STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

ETABS

version 8

Integrated Building Design Software

USER INTERFACE REFERENCE MANUAL

A Product of Computers & Structures, Inc.

ETABS®

Integrated Building Design Software

User Interface Reference Manual

Computers and Structures, Inc.
Berkeley, California, USA

Version 8.00
January 2002

Copyright

The computer program ETABS 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

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2002.
The CSI Logo is a trademark of Computers and Structures, Inc.
ETABS is a trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.
Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABS. 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 ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Contents

User's Manual

INTRODUCTION

Welcome to ETABS! 1

Highlights of the ETABS Programs 2

Organization of this Manual 3

Getting Help 4

On-Line Help 4

Phone, Fax and E-Mail Technical Support 5

Telephone Support 5

Fax Support 6

Email Support 6

Help Us to Provide You Technical Support 7

1 GRAPHIC USER INTERFACE

Features 1-1

Main Window 1-1

Main Title bar 1-2

Menu Bar 1-2

Toolbars and Buttons 1-3
Display Windows 1-3
Display Title Bar 1-4
Status Bar 1-4
Mouse Pointer Position Coordinates 1-5
Plan View Drawing and Assignments Options (Similar Stories Feature) 1-6
Current Units 1-6
The Aerial View 1-7
Using the Mouse 1-9
Starting a Model 1-10
Two Modes 1-11
Locking and Unlocking a Model 1-12
Undo Features 1-13
Tips for Using the Graphical User Interface 1-14

2 OVERVIEW OF A MODEL

The Concept of Objects 2-2
The Analysis Model 2-4

3 MODELING TIPS

Modeling Process 3-1
Modeling Tips 3-5

4 FILE MENU

General 4-1
New Model Command 4-1
Initialize a New Model 4-2
Creating .edb Files for Initializing Your Models 4-3
Building Plan Grid System and Story Definition Form 4-4

Grid Dimensions (Plan) - Define a Grid System 4-4
Story Dimensions - Define Story Data 4-5
Add Structural Objects 4-5
Steel Deck Button - Steel Floor System Form 4-8
Staggered Truss Button - Staggered Truss Form 4-10
Flat Slab Button - Flat Slab Form 4-12
Flat Slab with Perimeter Beams Button - Flat Slab with Perimeter Beams Form 4-14
Waffle Slab Button - Waffle Slab Form 4-17
Two-Way or Ribbed Slab Button - Ribbed Slab Form 4-21
Grid Only Button 4-24
Open Command 4-24
Backup Files 4-25
Save and Save As Commands 4-25
If a File Becomes Corrupted 4-26
Import Command 4-27
Export Command 4-28
Export Files for Use in CSI Programs 4-28
Export Files for Use in AutoCAD 4-29
Export Files for Use in Access 4-31
Export Files to an Enhanced Metafile 4-31
Export Files to a CIS/2 Step File 4-31
Export Files to a Steel Detailing Neutral File 4-32
Create Video Command 4-32
Time History Animation Command 4-32
Cyclic Animation Command 4-33
Print Setup Command 4-33
Print Preview for Graphics Command 4-34
Print Graphics Command 4-34

Print Tables Command 4-35
Input Command 4-35
Analysis Output Command 4-36
Print Design Tables Commands 4-36
User Comments and Session Log Command 4-37
Last Analysis Run Log Option 4-36
Display Input/Output Text Files Command 4-37
Delete Analysis Files Command 4-37
Exit the Program 4-38

5 EDIT MENU

General 5-1
Undo and Redo Commands 5-1
Cut, Copy and Paste Commands 5-2
Using a Spreadsheet to Create or Modify Model Geometry and Section Properties 5-3
Point Objects 5-3
Line Objects 5-4
Area Objects 5-6
Delete Command 5-8
Add to Model from Template Command 5-8
Add to Model from Template > Add 2D Frame Command 5-9
Add to Model from Template > Add 3D Frame Command 5-9
Replicate Command 5-10
Linear Tab 5-12
Radial Tab 5-12
Mirror Tab 5-13
Story Tab 5-14
Delete Original Check Box 5-15
Edit Grid Data Command 5-15

Edit Grid Data > Edit Cartesian/Cylindrical System Command 5-16
Add New System Button 5-16
Edit Grid Button 5-16
Grid Labels Button 5-19
Modify/Show System Button 5-20
Delete System Button 5-19
Edit Grid Data > Edit General System Command 5-20
Add New System Button 5-20
Add Copy to System Button 5-22
Modify/Show System Button 5-22
Delete System Button 5-24
Edit Grid Data > Convert System to General Command 5-24
Edit Grid Data > Glue Points to Grid Lines Command 5-24

Edit Story Data Command 5-24
Edit Story Data > Edit Command 5-25
Edit Story Data > Insert Story Command 5-26
Edit Story Data > Delete Story Command 5-27

Edit Reference Planes and Edit Reference Lines Commands 5-27

Merging Points Command 5-28

Align Points/Lines/Edges Command 5-29
Align Points/Lines/Edges Form 5-31
Align Options to Selected Objects 5-31
Align to X, Y, or Z Coordinate 5-32
Align to X or Y Grid Lines 5-33
Trim or Extend Selected Lines 5-34
Align Points To 5-37
Align Tolerance 5-37

Move Points/Lines/Areas Command 5-38
Moving Objects in the Z Direction 5-39

Expand/Shrink Areas Command 5-40

Merge Areas Command 5-42
Mesh Areas Command 5-43
Split Area Edges Command 5-44
Join Lines Command 5-44
Divide Lines Command 5-46
Extrude Point to Lines Command and Extrude Lines to Areas Command 5-48
Linear Extrusion 5-50
Radial Extrusion 5-50
Auto Relabel All Command 5-51
Nudge Feature 5-51

