Space Frame, Plane Frame, Plane Grid and Space Truss.

4 If a Dynamic Analysis is required, check the Dynamic Analysis box. Checking this box will display the Set Dynamic Parameters button. Clicking on this button will display the Dynamic Analysis Parameters dialog box.

In the Dynamic Analysis Parameters box:

- Type in the number of Modes.
- Select the Type of Analysis, Eigenvectors or Ritz Vectors.
- If Eigenvector Analysis is selected, then Eigenvalue parameters may be defined.
  - The default values shown in the edit boxes are adequate for a majority of cases.
  - Check Include Residual-Mass Modes if desired. This option is only available for Eigenvector analysis. With this option the number of Eigenvector modes recovered will be 3 less then the number specified and then up to 3 non-zero Residual-Mass Modes will be reported.

  Note: A minimum of 4 modes need to be requested if the Include Residual-Mass Modes option is checked. This is due to the last 3 modes being automatically reserved for the Residual-Mass Modes.
- Choosing Ritz Vector Analysis makes the Starting Ritz Vector dialog box available.
- Select Starting Ritz Load Vectors from the List of Loads. Use the ADD and REMOVE buttons to modify the list.
- Click OK

5 If P-Delta analysis is required, then check the Include P-Delta box. Checking this box will allow you to select the Set P-Delta Parameters button. Clicking on this button will display the P-Delta Parameters dialog box.

In the P-Delta Parameters dialog box:

- Enter maximum number of iterations or accept the default. A reasonable number is usually 5 or less.
- Enter displacement tolerance or accept the default. Failure to converge may result if too small a displacement is specified. The default tolerance is 0.001
- Enter force tolerance or accept the default.
- Define the load combination from existing Load Cases using appropriate scale factors. Use the Add, Modify and Delete buttons to set up this combination.
- Click OK.

6 If you want to have any analysis results saved to an output file, then check the Generate Output box . Checking this box will allow you to select the Select Output Options button. Click on this button to display the Select Output Results dialog box.

In the Select Output Results dialog box:

- Check the type of analysis results that you are interested in.
- Checking this box will allow you to select the Select/Show Loads button. You can press this button and select which load cases and load combinations to output information on for that category. You can not select the range of elements to include in the output.
- Click OK.

7 Specify the memory in Kilobytes (KB). The default memory size is 2000 kilobytes. For larger problems more

memory may be necessary.

8 Clicking the OK button will save the analysis parameters and exit the form.

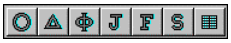
9 From the Analyze menu Click Run or Run minimized. Run minimized will perform the execution in the background i.e. will allow you to minimize SAP2000 while the analysis is being carried out.

Note: The program will seek the number of lowest frequency (longest period) Modes. Only the number of modes selected are available for any subsequent response-spectrum or time-history analysis processing. The number of modes is limited to the mass degrees of freedom.

Caution: P-Delta analysis in SAP2000 is an iterative analysis and the iterations do not include the change in geometry.

DISPLAY MENU

Display Options

Clicking on any of these buttons  from the Toolbar allows you to Display your selection with varied options.

Select various Display options by clicking as follows:

To Display	Do this
Undeformed geometry	Click on 
Isometric view (toggle)	Click on 
2-D View (XY, XZ or YZ plane)	Click on any one of the buttons  to display an elevation view in that specific plane.
3-D View	Click on 
Deformed Shape	Click on  or Display menu and Show Deformed Shape
Joint Reaction Forces	Click on  or Display menu and Show Element Forces/Stresses…Joints . Right click will show the details.
Frame Force Diagrams	Click on  or Display and Show Element Forces/Stresses…Frames. Right click will show the details.
Shell Force/Stress Contours	Click on  or Display menu and Show Element Forces/Stresses…Shells. Right click will show the details.
Joint, Frame or Shell Loads	Click on Display menu and Show Loads.
Patterns	Click on Display menu and Show Patterns.
Tabular Input	Click on Display menu and Show Input Tables.
Mode Shapes	Click on  or Display menu and Show Mode Shapes .
Tabular Output	Click on  or Display menu and Show Output Tables .
Response Spectrum	Click on Display menu and Show Response Spectrum Curves .
Time History	Click on Display menu and Show Time History Traces .
Shrunken Elements (toggle)	Click on  or click on  and select shrink.
Selective Elements	Click on  or View menu and Set Elements .

Graphical Output

What do you want to do?

- Display Undeformed Geometry
- Display Static Deformed Shape
- Display Mode Shape
- Display Member Force or Stress Diagram
- View Time History Results
- View Generated Response Spectra Curves
- Animate Deformed Shape
- View Real Time Animation for Time History Results
- Print Selected Graphical Output to a Printer or a File

Displaying Undeformed Geometry

On the Display menu, click Show Undeformed Shape, or click  from the Toolbar.

Display Static Loads

To display graphically the loads on the structure:

1. On the Display menu, click Show Loads.., select the element type you are interested in. This will display the Show *member type* Loads dialog box.
2. In the dialog box:
   - Select the Load Case for the loads you want to see.
   - Select an item from the Load Type area.
   - If provided select to have the Load Values Shown.
3. Click OK to view the loads in the active window.

Display Joint Patterns

To display Joint Patterns :

1. On the Display menu, click Show Joint Patterns
2. In the Select Pattern dialog box, select the pattern you are interested in.
3. Click OK to view the load pattern in the active window.

Display Bridge Lanes

To graphically display lanes:

1. On the Display menu, click Show Lanes .
2. In the Show Lane dialog box,
   - Select the Lane you are interested in.
   - Check Show Eccentricity if you want to see the lane eccentricities.
3. Click OK to view the lane in the active window.

Also See:

Display Input in Tabular Format

To view your input data as it appears in an S2K file or by printing Input Tables from the file menu:

1. On the Display menu, click Show Input Tables… Geometry Data or Loading Data.
2. In the Display Geometry/Loading Options dialog box
   - Select the Type of input information you are interested in.
3. Click OK to view the Input Table.
   - Under the File menu, in the form that appears, there are printing options.
   - Close the form by clicking on the X in the top right hand corner of the form.

Display Static Deformed Shape

1. On the Display menu, click Show Deformed Shape..., or press  on the Toolbar. This will display the Deformed Shape dialog box.
2. In the Deformed Shape dialog box:
   - Select the Load Case or Combination Name from the Load drop down list.
   - Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the displacement vector components. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto option.
   - Checking the Wire Shadow box will also display the undeformed shape as a reference for comparison with the deformed shape.
   - Checking the Cubic Curve box will display the deformed elements with a cubic curve fit.
   - After selecting the options, click OK and the screen display will be updated.
3. Clicking the **Start Animation** button on the Status Line will animate the deformed shape of the model. Animation speed is controlled by + and - buttons next to the animation button.

To view the displacement components for a single joint do the following:

- Right click once on the joint. The selected joint is highlighted and values are displayed in a floating window called the Joint Displacements window.
- Clicking on any other joint updates the display.
- Clicking anywhere else closes the floating window.

Caution: The deformed shape display for dynamic Load Cases is based on the absolute values of maximum displacements.

Note: Joint displacements can not be displayed during animation. To stop the animation, press the Stop Animation button.

Display Mode Shape

1. On the Display menu, click Show Mode Shape..., or click  on the Toolbar. This will display the Mode Shape dialog box.
2. In the Mode Shape dialog box:
   - Select the mode number by either typing it in the edit box or clicking on the spin button.
   - Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the vector components. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto option.
   - Checking the Wire Shadow box will also display the undeformed shape as a reference for comparison with the mode shape.
   - Checking the Cubic Curve box will display the deformed elements with cubic curve fit.
   - After selecting the options, click OK and the screen display will be updated.
3. Clicking the **Start Animation** button on the Status Line will animate the deformed shape of the model. Animation speed is controlled by a sliding button next to the animation button. Clicking on + and - buttons will change the display of the mode shape number by +1 or -1.

To view the normalized modal components for a single joint do the following:

- Right click once on the joint. The selected joint is highlighted and values are displayed in a floating

window called the Joint Modal Components window.
- Clicking on any other joint updates the display.
- Clicking anywhere else closes the floating window.

Note: Modal components can not be displayed during animation. To stop the animation, press the Stop Animation button.

Animate Deformed Shape

1 Plot the desired deformed shape on the Screen.

2 Clicking the **Start Animation** button on the Status Line will animate the deformed shape of the model. Animation speed is controlled by a sliding button next to the animation button. Clicking on + and - buttons will change the Analysis Case for the display by +1 or -1.

Display Member Force or Stress Diagram

1 On the Display menu, click Show Element Force/Stresses…, or click the  (Frame),  (Shell) or  (Joints) buttons on the Toolbar. This will display the Member Force Diagram dialog box.

2 In the Member Force Diagram dialog box.

If **Frames** is selected

- Select the Load Case or Combination Name from the Load drop down list.
- Select the desired member force component, that is, Axial Force, Shear 2, Shear 3, Torsion, Moment 2-2, or Moment 3-3 .
- Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the Member Force diagrams. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto.
- Check on Fill Diagram to see the diagrams with colored fill.
- Check on Show Values on Diagram to print numerical values on the diagram.

If **Shells** is selected

- Select the Load Case or Combination Name from the Load drop down list.
- Select output Type as Resultants or Stresses.
- Select the desired member force Component (If Resultants F11, F22, F12, FMAX, FMIN, FVM, M11, M22, M12, MMAX, MMIN, V13, V23, VMAX and if Stresses S11, S22, S12, SMAX, SMIN, SVM, S13, S23, SMAXV).
- Select the contour range.
- Select if the Stress should be averaged at joints.
- Select if the Stress contours to be displayed on deformed shape.

If **Joints** is selected

- Select the Load Case or Combination Name from the Load drop down list.
- Select either Reactions or Spring Forces.

3 Click OK. The screen display will be updated with the options selected.

Hints: To display the details of a force diagram for a single member, right click on the member. This will open a floating window which displays the variation of the selected component over the length of the element. If more than one element exists within the tolerance of the cursor, a list box of the elements is displayed. You can then select the desired element.

Move the pointer along the length of the member to view the Distance and Value at that point. Alternatively, click on the diagram in the floating window at the desired location. The magnitude of force component and associated location are displayed. The element ID number is displayed in this window.

Click anywhere in the main display window to close the Member Force Diagram window.

Display Energy Diagrams

The Energy Diagrams display is used show the percentage of virtual work of an element relative to the rest of the structural members. It can be used to reduce structural deflection by indicating which elements have the highest percentage of energy and thus will most effect the deflection if their stiffness is modified.

1 On the Display menu, click Show Energy Diagram... . This will display the Energy Diagram dialog box.

2 In the Energy Diagram dialog box:

- Select the load case or combination from which the element forces will be used.
- Select the load case or combination from which the element deflections will be used. This is usually a load case that has forces at the location and in the direction of the displacement that is of interest.

3 Select Show values if you want the element's percent relative energy shown as well as the element color-coding.

4 Click the OK button to display the percentage of energy the element is experiencing relative to the other elements in the structure.

Example: If the user is interested in reducing the roof story drift due to a wind load. A load combination that includes the wind load case is selected for the Force. Then for Displacement, a load case is used which has lateral forces only at the roof level even though the real displacement is caused by a distributed wind load. The structure is then displayed with the percent of virtual work energy that the element is experiencing from the stresses induced by the Force load combo and the displacement from the Displacement load case. This will essentially mean that the elements with the highest ration will effect the deflection the most if their stiffness is modified.

View Time History Results

Displaying of the time history results is a four-step process.

Step 1: Choose a Time History Case for which you want to see the traces.

Step 2: Select Joints or Elements for which you want to display Time History traces.

Step 3: Define Output Functions for Selected Joints or Elements and add to Display List.

Step 4: Plot Time History traces added to Display List.

What do you want to do?

 Display Input Time History Functions

 Display Joint Output Time History Trace

 Display Frame Element Force Output Time History Trace

 Display Shell Element Stress Output Time History Trace

 Display Structural Energy Time History Trace

 Display Base Function Time History Trace

Display Input Time History Functions

To display Input Time History Functions do the following:

1 On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.

2 To Add the Input Function:

- Click the Define Functions button. This will display the Time History Function dialog box.
- Click on Add Input Functions from the Add drop down list box.
- In the Time History Input Function dialog box:
  - Select the Input Function from the Function Name drop down list box.
  - Click OK.
- The Function Name will be Added in to the Functions list box.