6 VIEW MENU

General 6-1
Set 3D View Command 6-1
Set 3D View Form 6-4
Adjust the 3D View 6-5
Set Plan View Command 6-5
Set Elevation View Command 6-6
Perspective Toggle Button 6-8
Perspective Toggle in Plan View 6-8
Perspective Toggle in Elevation View 6-8
Perspective Toggle in Three-Dimensional View 6-9
Set Building View Limits Command 6-9
Set Building View Options Command 6-10
View by Colors of 6-11
Special Effects 6-12
Object Present in View 6-13
Object View Options 6-16
Piers and Spandrels 6-17
Visible in View 6-18
Special Frame Items 6-20

Other Special Items 6-22

Zoom Commands 6-23
Rubber Band Zoom 6-24
Restore Full View 6-24
Previous Zoom 6-25
Zoom in One Step 6-25
Zoom Out One Step 6-25

Pan Feature 6-26

Measure Command 6-26

Change Axes of Location Command 6-27

Show Selection Only and Show All Commands 6-28

Save Custom View and Show Custom View Commands 6-28

Refresh Window and Refresh View Commands 6-28

Show Rendered View 6-29

7 DEFINE MENU

General 7-1

Material Properties Command 7-1
Material Property Data Form 7-2

Frame Sections Command 7-6
Importing Sections from a Database 7-7
Adding User-Defined Frame Section Properties 7-8
Steel Joist Section Properties 7-11
Adding Frame Section Properties Using Section Designer 7-12
Nonprismatic Sections 7-13
Nonprismatic Section Definition Form Button 7-16
Starting and Ending Sections 7-17
Segment Lengths and Types 7-17
Variation of Properties 7-18
Effect on End Offsets Along the Length of Frame Elements 7-19

Reinforcing for Concrete Frame Section Properties 7-19
Reinforcing Information for Columns 7-20
Reinforcing Information for Beams 7-22
Wall/Slab/Deck Sections Command and Form 7-23
Wall/Slab Sections Form 7-23
Deck Sections Form 7-25
Link Properties Command 7-28
Frame Nonlinear Hinge Properties Command 7-29
Groups Command 7-29
Section Cuts Command 7-30
Section Cuts Form 7-30
Response Spectrum Functions Command 7-32
Add Response Spectrum from File Form 7-32
User-Defined Response Spectrum Functions 7-34
Code-Specific Response Spectrum Functions 7-35
1994 UBC Parameters for a Response Spectrum Function 7-36
1997 UBC Parameters for a Response Spectrum Function 7-37
1996 BOCA Parameters for a Response Spectrum Function 7-37
1995 NBCC Parameters for a Response Spectrum Function 7-38
IBC2000 Parameters for a Response Spectrum Function 7-38
1997 NEHRP Parameters for a Response Spectrum Function 7-39
1998 Eurocode 8 Parameters for a Response Spectrum Function 7-39
1992 NZS 4203 Parameters for a Response Spectrum Function 7-39
Modifying and Deleting Response Spectrum Functions 7-40
Time History Functions Command 7-40

Add Functions from File Button 7-41
Add User Function 7-43
Add (Template) Functions - Sine, Cosine, Ramp, Sawtooth, and Triangle 7-44
Sine Time History Function Template Parameters 7-45
Cosine Time History Function Template Parameters 7-46
Ramp Time History Function Template Parameters 7-46
Sawtooth Time History Function Template Parameters 7-47
Triangular Time History Function Parameters 7-48
Add User Periodic Function 7-48
Static Load Cases Command 7-49
Define Static Load Case Form 7-52
Modify an Existing Static Load Case 7-52
Delete an Existing Static Load Case 7-53
Response Spectrum Cases Command 7-53
Response Spectrum Case Data Form 7-53
Spectrum Case Name 7-54
Structural and Function Damping 7-54
Modal Combination 7-55
Directional Combination 7-57
Input Response Spectra 7-58
Excitation Angle 7-59
Time History Cases Command 7-59
Time History Case Data Form 7-59
History Case Name 7-60
Options 7-60
Load Assignments 7-63
Load Combinations Command 7-66
Mass Source Command 7-67

8 DRAW MENU

Select Object Command 8-1

The Similar Stories Feature 8-2

Reshape Object Command 8-3
- Reshaping Area Objects 8-4
- Reshaping Line Objects 8-5
- Reshaping Dimension Lines 8-6
- Reshaping Point Objects 8-6
- Moving/Reshaping Objects in the Z Direction 8-7

Draw Point Objects Command 8-7

Draw Line Objects Command 8-8

Draw Area Objects Command 8-12

Draw Developed Elevation Definition Command 8-16

Draw Dimension Line Command 8-20

Draw Reference Point Command 8-21

Snap To Command 8-21

9 SELECT MENU

General 9-1

Basic Methods of Selecting Objects 9-1

Menu Methods of Selecting Objects 9-4

Deselecting Objects 9-6

Get Previous Selection 9-7

Clear Previous Selection 9-7

10 ASSIGN MENU

General 10-1

Assign Joint/Point Commands 10-1
- Joint/Point > Rigid Diaphragm Command 10-2
- Joint/Point > Panel Zone Command 10-3

Properties 10-4
Connectivity 10-6
Local Axis 10-7
Options 10-8
Joint/Point > Restraints (Supports) Command 10-9
Joint/Point > Point Springs Command 10-10
Coupled Springs 10-11
Joint/Point > Link Properties Command 10-12
Joint/Point > Additional Point Mass Command 10-13

Frame/Line Commands 10-15
Frame/Line > Frame Section Command 10-16
Frame/Line > Frame Release/Partial Fixity Command 10-16
Unstable End Release 10-17
Frame/Line > End (Length) Offsets Command 10-17
End (Length) Offsets 10-18
Automatically Calculated End Offset Lengths 10-18
Rigid Zone Factor 10-19
Frame/Line > Insertion Point Command 10-20
Frame/Line > Frame Output Stations Command 10-23
Frame/Line > Local Axes Command 10-24
Frame/Line > Frame Property Modifiers Command 10-26
Frame/Line > Frame Line Type Command 10-27
Frame/Line > Link Properties Command 10-27
Frame/Line > Frame NonLinear Hinges Command 10-28
Frame/Line > Pier Label Command 10-29
Frame/Line > Spandrel Label Command 10-30
Frame/Line > Line Springs Command 10-32
Frame/Line > Additional Line Mass Command 10-33
Frame/Line > Automatic Frame Subdivide Command 10-34
Frame/Line > Use Line for Floor Meshing Command 10-35