3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

 Plot the variations of Function against Time (F(t) vs. t)

 Change Line Type, Line Color and Scale Factor for the Plots

 Zoom in on a smaller area of the plot to see the details

 Print Time History Plots or Tables

Display Joint Output Time History Trace

To display Joint Output Time History Trace by selecting the joint first, do the following:

1. Select the Joints for which you want to Display Time History traces.
2. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box with the Joint Labels listed as Functions.
3. To Modify an existing Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - In the Time History Function dialog box:
     - Click on a Function Name.
     - Click the Modify/Show TH Function button.
     - In the Time History Joint Function dialog box
       - Accept or modify the function name in the Function Name edit box.
       - Select the Vector Type (Displacement, Velocity, Acceleration, Absolute Acceleration, Reaction or Spring).
       - Select the Vector Direction (X Translation, Y Translation, Z Translation, X Rotation, Y Rotation, Z Rotation).
       - To recover the time history for a particular mode:
         - Select the Include One button.
         - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
       - Click OK.
4. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box

To display Joint Output Time History Trace directly from the Time History Display Definition dialog box, do the following:

1. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.
2. To Add a joint Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Joint Disps/Forces from the Add drop down list box.
   - In the Time History Joint Function dialog box:
     - Enter the JointID.
     - A function name will automatically appear in the Function Name text box.
     - Select the Vector Type (Displacement, Velocity, Acceleration, Absolute Acceleration, Reaction or Spring).
     - Select the Vector Direction (X Translation, Y Translation, Z Translation, X Rotation, Y Rotation, Z Rotation).
     - To recover the time history for a particular mode:
       - Select the Include One button.
       - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

 Plot the variation of Function against Time (F(t) vs. t)

 Plot the variation of Function against Function(F(t) vs. F(t))

 Change Line Type, Line Color and Scale Factor for the Plots

 Zoom in on a smaller area of the plot to see the details

 Print Time History Plots or Tables

Display Frame Element Force Output Time History Trace

To display Frame Output Time History Trace by selecting the frame first, do the following:

1. Select the Frames for which you want to Display Time History traces.
2. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box with the Frame Labels listed as Functions.
3. To Modify an existing Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - In the Time History Function dialog box:
     - Click on a Function Name.
     - Click the Modify/Show TH Function button.
     - In the Time History Frame Function dialog box
       - Accept or modify the function name in the Function Name edit box.
       - Select the Component (Axial Force, Shear 2, Shear 3, Torsion, Moment 2-2, Moment 3-3).
       - Select the Location Station for the Frame Element.
       - To recover the time history for a particular mode:
         - Select the Include One button.
         - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
       - Click OK.
5. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box

To display Frame Output Time History Trace directly from the Time History Display Definition dialog box, do the following:

1. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.
2. To Add a frame Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Frame Element Forces from the Add drop down list box.
   - In the Time History Frame Function dialog box:
     - Enter the ElementID.
     - A function name will automatically appear in the Function Name text box.
     - Select the Component (Axial Force, Shear 2-2, Shear 3-3, Torsion, Moment 2-2, Moment 3-3).
     - Select the Location Station for the Frame Element.
     - To recover the time history for a particular mode:
       - Select the Include One button.
       - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

 Plot the variation of Function against Time (F(t) vs. t)

 Plot the variation of Function against Function(F(t) vs. F(t))

 Change Line Type, Line Color and Scale Factor for the Plots

 Zoom in on a smaller area of the plot to see the details

 Print Time History Plots or Tables

Display Shell Element Stress Output Time History Trace

To display Shell Output Time History Trace by selecting the shell first, do the following:

1. Select the Shells for which you want to Display Time History traces.
2. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box with the Shell Labels listed as Functions.
3. To Modify an existing Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - In the Time History Function dialog box:
     - Click on a Function Name.
     - Click the Modify/Show TH Function button.
     - In the Time History Shell Function dialog box
       - Accept or modify the function name in the Function Name edit box.
       - Select output Type as Resultants or Stresses (top or bottom).
       - Select the Component (If Resultants; F11, F22, F12,FMAX,FMAIN,FVM, M11, M22, M12,MMAX,MMIN, V13, V23,VMAX and if Stresses; S11, S22, S12, SMAX, SMIN, SVM, S13, S23, SMAX).
       - Select the Joint of the Shell at which the Resultants or Stresses are required from the Joint drop down list.
       - To recover the time history for a particular mode:
         - Select the Include One button.
         - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
       - Click OK.
6. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box

To display Shell Output Time History Trace directly from the Time History Display Definition dialog box, do the following:

1. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.
2. To Add a shell Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Shell Element Forces from the Add drop down list box.
   - In the Time History Shell Function dialog box:
     - Enter the ElementID.
     - A function name will automatically appear in the Function Name text box.
     - Select output Type as Resultants or Stresses (top or bottom).
     - Select the Component (If Resultants; F11, F22, F12,FMAX,FMAIN,FVM, M11, M22, M12,MMAX,MMIN, V13, V23,VMAX and if Stresses; S11, S22, S12, SMAX, SMIN, SVM, S13, S23, SMAX).
     - Select the Joint of the Shell at which the Resultants or Stresses are required from the Joint drop down list.
     - To recover the time history for a particular mode:
       - Select the Include One button.
       - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

 Plot the variation of Function against Time (F(t) vs. t)

 Plot the variation of Function against Function(F(t) vs. F(t))

 Change Line Type, Line Color and Scale Factor for the Plots

Zoom in on a smaller area of the plot to see the details

Print Time History Plots or Tables

Display Structural Energy Time History Trace

1. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.
2. To Add the Energy Functions:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Energy Functions from the Add drop down list box.
   - In the Energy Functions dialog box:
     - Select the desired Energy Functions: Input Energy, Kinetic Energy, Potential Energy, MDamp Energy, NDamp Energy, NLLink , Energy Error .
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

Plot the variation of Function against Time (F(t) vs. t)

Plot the variation of Function against Function(F(t) vs. F(t))

Change Line Type, Line Color and Scale Factor for the Plots

Zoom in on a smaller area of the plot to see the details

Print Time History Plots or Tables

Display Base Function Time History Trace

1. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.
2. To Add the Base Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Base Functions from the Add drop down list box.
   - In the Base Functions dialog box:
     - Select the desired Base Functions: Base Shear X, Base Shear Y, Base Shear Z, Base Moment X, Base Moment Y, Base Moment Z.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

Plot the variation of Function against Time (F(t) vs. t)

Plot the variation of Function against Function(F(t) vs. F(t))

Change Line Type, Line Color and Scale Factor for the Plots

Zoom in on a smaller area of the plot to see the details

Print Time History Plots or Tables

Display Group Summation Forces Time History

To display Group Summation Forces Time History Trace from the Time History Display Definition dialog box, do the following:

1. On the Display menu, click Show Time History Traces… This will display the Time History Display Definition dialog box.
2. To Add a Group Summation Forces Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Group Summation Forces from the Add drop down list box.
   - In the Time History Group Function dialog box:
     - Select the Group name from the Group drop down list box.
     - A function name will automatically appear in the Function Name text box.
     - Select the Component Force or Moment in the three global directions.
     - To recover the time history for a particular mode:
       - Select the Include One button.
       - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3. Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

 Plot the variation of Function against Time (F(t) vs. t)

 Plot the variation of Function against Function(F(t) vs. F(t))

 Change Line Type, Line Color and Scale Factor for the Plots

 Zoom in on a smaller area of the plot to see the details

 Print Time History Plots or Tables

F(t) vs. t

To plot the variations of one function against time:

1. In the Time History Display Definition dialog box:
   - Choose the F(t) vs. t Tab.
   - From the drop down list box, select the Time History Case for which you want to display Input /Output Time History Traces.
   - Click on the Function name to be plotted in the List of Functions box.
   - Click on the Add button. The function name will be added to the Plot Functions list box.
   - Repeat to include other function names (if any) for simultaneous display of several functions.
   - Use the Remove button to remove a function from the Plot Functions list box.
   - Use the Show button to display the function's attributes (e.g. Vector Type, Direction etc.)
   - Enter the time range in the From and To edit boxes. By default, the full time range is selected.
   - Enter the axis labels in the Horizontal and Vertical edit boxes in the Axis Labels area.
   - Removing the check mark from the Grid Overlay box will disable the generation of a grid through the major divisions of the axes.
2. Click on Display. This will open a floating window labeled Time History Traces in which functions are graphically displayed. A small Legend window displays the maximum and minimum numerical values for the functions and another small window just below it displays the coordinates of the pointer.

 Plot the variations of Function against Time (F(t) vs. F(t))

 Change Line Type, Line Color and Scale Factor for the Plots

 Zoom in on a smaller area of the plot to see the details

 Print Time History Plots or Tables

F(t) vs. F(t)

To plot the variations of one function against another function:

1 In the Time History Display Definition dialog box:

- Choose the F(t) vs. F(t) Tab
- From the drop down list box, select the Time History Case for which you want to display Input /Output Time History Traces.
- Select the horizontal time function from the drop down list in the Choose Functions area.
- Select the vertical time function from the drop down list in the Choose Functions area.
- Enter the time range in the From and To boxes. By default, the full time range is selected.
- Enter the axis labels in the Horizontal and Vertical edit boxes in the in the Axis Labels area.
- Removing the check mark from the Grid Overlay box will disable the generation of a grid through the major divisions of the axes.

4 Click on Display. This will open a floating window labeled Time History Traces in which functions are graphically displayed. A small Legend window displays the Function names for the Horizontal and Vertical axes together with their attributes and another small window just below it displays the coordinates of the pointer.

 Plot the variations of Function against Time (F(t) vs. t)

 Change Line Type, Line Color and Scale Factor for the Plots

 Zoom in on a smaller area of the plot to see the details

 Print Time History Plots or Tables

Scale Factor, Line Type and Color

To change the **Line Type**

- Click on the function name in the Plot Functions list box.
- Select the line type to be Solid, Dashed or Dotted from the Line Options area.

To change the **Line Color**

- Click the Color button in the Line Color area. This will display the Color dialog box.
- Click on the desired color in the palette.
- Click OK.

To change the **Scale Factor**

The Scale Factors default to 1.

- Check the Horizontal/Vertical direction for which you want to change the Scale Factor.
  - Press the Define button that appears.
  - Enter in the Horizontal/Vertical Scale for the Maximum and Minimum Scale Factor in the appropriate edit box.
  - Click OK

Note: Maximum and Minimum Scale Factors of zero mean that the default scale factors are used.

Zoom into the Plot

To zoom on a specific part of the plot:

- Point to a corner of the region, hold down the mouse button and drag to define the rectangular zoom region, and release the mouse button.
- Clicking anywhere in the main display area brings back the full view.

Print Time History or Response Spectrum Plots or Tables

To print the **Plots** display do the following:

- From the Print menu, choose 'Print Graphics'.

To print the **Tables** (results in a numerically tabulated form, in an ASCII format,) do the following:

- From the Print menu, choose 'Print Tables'.

To save the **Tables** in a File, do the following:

- From the Print menu, choose 'Save Tables as File...'. This will display the Save File As dialog box.
- In the Save File As dialog box:
  - Select the drive and the directory in which the file is to reside.
  - Enter a new filename or overwrite an existing file.
  - Click the OK button.

Clicking anywhere else closes the floating window.

View Generated Response Spectra Curves

1. Select the Joint for which you want to Display Response Spectrum Curve.
2. On the Display menu, click Show Response Spectrum Curves... This will display the Response Spectrum Generation dialog box. The dialog box has five Tabs labeled **Define**, **Axes**, **Options**, **Frequency** and **Damping**. Clicking on any Tab highlights it.

 The Define Tab

 The Axes Tab

 The Options Tab

 The Frequency Tab

 The Damping Tab

After making the necessary changes in all the Tabs in the Response Spectrum Generation dialog box:

3. Click on Display to open a floating window labeled Response Spectrum Curves in which the response spectra are graphically displayed. A small Legend window displays the Joint number Direction, Scale Factor, Widening percentage and Damping Ratios color coded to correspond to the Spectra Curves. Another small window just below it displays the coordinates of the pointer.
4. Click on Done to save the changes made in all the Tabs and exit the Response Spectrum Generation dialog box.

 Zoom in on a smaller area of the plot to see the details

 Change Line Type, Line Color and Scale Factor for the Plots

 Print Response Spectrum Plots or Tables

Spectrum Generation Define Tab

In the **Define Tab**:

- From the drop down list select the Time History Case for which you want to generate a Response Spectrum.
- Click on a Joint in the **Choose a Joint** list and then click on the appropriate direction X, Y or Z from the **Vector Direction** option adjacent to it to define the direction for which you want to generate a spectrum. By default the X (global X-direction) is checked.

Spectrum Generation Axes Tab

In the **Axes Tab**:

- Select the **Abscissa** to show as either **Frequency** or **Period.**
- Select the **Ordinate** to be one of the five choices i.e. **SD** (Spectral Displacement), **SV** (Spectral Velocity), **PSV** (Pseudo Spectral Velocity), **SA** (Spectral Acceleration) and **PSA** (Pseudo Spectral Acceleration).

Spectrum Generation Options Tab

In the **Options Tab:**

- Set the **Abscissa** scale. By default **Log** is checked.
- Changing the percentage of **Spectrum Widening** will change the peaks of response spectra curves. The default is set to zero.
- Set the **Ordinate** scale. By default **Arithmetic** is checked.
- Changing the ordinate scale factor will scale the spectral values. The default is 1.0.
- Removing the check mark from the Grid Overlay box will disable the generation of a grid through the major divisions of the axes.

Spectrum Generation Frequency Tab

In the **Frequency Tab:**

- Removing the check mark from of the Default box will exclude the built-in set of frequencies in the generation of response spectrum.
- Removing the check mark from of the Structural box will disable merging of the structural frequencies into the set of frequencies used in the generation of response spectrum.
- To enter user-defined frequencies do the following:
  - Check the User box in the Include Frequencies area.
  - Enter the value in the User Frequencies edit box.
  - Click the **Add Value** button.
  - Repeat to include all the frequencies.
  - Use the **Change Value** and **Delete Value** buttons to edit or delete the values included.

Spectrum Generation Damping Tab

In the **Damping Tab:**

- Use this option to add a new damping value or edit the default damping values.
- To Add a new damping value do the following:
  - Type in a new value in the edit box.
  - Click the **Add Value** button.
- To change one of the default damping values do the following:
  - Click on the damping value in the Damping values list box.
  - Change the value in the edit box.
  - Click the **Change Value** button.
- To remove one of the values from the set do the following:
  - Click on the damping value.
  - Click the **Delete Value** button.

Display Group Joint Force Sums

The option is an easy way to find the sum of forces and moments on a Group of joints.

1. On the Display menu, click Show Group Joint Force Sums. This will display the Select Groups dialog box.
2. Select the Group or Groups for which you want the sum.
3. Click OK to view in tabular form the sum of the shears and moments in the group.
4. You can print the table from the File menu.
5. When finished, close the table by clicking on the close button at the top right of the form.

Note: The group for the Group Sum must be selected carefully. The group must consist of a set of joints and

the elements directly connected to only one side of them. For example if the base shear for a structure is required, the group should consist of the joints at the base of the structure and the frame/shell elements above them that are directly connected to them.

Display Joint Influence Lines

The influence lines can be displayed for any joint displacement, reaction or force component due to a unit load on a defined Bridge Lane in the structure.

1. On the Display menu, click Show Influence Lines …Joints. This will display the Show Joints Influence Line dialog box.
2. Select the Lane for which you want to see influence lines.
3. Select the Joint for which the results are reported.
4. Select the Vector Type as Displacement, Spring Force or Reaction. There may be no influence lines for some of these Vector Types, depending on the structural configuration.
5. Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the diagrams. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto.
6. Pressing the Table button will show the influence line data points in tabular form. The table includes the Lane name, Frame name, Location relative to the starting point of the lane, Location relative to the i end of the frame member, and the influence line value.
7. Click OK to view the Influence line or Cancel to close the dialog box without viewing the influence lines.

Display Frame Influence Lines

The influence lines can be displayed for any Frame moment, shear, torsion or axial load component due to a unit load on a defined Bridge Lane in the structure.

1. On the Display menu, click Show Influence Lines …Frames. This will display the Show Frames Influence Line dialog box.
2. Select the Lane for which you want to see influence lines.
3. Select the Frame for which the results are reported.
4. Select the Component as Moment, Shear, Torsion or Axial Load. There may be no influence lines for some of these Component, depending on the structural configuration.
5. Select the Station (Output Segment) for which you want the influence lines.
6. Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the diagrams. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto.
7. Pressing the Table button will show the influence line data points in tabular form. The table includes the Lane name, Frame name, Location relative to the starting point of the lane, Location relative to the i end of the frame member, and the influence line value.
8. Click OK to view the Influence line or Cancel to close the dialog box without viewing the influence lines.

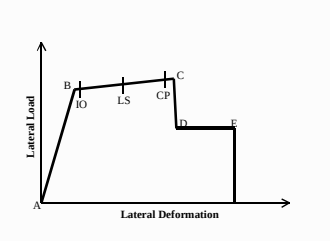
Display Joint or Member Text Output on Screen

1. On the Display menu, click Set Output Table Mode…, or click  on the Toolbar. This will display the Select Output dialog box.
2. In the Select Output dialog box:
   - Select the Load Cases and/or the Load Combinations labels for which the output is to be displayed in a tabulated form. Loads may be selected by clicking on the Load labels in the list box. For selecting a range of loads, click and drag, or for multiple loads hold down the Ctrl Key and click on different load labels.
   - Click OK to close the Select Output dialog box.
   - To display the member force, displacement or reaction output table, right click on the element or joint of interest. This will open a floating window which displays the output table. If more than one element or joint exists within the tolerance of the cursor, a list box of the elements and joints is displayed. You can then click on the desired element or joint.
   - Click anywhere in the main display window to close the output table window.

Pushover Curve

1. After the pushover analysis has been run, select Show Static Pushover Curve from the Display menu. This displays the Pushover Curve dialog box.
2. Select the Static Pushover Case to be displayed from the drop-down box in the upper right-hand corner of the dialog box.
3. In the Plot Type area select either the Base Shear vs Control Displacement option or the Capacity Spectrum option. The capacity spectrum option plots the pushover curve in the ADRS format.
4. The default color for the capacity curve is green, regardless of the format in which it is plotted. Click the color box in the Plot Type area to change the color of the capacity curve.
5. The demand spectrum area is only active when the Capacity Spectrum option is chosen in the Plot Type area.
   - The shape of the demand spectrum with 5% damping is controlled by the values input in the Seismic Coefficient Ca, and Seismic Coefficient Cv edit boxes. Refer to ATC-40, Chapter 4, for a discussion of, and appropriate values for, Ca and Cv. The default value for both Ca and Cv is 0.4. Demand spectra for other damping levels are created from the 5% damped spectrum using the spectral reduction factors described in Section 8.2.2.1.1 of ATC-40.
   - Checking the Show Family of Demand Spectra box in the Demand Spectrum area overlays a family of demand spectra on the capacity curve in the ADRS format.
   - The default color for the family of demand spectra curves is red. Click the color box adjacent to the Show Family of Demand Spectra check box label to change the color of the demand spectra.
   - The family of curves can include up to four demand spectra curves, each with a different effective damping ratio, βeff. By default, the program plots curves with βeff = 0.05, 0.1, 0.15 and 0.2. The damping ratios for any of the four curves can be changed by editing the value in one of the four Damping Ratios, βeff edit boxes. The values input into the βeff edit boxes must be between 0 and 1, inclusively. A value of 0, or a blank edit box, means to omit that demand spectrum curve.
   - Check the Show Single Demand Spectrum (Variable Damping) check box to display the demand spectrum as a single curve.
   - The default color for the single modified demand spectrum curve is yellow. Click the color box adjacent to the Show Single Demand Spectrum (Variable Damping) check box label to change the color of the demand spectrum.
   - The family of demand spectra and the single demand spectrum (variable damping) can either be displayed at the same time, or they can be displayed separately.
   - Check the Show Constant Period Lines At check box to display lines of constant period. These lines appear as radial lines on the capacity spectrum plot. By default the program plots lines for T = .5, 1, 1.5 and 2 seconds. The periods for any of the four curves can be changed by editing the value in one of the four associated edit boxes. A value of 0, or a blank edit box, means to omit that period line.
   - The default color for the period lines is gray. Click the color box adjacent to the Show Constant Period Lines At check box label to change the color of the period lines.
6. In the Damping Parameters area, type in a value in the Inherent/Additional Damping edit box. The value input into this box must be between 0 and 1, inclusively. The default value is 0.05. Referring to ATC-40 equation 8-8, the β0 term is automatically included by the SAP2000 analysis method, and the 5% inherent viscous damping term can be specified in the Inherent/Additional Damping edit box as 0.05. If there is additional viscous damping provided in the structure, perhaps by viscous dampers that are not specifically included in the model , then this damping should also be included in the Inherent/Additional Damping edit box. Thus if the damping inherent in the structure is assumed to be 5% of critical damping, and dampers which provide an additional 7% of critical damping are assumed to be added to the structure (although they are not actually in the model), then the value input in the Inherent/Additional Damping edit box should be 0.12, since 0.05 + 0.07 = 0.12.
7. Select one of the four Structural Behavior Type/Kappa Factor options in the Damping Parameters area. The Structural Behavior Types A, B and C default to the values defined for those structural behavior types in Section 8.2.2.1.1 of ATC-40 . The User Defined Kappa Factor option allows the user to input other kappa (κ) values.
8. If the User Defined Kappa Factor option is selected in the Damping Parameters area, then click the associated Modify/Show button to display the Override Structural Behavior Type dialog box.
9. Click the Override Axis Labels and Range button to modify the default values for the plotted axis labels and range. This will display the Override Axis Labels and Range dialog box.
10. To include additional notes on the printed output, type the notes in the edit box in the Additional Notes For Printed Output area.
11. To print the currently displayed pushover curve, click the File menu at the top of the Pushover Curve dialog box and select Print Graphics.
12. To display output tables for the currently displayed pushover curve, click the File menu at the top of the Pushover Curve dialog box and select Display Tables.
    - If the currently displayed pushover curve is in a base shear versus control joint displacement format, then the displayed table will include the following information for each step of the pushover: base shear, control joint

displacement, and the number of hinges beyond certain control points (B, IO, LS, CP, C, D and E) on their hinge property force-displacement curve.



- If the currently displayed pushover curve is in the ADRS format, then the displayed table will include the following information for each step of the pushover: effective period, effective damping, spectral coordinates (Sa, Sd) of the capacity curve (pushover), spectral coordinates (Sa, Sd) of the single modified demand spectrum curve, and the scaling factors used for converting the force-displacement curve to the ADRS format, PFPhi and Alpha (see Figure 8-5 in ATC-40). Note PFPhi is the denominator in equation 8-4 of ATC-40 and Alpha is the denominator in equation 8-3 of ATC-40 .

13. To print output tables for the currently displayed pushover curve, click the File menu at the top of the Pushover Curve dialog box and select Display Tables. This displays the tables on the screen. Then select Print Tables or Print Tables To File from the File Menu on the dialog box that displays the tables on the screen.
14. To reset the colors used for the pushover curve and the demand spectra to the default values, click the Reset Colors button.
15. The pushover curve is plotted in the upper left-hand corner of the dialog box. Below the plot are four boxes that display information about the plot.
    - The first box displays the coordinates of the cursor (pointer) when it it located over the plot.
    - The second box shows the performance point , that is, the intersection of the capacity curve and the single demand spectrum, in base shear versus control point displacement coordinates.
    - The third box shows the performance point , that is, the intersection of the capacity curve and the single demand spectrum, in spectral acceleration versus spectral displacement coordinates.
    - The fourth box shows the effective period and effective damping at the performance point.
16. Click the Done button to close the Pushover Curve dialog box.

See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Override Axis Labels And Range
Override Structural Behavior Type

Structural Behavior Type / Kappa Factor

Section 8.2.2.1.1 of ATC-40 defines a kappa factor, κ, which modifies the calculated area of the hysteresis loops for different types of structural behavior. Structural Behavior Types A, B, and C are described, and curves for the kappa factor associated with each behavior type are provided (see Figure 8-15 in ATC-40).

Viscous Dampers

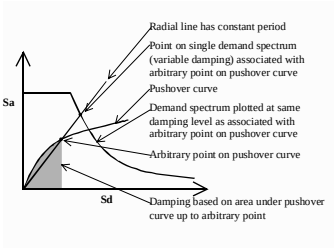
If NLLink elements are included in the pushover the program treats them as linear elements. Their stiffness is based on the linear effective stiffness (KE) and the damping is based on the linear effective damping coefficient (CE). The program use the linear effective damping coefficient and the effective period to calculate damping which is internally added to the specified Inherent/Additional Damping term.

Single Demand Spectrum (Variable Damping)

The single demand spectrum (variable damping) curve is constructed by doing the following for each point on the ADRS pushover curve:

1. Draw a radial line through the point on the ADRS pushover curve. This is a line of constant period.
2. Calculate the damping associated with the point on the curve based on the are under the curve up to that point.
3. Construct the demand spectrum, plotting it for the same damping level as associated with the point on the pushover curve.
4. The intersection point of the radial line and the associated demand spectrum represents a point on the Single Demand Spectrum (Variable Damping) curve.

Note that this method is similar to the method call Procedure B in Chapter 8 of ATC-40 except it does not make the simplifying assumption that the post yield stiffness remains constant.



Override Axis Labels And Range

1. After the pushover analysis has been run, select Show Static Pushover Curve from the Display menu. This displays the Pushover Curve dialog box. Click the Override Axis Labels And Range button to display the Override Axis Labels And Range dialog box.
2. To change the horizontal axis range, type values in the Min and Max edit boxes in the Horizontal Range area.
3. To change the vertical axis range, type values in the Min and Max edit boxes in the Vertical Range area.
4. To modify the horizontal and vertical axis labels, type new labels in the appropriate edit boxes in the Axis Labels area.
5. Click OK to accept the changes in the Override Axis Labels And Range dialog box and return to the Pushover Curve dialog box. Clicking Cancel returns to the Pushover Curve dialog box without saving the changes.

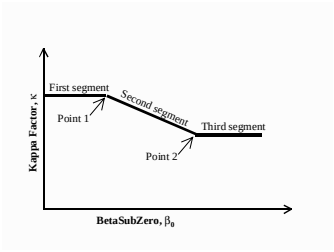
See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data

Assign Frame Hinges (Pushover)
Pushover Curve
Override Structural Behavior Type

Override Structural Behavior Type

1. After the pushover analysis has been run, select Show Static Pushover Curve from the Display menu. This displays the Pushover Curve dialog box. Click the Modify/Show button in the Damping Parameters area to display the Override Structural Behavior Type dialog box.
2. Type in the BetaSubZero (β0) and the Kappa Factor at the beginning of the second segment in the Point 1 area.
3. Type in the BetaSubZero (β0) and the Kappa Factor at the end of the second segment in the Point 2 area.
4. Click OK to accept the changes in the Override Structural Behavior Type dialog box and return to the Pushover Curve dialog box. Clicking Cancel returns to the Pushover Curve dialog box without saving the changes.

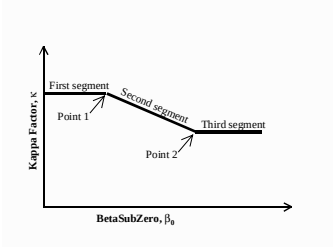


See Also

Nonlinear Static Pushover Analysis
Define Frame Hinge Properties
Frame Hinge Property Data
Frame Hinge Property Data For XXX
Frame Hinge Interaction Surface
Interaction Surface Definition
Define Static Pushover Cases
Static Pushover Case Data
Assign Frame Hinges (Pushover)
Pushover Curve
Override Axis Labels And Range

Kappa Factor

Section 8.2.2.1.1 of ATC-40 defines a kappa factor, κ, which modifies the calculated area of the hysteresis loops to account for pinching of the loops depending on the structural behavior of the building. Structural Behavior Types A, B, and C are described, and curves for the κ factor are provided (see Figure 8-15 in ATC-40). The curves describing the κ factor are trilinear, with the first segment having a constant κ value, the second segment having a linearly varying κ value, and the third segment having a constant κ value. Thus the κ factor curves can be described with just two points, those at the beginning and end of the second segment. The point at the beginning of the second segment is designated Point 1, and the point at the end is Point 2.



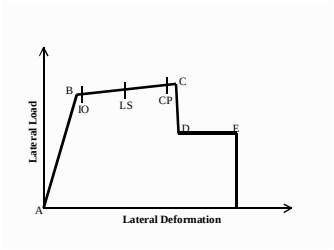
Acceleration-Displacement Response Spectrum (ADRS)

In the ADRS (Acceleration-Displacement Response Spectrum) format both the structural capacity (pushover) curve and the demand spectra are plotted in spectral acceleration versus spectral displacement coordinates. Refer to ATC-40 page 8-12 for additional information, including reference to the origin of this term.