Shell/Area Commands 10-36
Shell/Area > Wall/Slab/Deck Section Command 10-36
Shell/Area > Openings Command 10-37
Shell/Area > Rigid Diaphragm Command 10-37
Shell/Area > Local Axes Command 10-39
Shell/Area > Shell Stiffness Modifiers Command 10-39
Shell/Area > Pier Label Command 10-40
Shell/Area > Spandrel Label Command 10-41
Shell/Area > Area Springs Command 10-41
Shell/Area > Additional Area Mass Command 10-44
Load Commands 10-44
Joint/Point Loads > Force Command 10-44
Joint/Point Loads > Ground Displacement Command 10-46
Joint/Point Loads > Temperature Command 10-48
Frame/Line Loads > Point Command 10-49
Frame/Line Loads > Distributed Command 10-51
Frame/Line Loads > Temperature Command 10-55
Shell/Area Loads > Uniform Command 10-58
Shell/Area Loads > Temperature Command 10-60
Shell/Area Loads > Wind Pressure Coefficient 10-62
Group Names Command 10-62
Clear Display of Assigns Command 10-64
Copy Assigns 10-64
Paste Assigns 10-65

11 ANALYZE MENU

Set Analysis Options Command 11-1
Building Active Degrees of Freedom 11-1
Set Dynamic Parameters Button 11-3
Set P-Delta Parameters Button 11-5
Save Access DB File Button 11-8

Run Analysis Command 11-8
Analysis Window 11-9
Calculate Diaphragm Centers of Rigidity Command 11-10

12 DISPLAY MENU

General 12-1
Show Undeformed Shape Command 12-1
Show Loads Command 12-2
Show Loads > Joint/Point Command 12-2
Show Loads > Frame/Line Command 12-3
Show Loads > Shell/Area Command 12-5
Set Input Table Mode Command 12-6
Selection Only Check Box 12-7
Select Loads Button 12-7
Check/Uncheck All Button 12-7
Show Deformed Shape Command 12-8
Show Mode Shape Command 12-13
Show Member Forces/Stress Diagram Command 12-14
Show Member Forces/Stress Diagram > Support/Spring Reactions Command 12-15
Show Member Forces/Stress Diagram > Frame/Pier/Spandrel Forces Command 12-18
Show Member Forces/Stress Diagram > Shell Stresses/Forces Command 12-21
Miscellaneous Notes about Shell Element Forces and Stresses 12-26
Show Member Forces/Stress Diagram > Link Forces Command 12-27
Show Energy/Virtual Work Diagram Command 12-27
Show Response Spectrum Curves Command 12-30
Define Tab 12-31
Axes Tab 12-31

Options Tab 12-32
Frequency/Period Tab 12-33
Damping Tab 12-34
Show Time History Traces Command 12-35
Set Output Table Mode Command 12-40
Select Loads Button 12-40
Section Cuts Command 12-40

13 DESIGN MENU

Overview 13-1
Background 13-2
Default Design Procedure Assignments 13-2
Steel Frame Design Command 13-3
Steel Frame Design > Select Design Group Command 13-4
Steel Frame Design > Select Design Combo Command 13-4
Steel Frame Design > View/Revise Overwrites Command 13-5
Steel Frame Design > Set Lateral Displacement Targets Command 13-5
Steel Frame Design > Start Design/Check of Structure Command 13-7
Steel Frame Design > Interactive Steel Frame Design Command 13-7
Steel Frame Design > Display Design Info Command 13-7
Steel Frame Design > Make Auto Select Section Null Command 13-8
Steel Frame Design > Change Design Section Command 13-8
Steel Frame Design > Reset Design Section to Last Analysis Command 13-9
Steel Frame Design > Verify Analysis vs Design Section Command 13-9

Steel Frame Design > Verify all Members Passed Command 13-10
Steel Frame Design > Reset All Steel Overwrites Commands 13-10
Steel Frame Design > Delete Steel Design Results Command 13-10

Concrete Frame Design Command 13-11
Concrete Frame Design > Select Design Combo Command 13-11
Concrete Frame Design > View/Revise Overwrites Command 13-12
Concrete Frame Design > Start Design/Check of Structure Command 13-12
Concrete Frame Design > Interactive Concrete Frame Design Command 13-13
Concrete Frame Design > Display Design Info 13-13
Concrete Frame Design > Change Design Section 13-13
Concrete Frame Design > Reset Design Section to Last Analysis Command 13-14
Concrete Frame Design > Verify Analysis vs Design Section Command 13-14
Concrete Frame Design > Reset All Concrete Overwrites Command 13-15
Concrete Frame Design > Delete Concrete Design Results Command 13-15

Composite Beam Design Command 13-16
Composite Beam Design > Select Design Group Command 13-16
Composite Beam Design > Select Design Combo Command 13-16
Composite Beam Design > View/Revise Overwrites Command 13-18
Composite Beam Design > Start Design Using Similarity Command 13-18
Composite Beam Design > Start Design Without Similarity Command 13-19

Composite Beam Design > Interactive Composite Beam Design Command 13-20
Composite Beam Design > Display Design Info Command 13-20
Composite Beam Design > Make Auto Select Section Null Command 13-20
Composite Beam Design > Change Design Section Command 13-21
Composite Beam Design > Reset Design Section to Last Analysis Command 13-21
Composite Beam Design > Verify Analysis vs Design Section Command 13-22
Composite Beam Design > Verify All Members Passed Command 13-22
Composite Beam Design > Reset All Composite Beam Overwrites Command 13-23
Composite Beam Design > Delete Composite Beam Design Results Command 13-23
Steel Joist Design Command 13-23
Steel Joist Design > Select Design Group Command 13-23
Steel Joist Design > Select Design Combo Command 13-24
Steel Joist Design > View/Revise Overwrites Command 13-24
Steel Joist Design > Start Design Using Similarity Command 13-25
Steel Joist Design > Start Design Without Similarity Command 13-26
Steel Joist Design > Interactive Steel Joist Design Command 13-26
Steel Joist Design > Display Design Info Command 13-26
Steel Joist Design > Make Auto Select Section Null Command 13-27
Steel Joist Design > Change Design Section Command 13-27