Color Coded Hinge State

From the Options menu, select Colors, and then select the Output tab. The color coding for the pushover hinge state is controlled by the colors in the area labeled Contours. There are ten contour colors; these contour colors also are used in displaying stress contours for shell elements. Assume the color in the top box, next to the label "Min", is designated "Box 1", and the color in the bottom box, next to the label "Max", is designated "Box 10". Then the color coding for pushover hinge states is as defined in the table and figure below.

Point	Color
B	Box 2
IO	Box 3
LS	Box 4
CP	Box 5
C	Box 6
D	Box 7
E	Box 8



Step-By-Step Hinge Formation

To review the pushover hinge formation on a step-by-step basis, do the following:

1. From the Display menu select Deformed Shape.
2. In the Load area, select a pushover load from the drop down box and click the OK button
3. Click the arrow keys in the status bar at the bottom of the SAP2000 window to step through the sequence of hinge formation.

Step-By-Step Pushover Forces

To review the pushover forces on a step-by-step basis, do the following:

1. From the Display menu select Show Element Forces/Stresses, then select Frames.
2. In the Load area, select a pushover load from the drop down box.
3. In the Component area, select a force component to display.
4. Check either the Fill Diagram box or the Show Values on Diagram box and click OK.
5. Click the arrow keys in the status bar at the bottom of the SAP2000 window to step through the sequence of member forces.

DESIGN MENU

Steel Design

On the Design menu, click Steel Design. This will switch the program into the Steel Design mode.

What do you want to do?

- Define Section Properties
- Setting Steel Design Parameters
- Group Elements for Design
- Select/Redefine Design Load Combinations
- Overwrite Element Design Properties
- Replace Auto with Optimal Sections
- Display Steel Stress Check/Design Ratios on Screen
- Display Steel Stress Check/Design Input on Screen
- Review Steel Stress Check/Design Detail
- Redesign a Steel Member
- Start Design/Check of Structure
- Update Analysis Sections
- Reset Design Sections
- Print Design Results to a Printer or a File
- Defining Elements as a single Member for bending

Concrete Design

On the Design menu, click Concrete Design. This will switch the program into the Concrete Design mode.

What do you want to do?

- Define Section Properties
- Setting Concrete Design Parameters
- Select/Redefine Design Load Combinations
- Overwrite Element Design Properties
- Display Concrete Design/Check Output on Screen
- Display Concrete Design/Check Input on Screen
- Review Concrete Design/Check Detail
- View Interaction Diagram for a Concrete Column

 Redesign a Concrete Member
 Start Design/Check of Structure
 Update Analysis Sections
 Reset Design Sections
 Print Design Results to a Printer or a File
 Defining Elements as a single Member for bending

Group Elements for Design

Grouping elements for design will design all the elements in the group to the same section or steel reinforcement area. Using the section or reinforcing area that will work for all of the members in the group.

NOTE: Grouping elements for design currently only works on steel members with Auto Select sections assigned to them

1. On the Design Menu, click Select Design Groups... This will display either the Steel Design Group Selection or Concrete Design Group Selection dialog box.
2. In the Steel or Concrete Design Group Selection box:
   - Select previously defined Groups from the List of Groups
   - Click on the Add button to add Groups to the Design Groups list
   - Clicking on the Remove button will remove the selected Groups from the Design Groups list
   - Click OK

Start Design/Check of Structure

1. Select the Frame Elements to be designed or checked
2. On the Design Menu, click Start Design/Check of Structure...
3. This will immediately start the stress check or design of all the selected frame members in the model.
4. The ratio of actual to allowable stress/capacity or member design reinforcement, based upon the previously selected combos, will be displayed.
5. Subsequent member selections can be Designed/Checked and added to the display.
6. The Check/Design results display will only affect the selected elements and all other elements will show the previous values, if any.

Note: If no Elements are selected then the program will automatically select all Frame Elements and perform the Design or Check.

Selecting Design Load Combinations

1. On the Design Menu, click Select Design Combos**...** This will display the Design Load Combinations Selection dialog box.
2. In the Design Load Combinations Selection box:
   - Select previously defined Combos from the List of Combos
   - Click on Add button to add Combos to the Design Combos list
   - Clicking on Remove button will remove the selected Combos from the Design Combos list
   - Click OK

Note: The Design Load Combinations Selection menu lists a series of default load combinations labeled DCON# for Concrete design and DSTL# for Steel design. The number and type of default combinations depends upon the static Load Cases previously defined. Other user defined Load Combos that were indicated as design combos in the Load Combination Data dialog box will also be included in the Design Combos

If other load combinations have been defined and were not indicated as design combos in the Load Combination Data dialog box, they are shown under List of Combos. The Design Combos list can be modified by removing one or more default Combinations and/or by adding from the List of Combos.

Clicking on the Show button will present the Load Cases and Scale factors for the highlighted

Combo.

Overwrite Element Design Data

1. Select the Elements for which you want to overwrite assignments.
2. On the Design Menu, click ReDefine Element Design Data... This will display the Element Overwrite Assignment menu.
3. In the Element Overwrite Assignments dialog box:
   - Check the desired Element assignments to be overwritten by clicking on the relevant check boxes for K-factors, Unbraced Lengths, Cm, Cb, Live Load Reduction Factor or Magnification Factors
   - Provide the new value in the edit boxes
   - Change the Element Section by clicking on the Change button
   - If Concrete sections are selected, choose the Element Types to be Special, Intermediate or Ordinary.
   - If Steel sections are selected, click the Overwrite Allowable Stresses button to overwrite axial, bending and shear allowable stresses.
   - To define a set of elements to be a part of the same bending member see: Defining Elements as a Single Member for Bending .
   - Click the OK button

Note: Entering a value of **Zero** sets the value back to the program default.

The Element Overwrite menu lists a series of assignments, which can be overwritten. The selection options are mutually exclusive and may be selected in any order.

Selecting and modifying the appropriate factor will overwrite the default or computed value.

The new value(s) are displayed in the active window.

The Effective Length Factors K33 (major) and K22 (minor) can be any positive reasonable number.

The Unbraced Length Ratios L33 (major) and L22 (minor) are to be input as a positive ratio of the full length, i.e. 0.33, 0.75, etc.

The bending coefficient Cm can be any positive number less than or equal to 1.0.

The moment gradient coefficient Cb can be any positive number from 1.0 to 2.3.

The Live Load Reduction Factor is a positive number between 0 and 1.0 and represents a fraction of the full live load previously input.

The Non Sidesway and Sidesway Magnification Factors DB33, DB22, DS33, DS22 are numbers greater than or equal to 1.0. If P-delta is carried out DS33 and DS22 may be set to 1.0

Tip: Element Class specification (Column or Beam) for Concrete Sections is done at the Section definition level.

Defining Elements as a Single Member for Bending

*********** NOTE: THIS OPTION IS NOT ACTIVE IN THE CURRENT VERSION OF THE PROGRAM **********

It is often practical to divide structural members into multiple elements when developing your computer model. However for design it is important to know the actual member start and end point. This information is used for defining un braced length in the major and minor access bending, Cb and Cm values where applicable as well as the provisions relating to the distribution of moment and shear along a member for Special and Intermediate moment resisting frames.

SAP2000 provides various ways to enter this information for design. L22,L33 Cb and Cm can be defined directly for each element in the model using the Element Overwrite Assignments dialog box. This will however not supply the design module with the distribution of moment and shear along a member.

To Define a set of frame elements to be part of a single Frame Member for bending in the major and/or minor direction:

1. Select the frame elements that you want to assign as a single Member for bending in the major and/or minor direction.
2. On the Design Menu, click ReDefine Element Design Data... This will display the Element Overwrite Assignment menu.
3. In the Element Overwrite Assignments dialog box:
   - Check Define Member for Major Bending to assign the selected frame elements to be part of the same frame member for major axis bending.
   - Check Define Member for Minor Bending to assign the selected frame elements to be part of the same frame member for minor axis bending.
   - Click the OK button.

Note: Values overwritten explicitly in the Element Overwrite Assignments dialog box will take precedence over the values calculated by the defined bending members. All elements selected as a part of the defined Bending Member must be connected. Only one Member can be defined at one time.

Replace Auto with Optimal Sections

This selection will replace the “AUTO” sections with the optimally section chosen from the auto select group. This is an irreversible action i.e. once the AUTO sections are replaced they will no longer be available for optimal design. You will have to re-assign them as AUTO if you want them included in Optimization.

1. Select the frame members that you want to replace the analysis sections for.
2. On the Design Menu, click Replace Auto w/Optimal Sections.
3. This will issue a warning message that Replacing Auto-Selected Sections will change the analysis model and cause analysis results to be deleted.
4. Click OK if acceptable otherwise click Cancel.

Note: This option should only be used after the final design cycle. Use Update Analysis Sections to update the sections chosen at the design/redesign level as it will update the sections properties for analysis without changing the sections from being “AUTO”.

Display Design Input/Results on Screen

What do you want to do?

Display Steel Stress Check/Design Input on Screen

Display Steel Stress Check/Design Ratios on Screen

Display Concrete Design/Check Input on Screen

Display Concrete Design/Check Output on Screen

Display Steel Stress Check/Design Ratios on Screen

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose either P-M ratios or Shear Ratios to be displayed.
4. Click OK. This will refresh the window and display the Stress check Ratios.

Display Concrete Design/Check Output on Screen

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose either Longitudinal Reinforcement or Shear Reinforcement to be displayed.

4 Click OK. This will refresh the window and display the area of Reinforcement.

Display Steel Stress Check/Design Input on Screen

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results, click on Design Input, then from the drop down list box you may choose Design Sections, K-factors, Effective Length Ratios, Cm Factors, Cb Factors, Live Load Reduction Factors, DS Factors, DB Factors, Design Type, Axial Allowables, Bending Allowables or Shear Allowables to be displayed.
4 Click OK. This will refresh the window and display the input parameter selected.

Display Concrete Design/Check Input on Screen

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results, click on Design Input, then from the drop down list box you may choose Design Sections, K-factors, Effective Length Ratios, Cm Factors, Cb Factors, Live Load Reduction Factors, DS Factors, DB Factors or Design Type to be displayed.
4 Click OK. This will refresh the window and display the input parameter selected.

Review Stress Check/Design Details

What do you want to do?

 Review Steel Stress Check/Design Detail

 Review Concrete Design/Check Detail

Review Steel Stress Check/Design Detail

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results box, click on Design Output, then from the drop down list box choose either P-M ratios or Shear Ratios to be displayed.
4 Click OK. This will refresh the window and display the Stress Check Ratios.
5 Right Click on a member for which you want see the Design Details. This will display the Steel Stress Check Information box with a breakdown of the Stress Ratio (Axial, Major and Minor Bending, and Major and Minor Shear).
6 In the Steel Stress Check Information box, click Details to see more details (Actual Stress, Allowable Stress, Cm, Cb, Fe' etc.) pertaining to the calculation of the Stress Ratio.

Review Concrete Design/Check Detail

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results box, click on Design Output, then from the drop down list box choose either Longitudinal Reinforcement or Shear Reinforcement to be displayed.
4 Click OK. This will refresh the window and display the area of Reinforcement.
5 Right Click on a member for which you want see the Design Details. This will display the Concrete Design Information box with Flexural and Shear Reinforcement areas listed (Top Reinforcement, Bottom Reinforcement and Shear Reinforcement for Beams and Longitudinal and Major and Minor Shear Reinforcement for Columns).
6 In the Concrete Design Information box, click Details to see more details (Vn, Vc, Vs, Cm Major and Minor Magnification factors etc.) pertaining to the calculation of the Reinforcement for each of the load

combinations.

View Interaction Diagram for a Concrete Column

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose Longitudinal Reinforcement to be displayed.
4. Click OK. This will refresh the window and display the area of Longitudinal Reinforcement.
5. Right Click on a Column for which you want to see the Interaction Diagram. This will display the Concrete Design Information box.
6. In the Concrete Design Information box, click Interaction. This will display the interaction diagram. Click on pxy to see the 3-D interaction, xy to see the interaction between Mx and My, px to see the interaction between P and Mx and py to see the interaction between P and My.
7. Use the spin buttons to change the Plan and Elevation angles.
8. To zoom on a specific part of the plot:
   - Point to a corner of the region, hold down the mouse button and drag to define the rectangular zoom region, and release the mouse button.
   - Clicking anywhere in the main display area brings back the full view.
9. Use the Print menu to print Interaction Cures.

Redesign a Steel Member

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose either P-M ratios or Shear Ratios to be displayed.
4. Click OK. This will refresh the window and display the Stress Check Ratios.
5. Right Click on a member for which you want see the Design Details. This will display the Steel Stress Check Information box with a breakdown of the Stress Ratio (Axial, Major and Minor Bending, and Major and Minor Shear).
6. In the Steel Stress Check Information box, click Redesign. This will display the Element Overwrite Assignments box.
7. Make appropriate changes to the assignments and click OK. This will bring you back to the Steel Stress Check Information box with the redesign carried out and the new results displayed in the design information box.

Redesign a Concrete Member

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose either Longitudinal Reinforcement or Shear Reinforcement to be displayed.
4. Click OK. This will refresh the window and display the area of Reinforcement.
5. Right Click on a member for which you want see the Design Details. This will display the Concrete Design Information box with Flexural and Shear Reinforcement areas listed (Top Reinforcement, Bottom Reinforcement and Shear Reinforcement for Beams and Longitudinal and Major and Minor Shear Reinforcement for Columns).
6. In the Concrete Design Information box, click Redesign. This will display the Element Overwrite Assignments box.
7. Make appropriate changes to the assignments and click OK. This will bring you back to the Concrete Design Information box with the redesign carried out and the new results displayed in the design information box.