Steel Joist Design > Verify Analysis vs Design Section Command 13-28
Steel Joist Design > Verify All Members Passed Command 13-28
Steel Joist Design > Reset All Steel Joist Overwrites Command 13-29
Steel Joist Design > Delete Steel Joist Design Results Command 13-29
Shear Wall Design Command 13-29
Shear Wall Design > Select Design Combo Command 13-29
Shear Wall Design > View/Revise Pier Overwrites Command 13-30
Shear Wall Design > View/Revise Spandrel Overwrites Command 13-30
Shear Wall Design > Define General Pier Sections Command 13-31
Shear Wall Design > Assign Pier Sections Type Command 13-31
Shear Wall Design > Start Design/Check of Structure Command 13-31
Shear Wall Design > Interactive Wall Design Command 13-32
Shear Wall Design > Display Design Info Command 13-32
Shear Wall Design > Reset All Pier/Spandrel Overwrites 13-32
Shear Wall Design > Delete Wall Design Results Command 13-32
Overwrite Frame Design Procedure Command 13-33

14 OPTIONS MENU

General 14-1
Preferences Command 14-1
Preferences > Dimensions/Tolerance Command 14-2
Preferences > Output Decimals Command 14-5

Preferences > Steel Frame Design Command 14-7
Preferences > Concrete Frame Design Command 14-7
Preferences > Composite Beam Design Command 14-8
Preferences > Shear Wall Design Command 14-8
Preferences > Reinforcement Bar Sizes Command 14-8
Reinforcing Bar Sizes Form 14-9
Preferences > Live Load Reduction Command 14-10
Method Area of the Live Load Reduction Factor Form 14-10
Minimum Factor Area of the Live Load Reduction Factor Form 14-12
Application Area of the Live Load Reduction Factor Form 14-13
Application to Columns Area of the Live Load Reductions Factor Form 14-13
Colors Command 14-14
Colors > Display Command 14-14
Colors > Output Command 14-16
Other Option Menu Commands 14-19
Windows Command 14-20
Show Tips at Startup Command 14-20
Show Bounding Plane Command 14-20
Moment Diagrams on Tension Side Command 14-21
Sound Command 14-21
Lock Model Command 14-22
Show Aerial View Window Command 14-22
Show Floating Property Window Command 14-22
Crosshairs Command 14-23
Reset Toolbars Command 14-23

REFERENCES

Welcome to ETABS!

ETABS is a special purpose computer program developed specifically for building systems. The concept of special purpose programs for building type structures was introduced more than 35 years ago [R. W. Clough, et al., 1963]. However, the need for special purpose programs, such as ETABS, has never been more evident as Structural Engineers put nonlinear static and dynamic analysis into practice and use the greater computer power available today to create larger, more complex analytical models.

With ETABS, creating and modifying a model, executing the analysis, design, and optimizing the design are all done through a single interface that is completely integrated within Microsoft Windows. Graphical displays of the results, including real-time display of time-history displacements, are easily produced. Printed output, to a printer or to a file, for selected elements or for all elements, is also easily produced. This program provides a quantum leap forward in the way models are created, modified, analyzed and designed.

The analytical capabilities of ETABS are just as powerful, representing the latest research in numerical techniques and solution algorithms.

Note:

Both ETABS Plus and ETABS Nonlinear have no limits set on the allowable number of joints and/or equations.

ETABS is available in two versions, ETABS Plus and ETABS Nonlinear. Both versions are comprised of the following modules integrated into and controlled by a single Windows-based graphical user interface:

- Drafting module for model generation.
- Seismic and wind load generation module.
- Gravity load distribution module for the distribution of vertical loads to columns and beams when plate bending floor elements are not provided as a part of the floor system.
- Output display and report generation module.
- Steel frame design module (column, beam and brace).
- Concrete frame design module (column and beam).
- Composite beam design module.
- Shear wall design module.

Note:

All of the ETABS modules are integrated into a single, user-friendly graphical user interface.

ETABS Plus also includes the finite-element-based *linear* static and dynamic analysis module, while **ETABS Nonlinear** includes the finite-element-based *nonlinear* static and dynamic analysis module.

Highlights of the ETABS Programs

The ETABS programs were the first to take into account the unique properties inherent in a mathematical model of a building, allowing a computer representation to be constructed in the same fashion as a real building: floor by floor, story by story. The terminology used in this program is column, beam, brace, and wall, rather than nodes and finite elements.

For buildings, ETABS provides automation and specialized options to make the process of model creation, analysis, and design fast and convenient. ETABS provides tools for laying out floor framing, columns, frames and walls, in either concrete or steel, as well as techniques for quickly generating gravity and lateral loads. Seismic and wind loads are generated automatically according to the requirements of the selected building code. All of these modeling and analysis options are completely integrated with a wide range of steel and concrete design features.

While easy to use, ETABS offers sophisticated analytical and design capabilities. Full dynamic analysis is provided, including nonlinear time-history capabilities for seismic base isolation and viscous dampers, along with static nonlinear pushover features. You can use powerful features to select and optimize vertical framing members as well as identify key elements for lateral drift control during the design cycle. In addition, the transfer of data between analysis and design programs is eliminated because ETABS accomplishes both tasks. This design integration, combined with the ETABS capability to generate CAD output files, means that production drawings can be generated faster and with greater accuracy.

Organization of This Manual

This manual begins with general information about how to use the program. This is followed by descriptions of the menu items, interspersed with insights with respect to using the program to maximum advantage.

- **Chapter 1** describes the components of the graphical user interface
- **Chapters 2 and 3** provide a general overview of ETABS and information helpful to creating models, respectively.
- **Chapters 4 through 14** describe the program's primary menu items and their associated submenus, forms, and command buttons. In most cases, chapter headings that correspond to the

menu items include the word command. Chapter headings that do not include the word command are submenus, forms, command buttons, or general topics provided for informational purposes.