Update Analysis Sections

SAP2000 allows you to change section properties while in the Design Mode (Steel or Concrete) and redesign the member. Redesign uses the analysis results from the original properties unless you update analysis section properties and re-run the analysis. It is advisable to use the Update Analysis Section option after you have reviewed the entire design as the analysis results will no longer be available after the properties are updated.

1. Select the frame elements that you want to update.
2. On the Design Menu, click Update Analysis Sections...
3. This will issue a warning message that Updating Analysis Sections will change the analysis model and cause analysis results to be deleted.
4. Click OK if acceptable, otherwise click Cancel.

Reset Design Sections

SAP2000 allows you to change section properties while in the Design Mode (Steel or Concrete) and redesign the member. Some times you may find that you want to revert to having the analysis section used for design.

1. Select the frame elements that you want to update.
2. On the Design Menu, click Reset Design Sections...
3. This will reset the same sections used in the analysis for design.

OPTIONS MENU

Preferences

SAP2000 has some built-in Tolerances and default values for some parameters and Design Codes. It is possible to change most of those default values by editing them in the three Preference Tabs, namely Dimensions, Steel and Concrete Design Coeds and parameters.

1. On the Options menu, click Preferences… This will display the Preferences dialog box with three Tabs namely Dimensions, Steel and Concrete.
2. Click on the appropriate tab to modify the default options.
3. Click OK.

Also see:

Setting Steel Design Parameters

Setting Concrete Design Parameters

Setting Steel Design Parameters

1. Select Preferences from the Options menu.
2. Press on the Steel Tab to view the Steel Design Preferences .
6. From the Steel Design Code drop down list box select the code you want used in steel design.
7. If the Section Properties file is not the one you want used in the design then press the Choose File button to select another file.
8. Select the preferred design method for multi valued load cases if you have any in the design.
   - The envelope option will design for the maximum and minimum values in a member.
   - The Time Step option will check the section for each time step of a Time History analysis. This method is very time consuming and should be used mainly for the final design.
   - The Max/Min Correspondence will design for the maximum and minimum moment/ axial force and its corresponding axial force/moment.
9. Click on OK when finished.

Setting Concrete Design Parameters

1. Select Preferences from the Options menu.

2 Press on the Concrete Tab to view the Concrete Design Preferences .
3 From the Concrete Design Code drop down list box select the code you want used in concrete design.
4 Edit any Strength reduction factors that may not be satisfactory.
5 Edit the Interaction Diagram Parameters that are used to develop for the column design.
6 Select the preferred design method for multi valued load cases if you have any in the design.
- The envelope option will design for the maximum and minimum values in a member.
- The Time Step option will check the section for each time step of a Time History analysis. This method is very time consuming and should be used mainly for the final design.
- The Max/Min Correspondence will design for the maximum and minimum moment/ axial force and its corresponding axial force /moment.

7 Click on OK when finished.

Setting Colors

In SAP2000 you have the ability to change the colors used by the program for both viewing in the graphic interface and for printing to a color or monochrome printer.

1 On the Options menu, click Colors… This will display the Assign Colors dialog box with a Display and Output tab.
- Under the Display tab you can set colors for viewing the structural configuration including: Joints, Frames Shells, Grids and the Background.
- Under the Output tab you can set the colors for the analysis and design results.

2 On either form you can choose to set the colors for the Screen and the Printer.
- Select Screen or Printer.
- Change the colors by pressing the appropriate button under the Display tab or by double clicking on the appropriate color under the Output tab.
- Then select a new color and press the OK button.

3 If you want to set the colors back to the SAP2000 default values, press the Reset button.
4 Click OK.

Note: The Default printer colors are set for a monochrome printer. To print in color you will need to manually select the colors you want for the printer.

Editing Toolbars

Both the Toolbar and the Floating Toolbar can be edited to suit the way you work with SAP2000.

1 Double click on the Toolbar or Floating Toolbar away from a button, e.g. between two buttons or to the right of the last button
2 Add and Remove buttons as required.
3 Click on Move Up and Move Down to locate the buttons as desired.
4 The Reset button sets the Toolbar to its default settings.
5 Press the Close button when finished.

User Defined Coordinate System

The default coordinate system is the global coordinate system. All user defined coordinate systems are defined relative to this global coordinate system. All coordinate systems follow the right-hand rule.

Also See:

 Defining a new Coordinate System

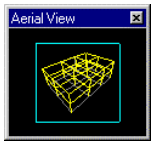
Aerial View

The Aerial View is a tool that helps you in viewing the model. It displays a full view of the active window's drawing in a separate window so that you can quickly zoom into any area of your model without having to restore the full view first. It can also be used to show you which part of the model you are zoomed into when working with multiple windows and large models. Each time the model is edited the Aerial View is updated.

The light blue rectangle in the Aerial View window is the viewing limits in the current window.

To zoom into an area of the model using the Aerial View simply point to a corner of the region, hold down the mouse button then drag to define the rectangular zoom region, and release the mouse button.

Note: The Aerial View window can be made visible by selecting Show Aerial View from the Options menu. It can be resized like any other window and can be moved anywhere on your desktop. The light blue "zoom area" box will always keep the same aspect ratio as the active window.



End

FINAL NOTES

General Tips

1. You can view the time history displacements in real time video via the SAP2000 video creation option.
2. You can use the new nonlinear link element to model base isolators and dampers.
3. You can animate deformed shapes and mode shapes in 3-d perspective by a simple click of the 'Start Animation' button.
4. You can animate shell structures displaying stress contours with the corresponding deformed shapes in full in 3-d perspective.
5. You can automatically break frame members into elements at their intersection points in 3-d space.
6. You can automatically generate a macro shell element by a single click on the working grid and then mesh it into finer elements with simple meshing parameters.
7. Large models can be generated automatically using the SAP2000 templates. The models can then be sculptured with on screen editing to satisfy specific situations.
8. In SAP2000 you only define elements. All joints needed by the elements are automatically generated. No pre-definition of joints is needed.
9. You can display information about any joint or element by pointing to it and clicking the right mouse button"
10. The SAP2000 model generation features allow on screen linear and cylindrical replication and mirroring of selected portions of the model.
11. Joints can be glued to a grid line. Moving the grid line will modify the location of all joints on the grid line and stretch or shrink all elements connected to those joints.
12. You can double click a grid line to bring up an edit box that will allow you to accurately reposition the grid line.
13. You can unlock the gridlines and drag and drop them on the screen to new locations.
14. You can add a new grid line by holding the CTRL-KEY down and clicking on an existing grid line and dragging away a duplicate.
15. Clicking on a view with the CTRL-KEY down will give you a list of the joints and elements within the click tolerance.
16. The SAP2000 nonlinear time history analysis is based upon a new special purpose algorithm called the Wilson-FNA method.
17. You can produce scaled perspective extrusions of the structural sections by simply activating the SAP2000 'extrude' option.
18. You can copy a selected three dimensional region of your model to the clipboard and paste it back at any location in 3d space.
19. While displaying mode shapes or deformed shapes the vector being displayed can be instantaneously changed with the '+/-' buttons that will appear at the bottom of the screen.
20. You can switch units at any time while creating the model.
21. Back in 1970, with the first release, the slang name SAP was selected to remind users that this program, like all computer programs, lacks intelligence and that it is the responsibility of the engineer to use the program correctly.

22 'With good engineering judgment you can produce on the back of an envelope that which otherwise cannot be produced with a ton of computer output'. -- Anonymous Circa 1974

Specific Tips

SAP2000 Versions and Limitations

SAP2000 may be purchased in the following four versions:

SAP2000 Standard

Capacity	1500 Nodes
Analysis	Static and Dynamic Response Spectrum Analysis
Elements	FRAME and SHELL Elements only
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

SAP2000 Plus

Capacity	No Practical Limit
Analysis	Static, Dynamic Response Spectrum, Time History and Bridge Analysis
Elements	FRAME, SHELL, PLANE, SOLID and ASOLID Elements
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

SAP2000 Nonlinear

Capacity	No Practical Limit
Analysis	Static, Dynamic Response Spectrum, Time History, Bridge and Dynamic Nonlinear Time History Analysis
Elements	FRAME, SHELL, PLANE, SOLID, ASOLID, and NLLINK (as external Damping , Base Isolators, Gap and Hook Elements)
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

SAP2000 Educational

Capacity	30 Nodes
Analysis	Static, Dynamic Response Spectrum, Time History, Bridge and Dynamic Nonlinear Time History Analysis
Elements	FRAME, SHELL, PLANE, SOLID, ASOLID, and NLLINK (as external Damping, Base Isolators, Gap and Hook Elements)
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

Error Messages

Phone and Fax Support

Support is available, free of charge, for 90 days after purchase and for the term of the maintenance agreement. Please call CSI's offices to inquire about the maintenance agreement.

Standard phone and fax support is available in the United States, from CSI support engineers via a toll call between 8:30 A.M. and 5:00 P.M. Pacific time, Monday through Friday, excluding holidays.

If you are experiencing problems using the software:

- Consult the documentation and other printed information included with the software.
- Check the online Help.

If you cannot find a solution then:

- Contact CSI's Offices via **phone** at (510) 845-2177.
- Send a **fax** with questions and a sketch of the model to CSI at (510) 845-4096.
- You may also send the input file via e-mail to support@csiberkeley.com .

When you call, you should be at your computer and have the appropriate documentation (User's manual) at hand. Please have the following information:

- The version number of the program that you are using.
- The hardware configuration (computer make, operating system, hard disk size and RAM size).
- The exact wording of error messages that appeared on your screen.
- A description of what happened and what you were doing when the problem occurred.
- A description of how you tried to solve the problem.

When you send a Fax or e-mail, include the following information:

- Your name, Company name, and phone number.
- The version and version number of the program that you are using.
- The hardware configuration (computer make, operating system, hard disk size and RAM size).
- The exact wording of any error messages that appeared on your screen.
- A full description of what you are having problem with.
- A full description of how you tried to solve the problem.
- An attachment including your model and any other pertinent data files. (For e-mail only!)

Online Support

Online support is available by:

Sending an e-mail and your model file to support@csiberkeley.com

Visiting CSI's web site at http://www.csiberkeley.com (frequently asked questions)

When you send a Fax or e-mail, include the following information:

- Your name, Company name, and phone number.
- The version and version number of the program that you are using.
- The hardware configuration (computer make, operating system, hard disk size and RAM size).
- The exact wording of any error messages that appeared on your screen.
- A full description of what you are having problem with.
- A full description of how you tried to solve the problem.
- An attachment including your model and any other pertinent data files. (For e-mail only!)

Frequently Asked Questions

Q. Why do 2-D views show objects which are not on that plane?

R. The length tolerance, given as Auto-merge tolerance in the Dimension tab of Preferences in the Options pull down menu may be too large. The default value is 0.1 inches. The value of this tolerance determines the clipping thickness of the 2-D view plane and all joints within that clipping depth will be available on that plane.
Caution: This length tolerance is also used to merge together joints automatically generated.

Q. What does the warning message "lost 6.4 digits of accuracy" mean?

R. Equation solution in SAP2000 is done in double precision arithmetic. This allows for 15 significant figures to be carried in the calculation. An internal check estimates the number of digits of accuracy lost during the reduction of each degree of freedom. A warning is issued when the loss is estimated to be above 6 digits and the degree of freedom is listed when this occurs. Users should check the displacements, reactions and global equilibrium balances to assure that the digits loss has not unacceptably degraded the results. When the program detects over 11 digits accuracy loss it stops further processing as the results are suspect. Users should check for global instability and local instability at or around the degree of freedom reported.

Q. What do I need to know when running a Non-linear Analysis

R.

- Defining NLLink Members

- Assign mass to nonlinear degrees of freedom of NLLink members. Otherwise Ritz modes may be missing or inaccurate for the NLLink's nonlinear degree of freedom.
- Do not use very large stiffness values. If you want something to be very stiff use a value 100 to 1000 times stiffer then connecting elements.
- Use reasonable values provided by manufacturer for NLLink member properties.
- Assign enough member properties for the NLLink member so that it is stable. If some rotational and displacement properties are not set the structure may become locally unstable.

- Defining Time History Analysis
  - Select Nonlinear Analysis type in the Time History Case Data dialog box. NLLink members act nonlinearly ONLY in a Nonlinear Time History analysis.
  - For Isolators and energy absorbing devices, use a small reasonable value for modal damping since the NLLink elements are expected to provide the damping for the structure.
- Setting Options for analysis under Dynamic Analysis Parameters
  - Select enough modes to capture complete behavior:
  - Start with the number of modes required for a linear Time History.
  - Add at least one mode for each static load vector(e.g. gravity)
  - Add at least one mode for each nonlinear DOF
  - Select Ritz Analysis Type. Eigen analysis is not appropriate unless ALL possible eigen modes are found.
  - Select the appropriate starting Ritz vectors. If static loads are used with loading functions in the time history analysis then those loads need to also be included in the list of starting load vectors.
  - Check the Include NLLink Vectors option.

Q. Now I am doing nonlinear analysis for a structure consist of cables. In the output file, there are still some compressive axial forces in the hook-type cable. Could you give me some advice? I appreciate your time and help.

R. The NLLINK elements have a dual behavior: they behave nonlinearly ONLY during a *nonlinear time-history analysis*; for all other analyses, they behave linearly.

The results in the output file are for linear analyses, hence the elements can carry tension and compression.

To perform a nonlinear static analysis, define a nonlinear time-history analysis, ramp up the static load slowly, and use a large modal damping, say 99%. See pages 250, 265, 319, and 320 in the *SAP2000 Analysis Reference, Volume 1*. Be sure to use Ritz analysis with the proper starting load vectors, pages 293-297 in *Volume 1*.

Q. How can I obtain the mode shape joint coordinates from a modal analysis in SAP2000.

R. The way to get the Relative joint displacement for a mode is to run the modal analysis, then place a mode as the only load in a load combo. Then print the displacements for the combo with the mode in it. You can do this for any of the modes that have been recovered.

You can also select from the Analyze... Set options menu to have the modes printed in the *.OUT file by selecting the load combos with the mode shape in it

Program Manuals

1 SAP2000 Volume I

This volume describes all of the theoretical concepts behind the modeling and analysis features offered by the SAP2000 structural analysis program. The focus of this manual is on the analysis portion of the program.

2 SAP2000 Volume II

This volume describes the contents of SAP2000 text input data file. Most users can skip this volume.

3 SAP2000 Getting Started

This manual is designed to help you become quickly productive with SAP2000. It gives an introduction to the basic concepts of the graphical user interface and includes a tutorial on using the program. It is strongly recommended that you read this manual and work the tutorial before attempting a real project with SAP2000.

4 SAP2000 Verification Manual

This manual contains example problems solved using SAP2000. The problems demonstrate many of the capabilities of SAP2000. For purposes of verification, key results from the examples are compared with theoretical or published results.

5 **SAP2000 Steel Design Manual / SAP2000 Concrete Design Manual**
This manual includes a steel/ concrete design tutorials and description of the implementation of the design coeds in SAP2000.

6 **SAP2000 Tutorial Manual**
This manual has various tutorial to help users make the most efficient use of SAP2000.

Primers

Newsletters

Text Books

1 **Three Dimensional Analysis of Structures**
With Emphasis on Earthquake Engineering
Edward L. Wilson
If you are interested in obtaining a copy of this book, contact CSI for cost and shipping.

References

AASHTO, 1992
Standard Specifications for Highways Bridges, 15th Edition, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

AASHTO, 1994
LRFD Bridge Design Specifications, Customary U.S. Units, 1st Edition, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

ACI, 1995
Building Code Requirements for Structural Concrete (ACI 318-95) and Commentary (ACI 318R-95), American Concrete Institute, Farmington Hills, Mich.

AISC, 1994
Manual of Steel Construction, Load & Resistance Factor Design, 2nd Edition, American Institute of Steel Construction, Chicago, Ill.

ATC, 1996
Seismic Evaluation and Retrofit of Concrete Buildings, Volume 1, ATC-40 Report, Applied Technology Council, Redwood City, California.

K. J. Bathe, 1982
Finite Element Procedures in Engineering Analysis, Prentice-Hall, Englewood Cliffs, N.J.

K. J. Bathe and E. L. Wilson, 1976
Numerical Methods in Finite Element Analysis, Prentice-Hall, Englewood Cliffs, N.J.

K. J. Bathe, E. L. Wilson, and F. E. Peterson, 1974
SAP IV — A Structural Analysis Program for Static and Dynamic Response of Linear Systems, Report No. EERC 73-11, Earthquake Engineering Research Center, University of California, Berkeley.

J. L. Batoz and M. B. Tahar, 1982
"Evaluation of a New Quadrilateral Thin Plate Bending Element," *International Journal for Numerical Methods in Engineering*, Vol. 18, pp. 1655–1677.

Caltrans, 1995
Bridge Design Specifications Manual, as amended to December 31, 1995, State of California, Department of Transportation, Sacramento, Calif.

R. D. Cook, D. S. Malkus, and M. E. Plesha, 1989
Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, New York, N.Y.

R. D. Cook and W. C. Young, 1985
Advanced Mechanics of Materials, Macmillan, New York, N.Y.

FEMA, 1997
NEHRP Guildelines for the Seismic Rehabilitation of Buildings, Developed by the Building Seismic Safety Council for the Federal Emergency Management Agency (Report No. FEMA 273), Washington, D.C.

A. K. Gupta, 1990
Response Spectrum Method in Seismic Analysis and Design of Structures, Blackwell Scientific Publications, Cambridge, Mass.

J. P. Hollings and E. L. Wilson, 1977
3–9 Node Isoparametric Planar or Axisymmetric Finite Element, Report No. UC SESM 78-3, Division of Structural Engineering and Structural Mechanics, University of California, Berkeley.

A. Ibrahimbegovic and E. L. Wilson, 1989
"Simple Numerical Algorithms for the Mode Superposition Analysis of Linear Structural Systems with Nonproportional Damping," *Computers and Structures*, Vol. 33, No. 2, pp. 523–531.

A. Ibrahimbegovic and E. L. Wilson, 1991
"A Unified Formulation for Triangular and Quadrilateral Flat Shell Finite Elements with Six Nodal Degrees of Freedom," *Communications in Applied Numerical Methods*, Vol. 7, pp. 1–9.

L. E. Malvern, 1969
Introduction to the Mechanics of a Continuous Medium, Prentice-Hall, Englewood Cliffs, N.J.

S. Nagarajaiah, A. M. Reinhorn, and M. C. Constantinou, 1991
3D-Basis: Nonlinear Dynamic Analysis of Three-Dimensional Base Isolated Structures: Part II, Technical Report NCEER-91-0005, National Center for Earthquake Engineering Research, State University of New York at Buffalo, Buffalo, N. Y.

Y. J. Park, Y. K. Wen, and A. H-S. Ang, 1986
"Random Vibration of Hysteretic Systems under Bi-Directional Ground Motions," *Earthquake Engineering and Structural Dynamics*, Vol. 14.

R. J. Roark and W. C. Young, 1975
Formulas for Stress and Strain. 5th Edition, McGraw-Hill, New York, N.Y.

R. L. Taylor and J. C. Simo, 1985
"Bending and Membrane Elements for Analysis of Thick and Thin Shells," *Proceedings of the NUMEETA 1985 Conference,* Swansea, Wales.

K. Terzaghi and R. B. Peck, 1967
Soil Mechanics in Engineering Practice, 2nd Edition, John Wiley & Sons, New York, N.Y.

S. Timoshenko and S. Woinowsky-Krieger, 1959
Theory of Plates and Shells, 2nd Edition, McGraw-Hill, New York, N.Y.

Y. K. Wen, 1976
"Method for Random Vibration of Hysteretic Systems," *Journal of the Engineering Mechanics Division*, ASCE, Vol. 102, No. EM2.

D. L. White and J. F. Hajjar, 1991
"Application of Second-Order Elastic Analysis in LRFD: Research to Practice," *Engineering Journal*, ACI, Vol. 28, No.

4, pp. 133–148.

E. L. Wilson, 1970
SAP — A General Structural Analysis Program, Report No. UC SESM 70-20, Structural Engineering Laboratory, University of California, Berkeley.

E. L. Wilson, 1972
SOLID SAP — A Static Analysis Program for Three Dimensional Solid Structures, Report No. UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkeley.

E. L. Wilson, 1985
"A New Method of Dynamic Analysis for Linear and Non-Linear Systems," *Finite Elements in Analysis and Design*, Vol. 1, pp. 21–23.

E. L. Wilson, 1993
"An Efficient Computational Method for the Base Isolation and Energy Dissipation Analysis of Structural Systems," ATC17-1, *Proceedings of the Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control*, Applied Technology Council, Redwood City, Calif.

E. L. Wilson and M. R. Button, 1982
"Three Dimensional Dynamic Analysis for Multicomponent Earthquake Spectra," *Earthquake Engineering and Structural Dynamics*, Vol. 10.

E. L. Wilson, A. Der Kiureghian, and E. P. Bayo, 1981
"A Replacement for the SRSS Method in Seismic Analysis," *Earthquake Engineering and Structural Dynamics*, Vol. 9.

E. L. Wilson and I. J. Tetsuji, 1983
"An Eigensolution Strategy for Large Systems," *Computers and Structures*, Vol. 16.

E. L. Wilson, M. W. Yuan, and J. M. Dickens, 1982
"Dynamic Analysis by Direct Superposition of Ritz Vectors," *Earthquake Engineering and Structural Dynamics*, Vol. 10, pp. 813–823.

V. Zayas and S. Low, 1990
"A Simple Pendulum Technique for Achieving Seismic Isolation," *Earthquake Spectra*, Vol. 6, No. 2.

O. C. Zienkiewicz and R. L. Taylor, 1989
The Finite Element Method, 4th Edition, Vol. 1, McGraw-Hill, London.

O. C. Zienkiewicz and R. L. Taylor, 1991
The Finite Element Method, 4th Edition, Vol. 2, McGraw-Hill, London.

End User License Agreement for CSI Software

This is a legal agreement between you (either an individual or entity), the end user, (hereinafter referred to as "Licensee") and COMPUTERS AND STRUCTURES, INC. (here inafter referred to as "CSI"). If Licensee does not agree to the terms of this Agreement, Licensee shall promptly return the unopened disk pack age and any accompanying items (including written materials and binders or other containers) to the place Licensee obtained them for a full refund. This package is to be opened only by the Licensee (or authorized representative thereof). By opening this sealed diskette package, the Licensee indicates acceptance of this Agreement.

GRANT OF LICENSE.

Dedicated Use. CSI grants to Licensee the right to use one (1) copy of the software program identified above (the "Software") on a single computer ("Dedicated Computer"). Licensee may transfer the Software to another single com puter, provided Licensee does so no more often than once every ninety (90) days and no copies of the Software licensed herein are retained for use on any other computer.

Transferability. CSI grants to Licensee a nontransfer able License to use this Software. Multiple licenses of the Software to same licensee are available at discounted rates. This License limits the use of this copy of

the Software to projects of the Licensee. Application of the Software as a service to other engineer ing firms or individuals, or use of the Software by other engineering firms or individuals, is not allowed. Licensee shall not sell, rent, lease, trans fer, network, publish, dis close, display or otherwise make available any portions of the Software or copies thereof to others.

For the purposes of this section, "use" means loading the Software into RAM, as well as installation on a hard disk or other storage device (other than a network server). Licensee may access the Software from a hard disk, over a network, or any other method Licensee chooses, so long as Licensee otherwise complies with this CSI License Agreement.

TERM OF LICENSE.

The term of this License shall begin from the date of this License and will last as long as the Licensee complies with the terms of this Agreement. This License is subject to cancellation by CSI if Licensee fails to comply with the terms and conditions of this Agreement. Within five (5) days of such cancellation, Licensee shall return the Software to CSI.

COPYRIGHT.

The Software is owned by CSI and is protected by United States copyright laws and international treaty provisions. Therefore, Licensee must treat the Software like any other copy righted material (e.g., a book or musical recording) except that Licensee may either make one (1) copy of the Software solely for backup or archival purposes, or transfer the Software to a single hard disk, provided Licensee keeps the original solely for backup or archi val purposes. Licensee may not copy the written materials accompany ing the Software.

OTHER RESTRICTIONS.

This CSI License Agreement is Licensee's proof of license to exercise the rights granted herein and must be retained by Licensee. Licensee agrees to secure and protect the Software and copies thereof in a manner consis tent with the maintenance of the proprietary rights of CSI. Licensee agrees to take appropriate action, by instruction or agreement with its employees who are permitted access to the Software, to protect unauthorized pro liferation of the Software. Licensee may not reverse engineer, decompile, or disassemble the Software. If the Software is an update, any transfer must include the update and all prior ver sions. If the Software pack age contains both 3.5" and 5.25" disks, then Licensee may use only the disks appropriate for Licensee's single designated computer or network server. Licensee may not use the other disks on another computer or com puter network, or loan, rent, lease, or transfer them to another user except as part of a transfer or other use as expressly per mitted by this CSI License Agreement.

PROGRAM UPGRADES.

From time to time, CSI may, at its discretion, offer program upgrades. If offered, such upgrades will be provided at no charge to the Licensee for a period of ninety (90) days from the date of this Agreement. Thereafter, upgrades will be provided under the terms defined by the CSI Software Maintenance Agreement.

SUPPORT.

User support will be available to one (1) contact individual designated by the Licensee, for a period of ninety (90) days from the date of this Agreement. Thereafter, continued support will be available under the terms defined by the CSI Software Maintenance Agreement.

During the term of this License, CSI, at its discretion, will assist Licensee in resolving problems suspected of being caused by possible defects in the Software. However, if it is determined that the problem stems from a user data error, the Licensee agrees to reimburse CSI for the time spent in resolving the problem, at the prevailing professional rates of CSI.

LIMITED WARRANTY; NO OTHER WARRANTIES.

CSI warrants that the Software will perform substantially in accordance with the accompanying written materials for a period of ninety (90) days from the date of receipt. Any implied war ranties on the Software are limited to ninety (90) days. Some states do not allow limitations on duration of an implied war ranty, so the above limitation may not apply to Licensee.

CSI disclaims all other warranties, either express or implied, including but not limited to implied warranties of mer chantabil ity and fitness for a particular purpose, with respect to the Software, the accompanying written materials, and

any accompany ing hardware. This limited warranty gives Licensee specific legal rights. Licensee may have others, which vary from state to state.

CUSTOMER REMEDIES.

CSI's entire liability and Licensee's exclusive remedy shall be, at CSI's option, either (a) return of the price paid or (b) repair or replacement of the Software that does not meet CSI's Limited Warranty and that is returned to CSI with a copy of Licensee's receipt. This Limited Warranty is void if failure of the Software has resulted from accident, abuse or misap plication. Any replacement Software will be warranted for the remainder of the original warranty period or thirty (30) days, whichever is longer. **These remedies are not available outside of the United States.**

NO LIABILITY FOR CONSEQUENTIAL DAMAGES.

In no event shall CSI or its suppliers be liable for any damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business infor mation, or other pecuniary loss) arising out of the use of or inability to use this CSI product, even if CSI has been advised of the possibility of such damages. Because some states do not allow the exclusion or limitation of liability for conse quential damages, the above limitation may not apply to Licensee.