Getting Help

Help with the program is available "on-line" within in the program's Help menu, or through phone, fax and e-mail technical support.

On-Line Help

The program includes extensive on-line help that is available any time the graphical user interface is open. Access on-line help by clicking on the **Help** menu and selecting **Search For Help On…**, or by pressing the F1 function key on the keyboard. If you press the F1 key while a form is open, context-sensitive help related to that form will be provided.

Shortcut:

From within the graphical user interface, activate on-line help at any time by pressing the F1 function key on the keyboard.

On-line help is focused to assist you with the graphic interface. It can guide you through the process of entering data into the various forms used in the program. On-line help also makes the meaning of the data you enter into the forms more clear. For example, if you want to find out how to assign gravity load to a beam in the graphic interface, you can find the answer in the on-line help. If you want to find out what a scale factor used in a response spectrum load case actually scales, you can find it in the on-line help.

The on-line help also includes a Documentation library. The Documentation is provided in *.pdf format that can be viewed or printed using Adobe Acrobat Reader.

Phone, Fax and E-Mail Technical Support

Free technical support is available from CSI via phone, fax or e-mail for 90 days after the software has been purchased. Technical support is available after 90 days if you have a current maintenance agreement with CSI. Maintenance agreements also provide for free or reduced cost upgrades to the program. Please call CSI to inquire about a maintenance agreement.

Technical support is provided only according to the terms of the Software License Agreement that comes with the program.

If you are experiencing problems with the software, please:

- Consult the documentation and other printed information included with your product.
- Check the on-line help.
- If you cannot find a solution, contact us as described herein.

Telephone Support

Standard phone 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.

Note:

Our phone number is (510) 845-2177

You can contact CSI's office via phone at (510) 845-2177. When you call, please, if possible, be at your computer and have your program manuals at hand.

Note that sometimes when you call us with a technical support question, we will request that you e-mail us your input file addressed to support@csiberkeley.com so that we can better understand and determine the cause of your problem.

Fax Support

Note:

Our fax number is (510) 845-4096

You can fax CSI twenty-four hours a day at (510) 845-4096. Structural engineers are available to review and respond to your fax between 8:30 A.M. and 5:00 P.M., Pacific Time, Monday through Friday, excluding traditional U.S. holidays.

When you send a fax with questions about your model, please include a picture of your model if possible. This is often a considerable help to us in understanding your question.

When you send a fax, please be certain that you have provided us with your fax number so that we have somewhere to send our response. If your fax number is in your company letterhead in a relatively small font, it is helpful if you repeat the fax number in the body of your fax because often the small fax numbers in company letterheads are difficult to read or completely indecipherable when we receive the fax.

Note that it is generally more efficient for you to email your entire model (*.edb and/or *.$et and/or *.e2k input file) than to fax us pictures or descriptions of it.

E-Mail Support

You can e-mail CSI for technical support twenty-four hours a day at support@csiberkeley.com. Structural engineers are available to review and respond to your e-mail between 8:30 A.M. and 4:30 P.M., Pacific Time, Monday through Friday, excluding traditional U.S. holidays.

Note:

Our e-mail address for technical support is support@ csiberkeley.com

If your question is about a specific model, it is always helpful and sometimes necessary for you to include your model (*.edb and/or *.$et and/or *.e2k input file) as an attachment to your e-mail. When you send us a model as part of a technical support question, we will not reveal that model to anyone outside the company or use it in any advertising without first requesting and obtaining your permission in writing. Note that many of the models used in our advertising are actual models created by our customers.

Help Us to Provide You Technical Support

CSI takes pride in providing timely and effective technical support. If you send us a one word e-mail that says "Help," we will most certainly respond, perhaps with an equally wordy response such as "How?", but more likely with a response that asks you to provide us with some or all of the information listed in the bulleted items that follow. We recognize that much of the time engineers requesting help are under tight deadline pressure and thus would like to receive answers to their questions as quickly as possible. In light of that, whenever you contact us with a technical support question, if you provide us with all or even some of the following information, as appropriate to your circumstances, we will be able to serve you better and faster.

Shortcut:

For faster and more effective service when requesting technical support, please provide us with as much of the information listed in adjacent bulleted items as possible.

- The name and the version number of the program you are using.
- If you are faxing us a description of your model, include a picture of the model, if possible. E-mailing the entire model (*.edb and/or *.$et and/or *.e2k input file) is generally more efficient than faxing a description of it.
- A description of what problem occurred and what you were doing when the problem occurred.
- The exact wording of any error messages that appeared on your screen.
- Your computer configuration (make and model, processor, operating system, hard disk size and RAM size).
- Your name, your company's name and how we may contact you (e.g., your phone number or e-mail address).

Chapter 1

The Graphical User Interface

The graphical user interface is shown in Figure 1-1. Several important features of the interface are shown. Those features include the main window, main title bar, display title bar, menu bar, toolbars, display windows, status bar, mouse pointer position coordinates and the current units. Each of these items is described in this chapter.

Features

Main Window

The main window contains the entire graphical user interface. This window may be moved, resized, maximized, minimized, or closed using standard Windows operations. Refer to Windows help, available on the Start menu, for additional information on those items.