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT OF THE SOFTWARE, AND IT HAS BEEN THOROUGHLY TESTED AND USED. HOWEVER, EXCEPT AS OTHERWISE SPECIFICALLY PROVIDED HEREIN, NO WARRANTY IS MADE ON ITS ACCURACY OR RELI ABILITY. IT IS THE RESPONSIBILITY OF THE ENGINEER TO VERIFY THE RESULTS OBTAINED FROM THE SOFTWARE. IN THE EVENT THE SOFTWARE IS FOUND TO BE DEFECTIVE, CSI'S ONLY OBLIGATION IS TO REMEDY THE DEFECT. CSI WILL IN NO EVENT HAVE OBLIGATIONS OR LIABILITIES FOR INCIDEN TAL OR CONSEQUENTIAL DAMAGES ASSOCIATED WITH THE USE OF THE SOFTWARE. NO OTHER WARRANTY IS MADE.

U.S. GOVERNMENT RESTRICTED RIGHTS

The Software and documentation are provided with RESTRICTED RIGHTS. Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c)(1) and (2) of the Commercial Computer Software--Restricted Rights at 48 CFR 52.227- 19, as applicable.

GENERAL

This License shall be interpreted by the laws of the State of California. In the event that any portion of this Agreement is invalidated by court or legislation, the remaining portions of the Agreement shall remain in binding effect. In the event of legal action brought by either party, the prevailing party shall be entitled to reimbursement of actual legal fees and related expenses. The Licensee agrees that this Agreement shall inure to the benefit of CSI, its administrators, successors, heirs and assignees. This Agreement may only be changed by mutual written consent.

Should Licensee have any questions concerning this Agreement, or if Licensee desires to contact CSI for any reason, please write: CSI, Consumer Sales and Service, 1995 University Avenue, Berkeley, California 94704.

Questions and Answers about the End User License Agreement

» Can I sell my end user license agreement for SAP2000 to someone else?
» Can I sell or give away old versions of SAP when I acquire an upgrade?
» Can I make a second copy for my personal use?
» Can I rent or lease the Program to someone else?
» When I upgrade from SAP90, do my license rights for SAP2000 change?

In what ways can I use the software over a network?

Can I sell my end user license agreement for SAP2000 to someone else?

No. You may not sell, rent or lease SAP2000 on a temporary or permanent basis.

Can I sell or give away old versions of SAP when I acquire an upgrade?

No. The original SAP90 and the upgrade SAP2000 together are considered as a single software unit. You must retain the old SAP90 version as part of the license agreement. It is a good idea in any case to save the older versions in case you need to go back and run the old data files.

Can I make a second copy for my personal use?

CSI grants to Licensee a **nontransfer able** License to use this Software. Multiple licenses of the Software to same licensee are available at discounted rates. This License limits the use of this copy of the Software to projects of the Licensee. Application of the Software as a service to other engineer ing firms or individuals, or use of the Software by other engineering firms or individuals, is not allowed.

Can I rent or lease the Program to someone else?

Licensee shall not sell, rent, lease, trans fer, network, publish, dis close, display or otherwise make available any portions of the Software or copies thereof to others.

When I upgrade from SAP90, do my license rights for SAP2000 change?

Yes, the license agreement included with the SAP2000 upgrade sets forth the license rights for both the SAP90 and the SAP2000. With every new upgrade product, you will receive a new license agreement. Upgrade versions are treated as part of the whole product.

In what ways can I use the software over a network?

Network version of the software is available. Please contact CSI for pricing. For all other versions you must acquire a hardware protection device and dedicate one individual computer from which you use the software. For multiple computers you must purchase multiple licenses which are available at a discounted rate. A license to use a particular copy of the software may not be shared or used concurrently on different computers.

APPENDIX

Modal Combinations

Complete Quadratic Combination -- a modal combination technique which accounts for modal damping. Same as SRSS if damping is zero.

Square Root of Sum of Squares -- a modal combination technique which does not account for modal damping.

General Modal Combination -- also known as Gupta Method.

Absolute -- arithmetic summation without signs

EIGENVECTORS

Eigenvalue analysis determines the undamped free-vibration mode shapes and frequencies of the system. These natural modes provide an excellent insight into the behavior of the structure.

RITZ

Ritz vectors take into account the spatial distribution of the dynamic loading.

Rigid-body behavior

Rigid-body behavior, in which the constrained joints translate and rotate together as if connected by rigid links. The types of rigid behavior that can be modeled are:

Equal-displacement behavior

Equal-displacement behavior, in which the translations and rotations are equal at the constrained joints.

Symmetry Conditions

For a structure that is symmetric about a plane, symmetric loading causes symmetric displacements as follows:

- Forces and displacements parallel to the plane of symmetry are equal
- Forces and displacements normal to the plane of symmetry are opposite
- Moments and rotations parallel to the plane of symmetry are opposite
- Moments and rotations normal to the plane of symmetry are equal

Anti-symmetry Conditions

For a structure that is symmetric about a plane, anti-symmetric loading causes anti-symmetric displacements. All degrees of freedom that are equal when symmetric are opposite when anti-symmetric, and all degrees of freedom that are opposite when symmetric are equal when anti-symmetric. Thus the specification of the anti-symmetric degrees of freedom simply uses the opposite signs from the symmetric case.

Von Mises Stress

The Von Mises Stress provides a measure of the shear, or distortional, stress in the material. This type of stress tends to cause yielding in metals. It is independent of the amount of hydrostatic stress **(σ1= σ2= σ3)** action on the material.

The Von Mises Stress is identified in terms of the principal stresses as **σvm=$\sqrt{}$1/2[(σ1- σ2)^2+(σ1- σ3)^2+(σ2- σ3)^2].**

In a state of pure tension, say **σ11=σ** and all other stresses are zero, then **σvm=σ**. In a state of pure shear, say **σ12=τ** and all other stresses are zero, then **σvm=$\sqrt{}$3 τ**.

For materials, initial yielding can be expected when **σvm=σy**, where **σy** is the tensile yield stress, or when **σvm=$\sqrt{}$3 τy**, where **τy** is the yield stress in shear. For other materials, particularly frictional materials such as soil and concrete, the Von Mises Stress may have no value in predicting yield or failure.

Analysis Cases

Analysis Cases include:

- Static Load Cases
- Response Spectrum Analysis Cases
- Time History Analysis Cases
- Moving Load Case
- Modal case based on a specific vibration mode shape
- Load Combos located above the present Load Combo in the Combinations list box

Load Combination Type

ADD = All Analysis case results are multiplied by their scale factor and added together. Combo type can be used for static loads.

ENVE = An Max/Min Envelope of the defined Analysis cases is evaluated for each frame output segment and element joints. The Analysis cases that give the Maximum and Minimum components are used for this combo. Therefore the Load Combo holds two values for each output segment and joint. Combo type can be used for moving loads and any Analysis case where the load producing the maximum or minimum force/stress is required.

ABS = The absolute of the individual Analysis case results are summed and positive and negative value are automatically produced for each output segment and joint. Combo type can be used for lateral loads.

SRSS = The Square Root Sum of the Squares calculation is done on the Analysis cases and positive and negative value are automatically produced for each output segment and joint. Combo type can be used for lateral loads.

Linear Time History Analysis

The Time History functions specified are induced on the structure and all elements are taken to be linear.

Periodic Time History Analysis

The Time History functions are assumed to be periodic and the steady state results are displayed taking all elements to be linear.

NonLinear Time History Analysis

The Time History Analysis is performed with nonlinear link elements being allowed to act nonlinearly.

Ground Acceleration

The specified Time History Function will be induced on the structure as a ground acceleration through all support joints.

acc dir 1 = Ground Acceleration in the global X direction.

acc dir 2 = Ground Acceleration in the global Y direction.

acc dir 3 = Ground Acceleration in the global Z direction.

UBC94 Response Spectrum

UBC94S1 Response Spectrum for Soil Type 1.

UBC94S2 Response Spectrum for Soil Type 2.

UBC94S3 Response Spectrum for Soil Type 3

Energy Functions

Input Energy	Work done on structure by force and acceleration functions. In an exact solution this would equal the sum of the Kinetic, Potential, Mdamp, Ndamp and NLLink Energies.
Kinetic Energy	A function of the sums of the masses and their corresponding velocity for the structure.
Potential Energy	A function of the sums of the elastic constants and their corresponding displacements for the structure. The sum includes NLLink members.
MDamp Energy	(Modal Damping Energy) Is the energy absorbed through modal damping.
NDamp Energy	(NLLink Damping Energy) Is the energy absorbed through NLLink damping.
NLLink Energy	(NLLink Hysteretic Energy) Is the energy absorption by Plastic1, Isolator1 and Isolator2 NLLink elements. Damper, Gap and Hook NLLink elements are not included.
Error Energy	The is an estimate of the error in the analysis. It is the difference between the Input energy and the sum of the remaining energy components.

Base Functions

Base Shear: The total shear on all the supports in a given global direction.

Base Moment: The Overturning/Torsional moment on the supports of the structure in a given global direction.

DXF

DXF is a Computer Aided Drafting (CAD) file format used by many CAD programs.

Joint Stress Averaging

None:

This option is used to display contours with no averaging. It is used to see if the model is meshed properly. By selecting this option you can see whether there are large stress variations between elements. This can be an indication that the model is not properly meshed and may need to be refined to properly capture the variation in stress.

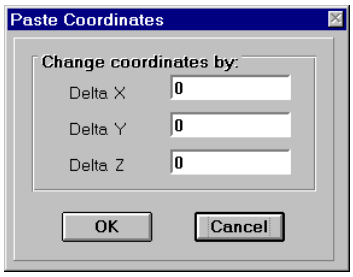
At All Joints:

Once you are confident that the model is acceptable, the stress Averaging will give better results of the stress in the elements by averaging the stress between elements and thus reducing the error due to the stress on each individual element.

At Select Joints:

If you have a discontinuity in the model, like two planes meeting at an angle, then you will need to do the averaging on each plane independently by using this option. This will avoid the problem of averaging across the two planes which would give incorrect result. This is due to the stress along the two planes not being continues relative to the elements

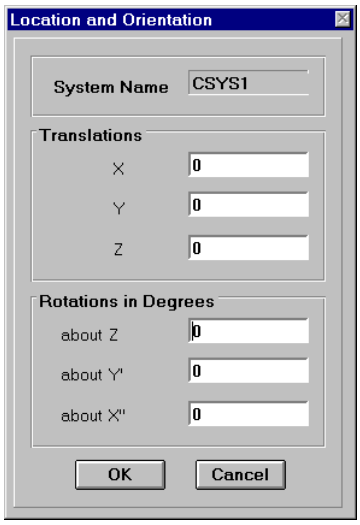
Paste Coordinates



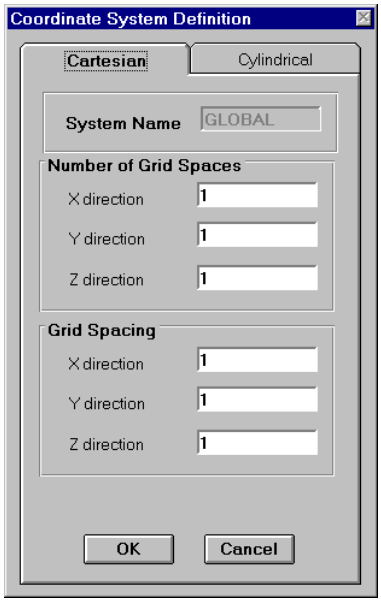
Insertion Point

This dialog box is used to locate the origin of the newly added template model's coordinate system or a newly defined coordinate system relative to the origin of the Global coordinate system. The origin of the newly added model is at the base of the model and centered on its local X-Y plane.

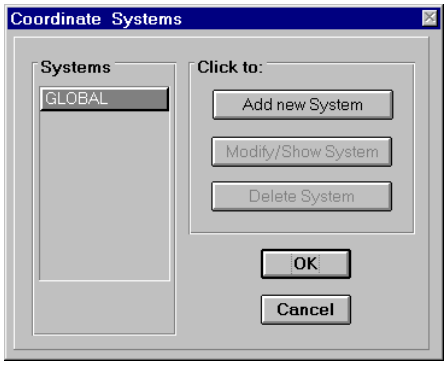
Note: The first coordinate system in the model is the Global coordinate system.