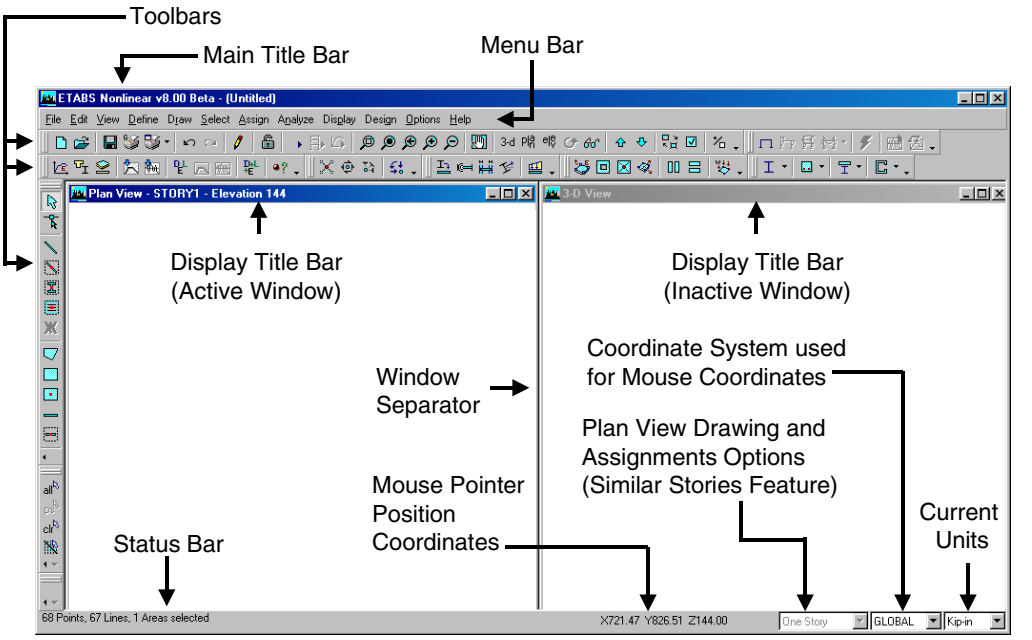


Figure 1-1:
The ETABS graphical user interface

Main Title Bar

The main title bar, located at the top of the main window, includes the program name and the model name. The main title bar is highlighted when the program is in use. You can move the main window by left clicking in the main title bar and holding down the mouse button as you drag the window around your screen.

Menu Bar

The menu bar contains all of the program's menus, which activate available operations. To access a menu, left click on it. This will activate a drop-down menu of commands. The individual commands are described in other chapters of this manual.

Notice that some of the commands on the menus have three dots after them, **...**, and others have a filled triangular section adjacent to the right-hand margin of the menu, ▸. The three dots indicate that a form will appear when you click on the menu item. The

Tip:

Some menu items can be accessed using shortcut keystrokes, which are displayed on the drop-down menu.

triangle indicates that a submenu will appear when you click on the menu item. The commands with neither of these items execute as soon as you click them. Those commands have no submenus or forms.

Toolbars and Buttons

Toolbars are comprised of buttons. For example and are familiar buttons for open and save commands. Virtually all menu commands have a corresponding toolbar button.

Toolbar buttons provide "one-click" access to commonly used commands, particularly file, viewing, and analysis output display options. To execute an operation/command, left click on the appropriate toolbar button.

Tip:

Hold your mouse pointer over a toolbar button for a few seconds and a pop up box describing the button's function will appear.

Holding the mouse pointer over a toolbar button for a few seconds without clicking or holding down any mouse buttons will display a short description of the button's function in a small text box.

The toolbars are customizable, which means you can show or hide toolbar buttons as well as develop a toolbar of buttons that fit your specific needs. To customize a toolbar, right click on an existing toolbar. A form will appear that lists the toolbar buttons and provides you with the options you need to select or deselect buttons as well as "drag and drop" buttons to your custom-designed toolbar.

Display Windows

A display window show the geometry of the model and may also include display of properties, loading and analysis or design results. Up to four windows may display at any one time.

Each display window may have its own view orientation, type of display, and display options. For example, the undeformed shape could be displayed in one window, applied loads in another, an animated deformed shape in a third, and design stress ratios in

the fourth window. Alternatively, the windows may provide four different views of the undeformed shape or other type of display, such as a plan view, two elevations, and a three-dimensional perspective view.

Shortcut:

Clicking the "X" in the upper right-hand corner will close the display window.

Only one display window is active at a time. Viewing and displaying actions only affect the active window. To make a display window active, click on its display title bar or anywhere within the window. A highlighted title bar indicates that the display window is active.

After performing certain operations, the display window may need to be redrawn. Normally the program automatically redraws the window. However, on some occasions, such as after you have deleted some elements, the display may need to be manually refreshed. Use the **View menu > Refresh Window** command, or the **Refresh Window** toolbar icon to redraw the window.

Close a display window by left clicking the "X" button at the top of the window to the right of the display title bar. At least one display window must be open or the program will close.

Display Title Bar

The display title bar is located at the top of the display window. The display title bar is highlighted when the associated display window is active. The text in the display title bar typically includes the type and location of the view in the associated display window. If you are displaying results on the model, the title bar typically also tells you which results are being displayed.

Status Bar

The status bar is located at the bottom of the main window. Text describing the current status of the program is displayed on the left side of the status bar. Most of the time this text provides information about the type and location of the view in the active display window. When you are displaying results on screen, the

Tip:

The status bar provides useful information and messages.

text may tell you what you can do. For example, when the deformed shape is displayed, the status bar text prompts you to "Right click on any point for displacement values."

The right side of the status bar includes the mouse pointer position coordinates and the associated coordinate system, a drop-down box with options for plan view drawing and assignment options for the similar stories feature (only available when you are in plan view), and a drop-down box for setting the current units.

When you are displaying deformed shapes, including mode shapes, animation controls are also available on the right-hand side of the status bar. When displaying element forces for a particular load case, arrow buttons are available on the right-hand side of the status bar that allow you to step the display forward and backward through the available load cases.

Mouse Pointer Position Coordinates

The mouse pointer position coordinates are displayed on the right-hand side of the status bar. The coordinates displayed here are always in the coordinate system specified in the drop-down box on the right-hand side of the status bar just to the left of the current units drop-down box. Note that the **Edit menu > Edit Grid Data** command can be used to define an alternate coordinate system.

A window does not need to be active for the mouse pointer position coordinates to be displayed. It is only necessary that the mouse pointer be over the window.

In a two-dimensional plan or elevation view, the mouse pointer position coordinates are always displayed. In a three-dimensional view, the mouse pointer position coordinates are only displayed when the mouse pointer snaps to a point or a grid line intersection.