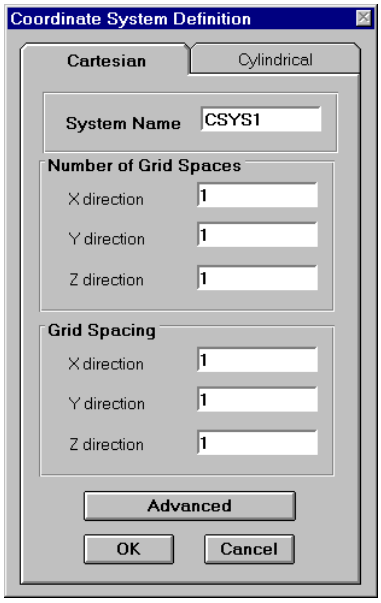
Coordinate System



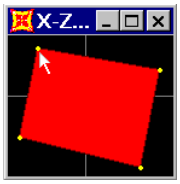
Set Coordinate System



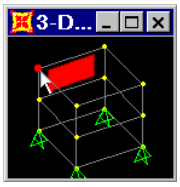
Coordinate System (Advanced)



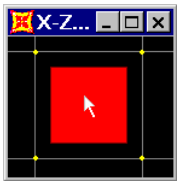
Shell Element (4 Joints)



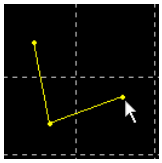
Shell Element (4 Joints 3-D)



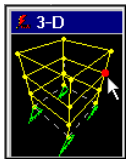
Shell Element (Quick)



Frame Element (2 Joint)



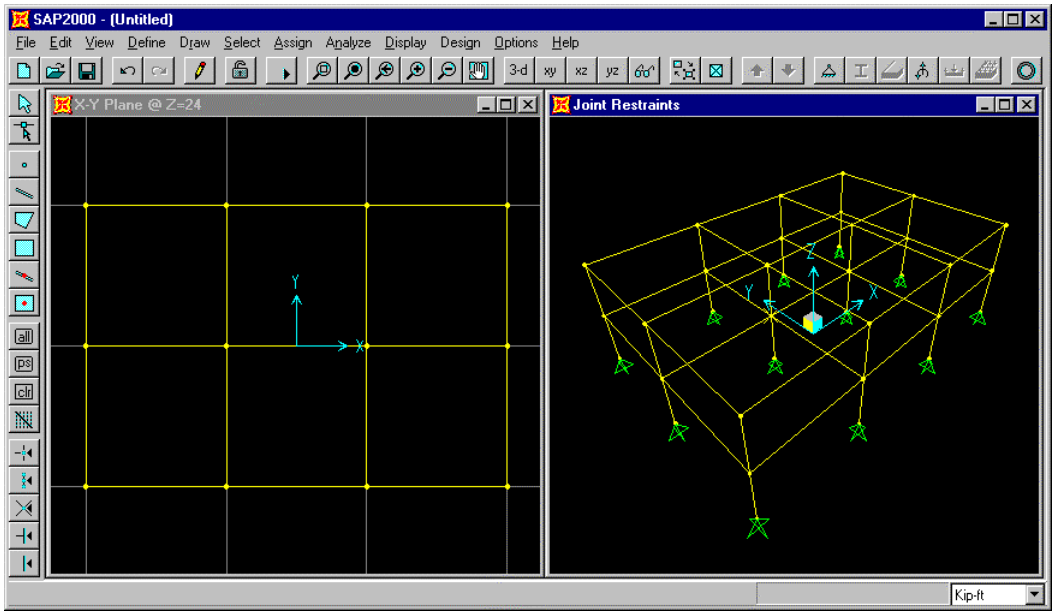
Frame Element (2 Joint 3-D)



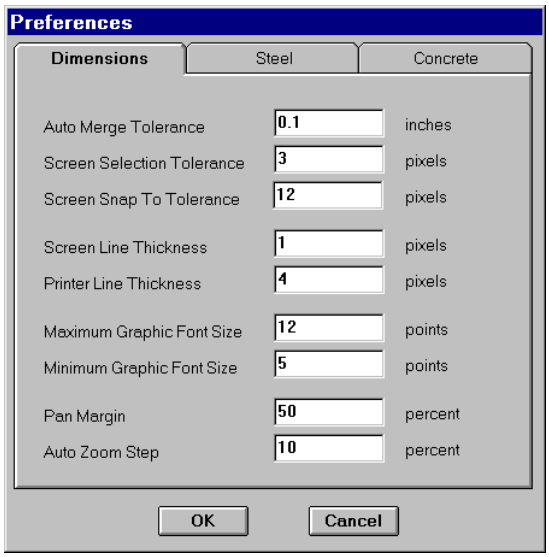
Side Toolbar



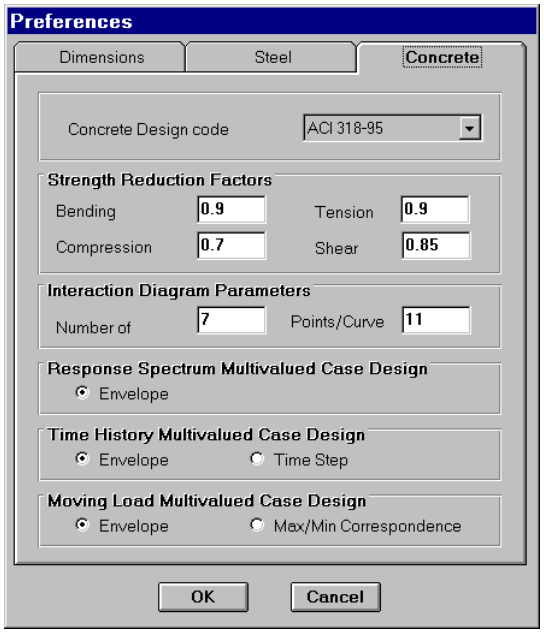
SAP2000 Screen



Dimension Tolerances



Concrete Design Parameters



INDEX

2

2 Modes of SAP2000 13

A

Add a Frame Section 37
Add a New Concrete Material Type 35
Add a New Other Material Type 36
Add a New Steel Material Type 35
Add a Shell Section 39
Add an Auto Select Frame Group 39
Add an NLLink Property 39
Add Special Joints 73
Aerial View 133
Animation 23, 107
Anti-symmetry About a Plane 90
Asolid 15
Asolid Element 15
Assign Dynamic Loads 68
Assign Frame End Offsets 94
Assign Frame Initial P-Delta Force 95
Assign Frame Lanes 96
Assign Frame Output Segments 95
Assign Frame Releases 94
Assign Gravity Loads to Frames 98
Assign Gravity Loads to NLLinks 101
Assign Gravity Loads to Shells 99
Assign Group Names 101
Assign Joint Constraints 85
Assign Joint Masses 92
Assign Joint Patterns 92
Assign Joint Restraints 92
Assign Joint Springs 91
Assign Loads or Displacements to Joints 98
Assign Local Axes for Frame Elements 93
Assign Local Axes for Joints 93
Assign Local Axes for Shell Elements 97
Assign Local Axis For NLLinks 98
Assign Options 84
Assign Point and Uniform Loads to Frames 99

Assign Pressure Loads to Shells 100
Assign Section Properties 84
Assign Static Loads 98
Assign Temperature Loads to Frames 99
Assign Temperature Loads to Shells 100
Assign Trapezoidal Loads to Frames 99
Assign Uniform Loads to Shells 100
Auto Select 39
Automatic Meshing 29
AVI 22
Axes 93, 94, 97
Axisymmetry 90

B

Base Functions 112
Base Shear 112
Beam 38
Break 29
Bridge 61, 62, 66, 67, 105, 106
Bridge Analysis 61

C

Change Labels 31
Color 132
Column 38
Concrete 24, 35, 36, 37, 38, 124, 125, 127, 128, 129, 131, 132
Concrete Axial Hinge 53
Concrete Shear Hinge 55
Connect 30, 31
Constraint 85, 86, 87, 88, 89, 90, 91
Constraints 79, 80
Coordinate System 132
Copy 26
Cut 25, 26
Cyclic symmetry 91

D

Default Hinge Properties 51
Define Bridge Response 67
Define Frame Hinge Properties 51
Define Group Names 60
Define Joint Patterns 68
Define Lanes 61
Define Materials 35
Define Response Spectrum Cases 70
Define Response Spectrum Functions 69
Define Section Properties 36
Define Static Pushover Cases 42
Define Time History Cases 69
Define Time History Functions 68
Define Vehicle Classes 66, 67
Define Vehicles 62
Defining Elements as a Single Member for Bending 126
Deformed Shape 106, 107
Delete 26

Deleting Members 26
Deselect 81
Design 24, 124, 125, 126, 127, 128, 129, 130, 131, 132
Designer's Convention 13
Disconnect 30
Displacement 98
Display Base Function Time History Trace 112
Display Concrete Design/Check Input on Screen 128
Display Concrete Design/Check Output on Screen 127
Display Design Input/Results on Screen 127
Display Frame Element Force Output Time History Trace 110
Display Group Joint Force Sums 116
Display Input Time History Functions 108
Display Joint or Member Text Output on Screen 117
Display Joint Output Time History Trace 109
Display Options 104
Display Shell Element Stress Output Time History Trace 111
Display Steel Stress Check/Design Input on Screen 128
Display Steel Stress Check/Design Ratios on Screen 127
Display Structural Energy Time History Trace 112
Divide 29
Divide or Break Frames 29
Draw 75
Draw a Frame Element from Joint to Joint 74
Draw a Quick Frame Element 74
Draw a Quick Shell Element 75
Draw a Rectangular Shell Element 75
Draw a Shell Element between 4 Joints 74
Draw an NLLink Element 75, 76
Draw Members 72
Draw Mode 72
Duplicates 31
DXF 21, 22
Dynamic 102

E

Edit Grids 76
End User License Agreement for CSI Software 140
Energy Diagrams 107
Energy Functions 112
Enhanced MetaFile 23
Export 20, 21, 22, 23
Export a DXF file 21
Export a SAP2000 text input file 20
Export an AVI file 22

F

Force Diagram 107
frame 95
Frame 14, 29, 30, 36, 37, 38, 39, 73, 74, 82, 84, 93, 94, 95, 96, 97, 99, 104, 110, 117
Frame Element 14
Frame Hinge Interaction Surface 58
Frame Hinge Property Data 56
Frame Hinge Property Data for XXX 57
Frame Hinges (Pushover) 96

Frequently Asked Questions 136

G

General Tips 134
Generalized Constraints 86
Graphical Output 105
Gravity 98, 99, 100, 101
Grid 14, 76
Group 60, 61, 82, 101, 113, 116, 117, 125

I

Import 20, 21
Import a DXF file 21
Import a Frame Section from the Section property file (e.g. Sections.pro) 37
Import a SAP2000 Job Run in DOS mode 22
Import a SAP2000 text input file 20
Import a SAP90 text input file 20
Import/Export Capabilities 20
Influence Lines 117
Input 106
Interaction Diagram 129
Interaction Surface Definition 59, 60
Intersecting Line 82
Intersections 78

J

Join Frames 30
Joint 68, 73, 74, 85, 91, 92, 93, 98, 104, 105, 109, 116, 117, 147
Joint and Element Labels 31

L

Label 31
Lane 62, 105
License 140, 141, 142, 143
Load 61, 69, 70, 71, 97, 98, 99, 100, 101, 105, 125
Load Cases 61
Load Combinations 70, 71, 125
Local Axis 98
Log 25

M

Mass 92
Material 35, 36
Merge Joints 28, 29
Mesh 29, 30
Mesh Shells 30
Mirror 28
Mode Shape 106
Move 26
Moving Loads 67, 68

N

Network 143
New Interface 13

NLLink 39, 40, 75, 76, 82, 98, 101
Nonlinear 16
Nonlinear Analysis 16
Nonlinear Static Pushover Analysis 16
Nonprismatic 38, 39
Noun-Verb 13

O

Output 102
Overwrite Element Design Data 126

P

Partial Connection 89
Paste 25, 26
P-Delta 95, 102, 103
Preferences 131
Pressure 100
prestress 95
Prestress 97
Print 23, 24, 25, 115, 129
Print Design Results to a Printer or a File 24
Print Input Tables to a Printer or a File 24
Print Output to a Printer or a File 24
Print Selected Graphical Output to a Printer or a File 23
Print Setup 23
Print Time History or Response Spectrum Plots or Tables 115
Program Manuals 137
Pushover 16, 17, 42, 43, 51, 56, 58, 59, 60, 96, 97, 118, 119
Pushover Curve 118, 119

R

Radial 27, 28
Redesign a Concrete Member 129
Redesign a Steel Member 129
Redo 26
References 138
Re-Label Previously Assigned Labels 31
Release 94
Replace Auto with Optimal Sections 127
Replicate 27, 28
Replicate by Using the Mirroring Option 28
Replicate in a Linear Array 27
Replicate in a Radial Array 27
Replicate in a Radial Array by Shifting the Origin 28
Reset Design Sections 130
Reshape Element 73
Response Spectrum 69, 70, 115, 116
Restraint 92
Review Concrete Design/Check Detail 128
Review Steel Stress Check/Design Detail 128
Review Stress Check/Design Details 128

S

SAP2000 ..... 20, 135, 137, 138, 142, 143, 151
SAP2000 Screen 151

SAP2000 Versions and Limitations 135
SAP90 20
Scale Factor
  Line Type and Color 114
Select 81
Select 2-D Views 32
Select 3-D Views 32
Select All Objects 83
Select Mode 13
Select Objects by 2D Planes 82
Select Objects by Constraints 83
Select Objects by Frame Sections 82
Select Objects by Groups 82
Select Objects by NLLink Properties 82
Select Objects by Shell Sections 82
Select Objects by Window 81
Selection Procedures 81
Set Elements 33
Set Limits 33
Setting Colors 132
Setting Concrete Design Parameters 131
Setting Steel Design Parameters 131
Setup Coordinate System 14
Shell 15, 30, 36, 39, 73, 74, 75, 82, 84, 85, 97, 100, 104, 111
Shell Element 15
Snap to Lines and Edges 79
Snap to Perpendicular 78
Snap Tools 76, 77, 78, 79
Solid 16
Solid Element 16
Spectrum Generation Axes Tab 115
Spectrum Generation Damping Tab 116
Spectrum Generation Define Tab 115
Spectrum Generation Frequency Tab 116
Spectrum Generation Options Tab 116
Spring 91
Standard Vehicles 63
Start Design/Check of Structure 125
Static Pushover Case Data 42, 43
Steel 24, 35, 124, 127, 128, 129, 131
Steel Axial Hinge 52
Steel Shear Hinge 53
Stress Averaging 147
Stress Diagram 107
Support 135, 136
Symmetry About a Plane 90
Symmetry About a Point 91

T

Temperature 99, 100
Template 19
Text Books 138
Text Editor 25
Text Output 24
Time History 68, 69, 108, 109, 110, 111, 112, 113, 114, 115, 131, 132

U

Undeformed Geometry 105
Undo..... 26
Undo and Redo Capabilities..... 26
Update Analysis Sections 130

V

Vehicle 62, 63, 66, 67
View 23, 32, 33, 104, 105, 108, 115, 129
View Generated Response Spectra Curves 115
View Interaction Diagram for a Concrete Column 129
View Real Time Animation for Time History Results 23
View Time History Results 108

Z

Zoom 33, 114, 129
Zoom into the Plot..... 114