Plan View Drawing and Assignment Options (Similar Stories Feature)

When drawing or making assignments while working in a plan view, the Plan View Drawing and Assignment Options drop-down box on the right side of the status bar controls the action. The three options in that drop-down box are as follows:

- **One Story:** An object is applied only to the story level on which it was drawn. An assignment is made only to the selected elements.
- **All Stories:** An object drawn in the plan view is applied to all story levels in the model at the same plan location. An assignment is made to the selected elements and to all other elements in the same plan location at all other story levels.
- **Similar Stories:** An object drawn in plan view is applied to all similar story levels in the model at the same plan location. An assignment is made to the selected elements and to all other elements in the same plan location at all similar story levels.

  Similar stories are specified in the Story Data form, which can be accessed by clicking the **Edit menu > Edit Story Data > Edit** command.

The three options apply only at the time the object is drawn or the assignment is made. The options do not apply retroactively. For example, if you draw an element at one story level ***before*** you change the Plan View Drawing and Assignment option to "All Stories," the program will apply the element only to the story level on which it was drawn, not to all stories.

Current Units

The current units are displayed in a drop-down box located on the far right-hand side of the status bar. Change the current units at any time by clicking on the drop-down box and selecting a

new set of units. The units can also be changed using drop-down boxes located inside some of the forms.

The Aerial View

The aerial view is a small window that can float over the main window. If the main window is not fully maximized, the aerial view can float outside the main window. An example of the aerial view is shown in Figure 1-2. Toggle the aerial view on or off using the **Options menu > Show Aerial View** command.

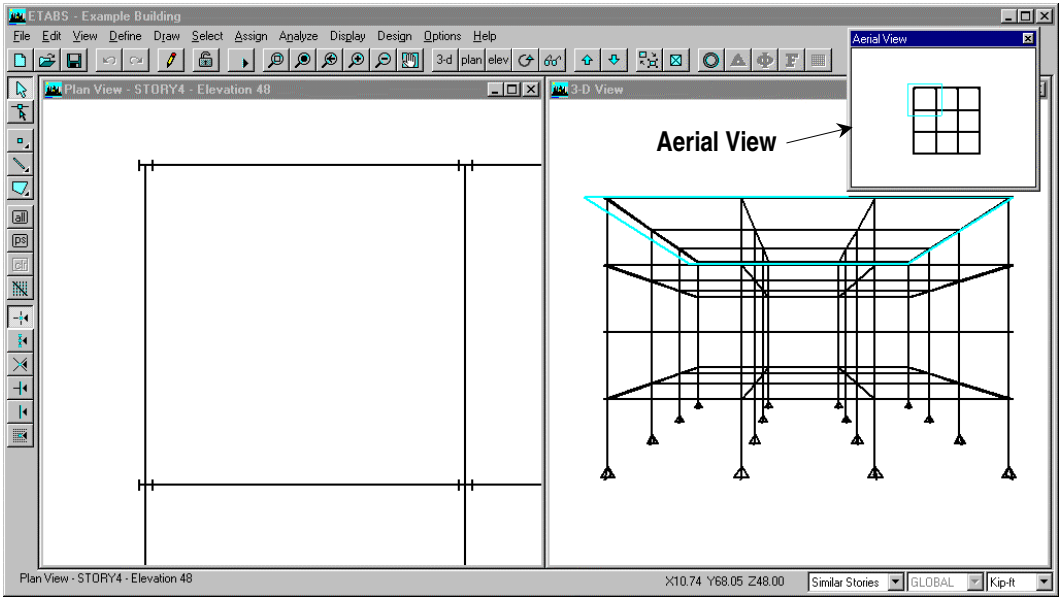


Figure 1-2:
Example of the aerial view

The aerial view window displays the entire drawing to help you move around the active window of a larger model and use the zoom feature to view smaller areas more easily. Also use the aerial view to track which part of the model is displayed in the active window. For example, assume that the active window is a zoom of a small area of story level 4. In that case, the aerial view will show the full view of story level 4, with a bounding box

outlining the portion of story level 4 that is displayed in the active window.

Each time the model is edited, the aerial view is updated.

To zoom into any area in the active display window, draw a bounding box in the aerial view window to specify the area for zooming. To draw the bounding box, put the mouse pointer in the aerial view at one corner of the box that you want to draw, then click the left mouse button and hold it down while you drag the mouse to the diagonally opposite corner of your box; release the mouse button. You will see the outline of the bounding box as you drag the mouse.

Tip:

If the aerial view window is not needed, click the "X" in the upper right-hand corner of the aerial view window to close it. Redisplay it using the Options menu.

Notice that in some instances, the bounding box will change shape after it has been drawn. This occurs because the program automatically adjusts the aspect ratio of the bounding box to match the aspect ratio of the active display window. When changing the aspect ratio of the bounding box, the program always maintains the longer dimension and changes only the smaller dimension of the box.

To move the bounding box to a new location in the aerial view window, click the right mouse button inside the bounding box and hold the button down as you move to the desired location. Release the right mouse button and the display in the active window will be updated accordingly.

Left clicking once in the aerial view window restores the full view in both the active and aerial view windows and eliminates the bounding box.

If you pan the view in the active display window, the bounding box in the aerial view will move also. To pan the view in the active display menu, choose the **Pan** feature from the View menu, or click the Pan button on the toolbar and then left click in the display window and hold down the mouse button while you drag the mouse.

Using the Mouse

The seven mouse actions are (1) left click, (2) right click, (3) hold down the Ctrl key on the keyboard and left click, (4) hold down the Ctrl key on the keyboard and right click, (5) hold down the Shift key on the keyboard and left click, (6) double click, and (7) drag.

Tip:

"To click on" something means to position the mouse pointer over that something and click the left mouse button.

Left Click. Left click means to press down the left button on the mouse and release it. In general, left click to select menu items, activate toolbar buttons, and select objects in the model. If the documentation reads "click on" an item, it means to left click. The documentation will specify "right click" when appropriate.

Right Click. Right clicking means to press down the right button on your mouse and release it. In general, right click on objects in the model to display their assignments.

Ctrl Key. Sometimes you may have objects located one on top of another in a model. In that case, if you want to select a specific object in the model, hold down the Ctrl key while left clicking once on the objects. The left click will bring up a form from which you can choose the object that you want to select. If you want to see the assignments for one of the objects, hold down the Ctrl key while right clicking once on the objects. This will again bring up a form from which you can choose the object whose assignments you want to see.

Within forms, use of the Ctrl key and the left mouse button will select multiple items from a list box. Hold down the Ctrl key and left click on an item to add that item to your selection. The selected items do not need to be adjacent to one another. This feature is only available in the list boxes where multiple selections are possible. An example of this type of list box will be displayed if you click **Display menu > Set Output Table Mode** and then click the **Select Loads** button.

Shift Key. Also within forms, to select multiple adjacent items from a list box, left click on the first item in the list box, hold down the Shift key, and left click on the last item of the list. This

will select the two items that you clicked, plus all of the items in between. This feature is only available in the list boxes where multiple selections are possible.

Double Click. Double click means to click the left mouse button twice in quick succession. Be careful not to move your mouse while you are double clicking. Double clicking is used as one method of completing some drawing operations.

Drag. Drag the mouse by clicking the left mouse button and holding it down while you slide the mouse to another location before releasing the button. Drawing a bounding box in the aerial view window to zoom in on a model is one example of the mouse dragging action. Another example is use of the **Draw menu > Reshape Object** command.

Starting a Model

Note:

Refer to Chapter 3 for more information on creating a model.

Start the graphical user interface by selecting the program from the Windows Start menu or by clicking a shortcut on your desktop. After the graphical user interface has started, start a new model by selecting **File menu > New Model**. Refer to Chapter 3 for more information on the modeling process. Refer to the section entitled "Starting a New Model" in Chapter 4 for more information on starting a new model.

After a model has been started, if needed, get additional help by clicking the **Help menu > Search for Help On** command. Alternatively, press the F1 key on your keyboard at any time to access online help. If you are in a form when you press the F1 key, context sensitive help for that form will automatically appear.

Save your model often using the **File menu > Save** command or the **Save** button on the toolbar. No AutoSave feature exists in this program so it is up to you to save your file on a regular basis. Also, occasionally copy backup copies of your input file to another location for safekeeping. The files to copy are the *.edb file, which is your input file in a binary format (edb is short for

ETABS database), and the *.$et or *.e2k file, which is a text backup file of your input data.

Two Modes

The two distinct modes in this program are the ***draw*** mode and the ***select*** mode. The draw mode allows you to draw objects. The select mode allows you to select objects and is used for editing operations, making assignments to objects, and viewing or printing results. By default, the program is in select mode.

The draw mode automatically enables when you select one of the submenu options of the following commands from the Draw menu or click on the corresponding button(s) on the toolbar:

- Draw Point Objects
- Draw Line Objects , or
- Draw Area Objects , or
- Draw Developed Elevation Definition
- Draw Dimension Line
- Draw Reference Point

The draw mode remains enabled until you do one of the following to return to the select mode:

- Click the Pointer button on the toolbar .
- Press the Esc key on the keyboard.
- Select a command from either the Select menu or the Display menu.

The mouse pointer indicates which mode is enabled. The mouse pointer is defined by the mouse pointer properties identi-

fied in the Windows Control Panel for the Normal Select Pointer and the Alternate Select pointer.

In select mode, the pointer is the Normal Select Pointer, and if you are using the default settings, the mouse pointer will look like this .

In draw mode, the mouse pointer is the Alternate Select pointer, and if you are using the default settings, the mouse pointer will look like this .

Note that while in draw mode if you run the mouse pointer over the toolbar buttons or the menus, the pointer temporarily changes to the selection pointer. If you do not click on one of the menus or toolbar buttons, the mouse pointer reverts to the draw mode pointer when you move back into the display window.

Note:

If you edit the mouse properties in the Windows Control Panel, the change applies to all of Windows, not just ETABS.

Typically, set the properties for the mouse by clicking on the Windows Start menu, then Settings, then Control panel and clicking on Mouse to bring up your Mouse properties form.

Other mouse properties used at various special times in the program include Help Select, Busy, Text Select, Vertical Resize, Horizontal Resize, and Move. The appearance of each of these mouse pointers will also change depending on the mouse pointer properties you specify.

Locking and Unlocking a Model

Shortcut:

Use the ***Lock/Unlock*** *button to quickly lock or unlock your model.*

A toggle switch that allows a model to be locked or unlocked is available on the Options menu and also as a toolbar button . When a model is locked, no changes can be made to it. Unlocking the model allows changes to be made.

After an analysis has been run, the program will automatically save the results and lock the model to prevent any changes that would invalidate the analysis results and subsequent design results that may be obtained.

To make changes to a model after an analysis has been run, first unlock the model. When you select the toggle switch to unlock the model, the program will warn that the analysis results will be deleted. If you do not want the analysis results to be deleted, save the model under a new name before unlocking it, then make any subsequent changes to the newly named model.

Undo Features

Shortcut:

Use the *Undo button on the toolbar to undo a previous operation.*

The Undo feature enables changes to model drawing (geometry changes) for multiple steps, back to the last time you saved your model. For example if you draw one or more objects and then decide you do not want them, use the **Edit menu > Undo** command to remove them. If you then decide you want them back, use the **Edit menu > Redo** command. The Undo and Redo commands work sequentially. In other words, if you have just finished the sixteenth operation since your last save, you can use the Undo feature to undo the sixteenth operation, then the fifteenth operation, and so on. However, you could not, for example, undo only the seventh operation.

The Undo and Redo features do not work for changes made in forms. When an item is changed in a form, the change is not actually implemented until the **OK** button has been clicked. If the **Cancel** button is clicked, the change is ignored and all original values are restored. If you are working in a sub-form, which is accessed through another form, the changes are not actually implemented until the **OK** button is clicked in the topmost form; that is, until the last form is closed by clicking the **OK** button.

Shortcut:

Use the *Redo button to redo a previously undone operation.*

Suppose for example that you are in a series of sub-forms that go five levels deep. For changes made at the fifth level to be accepted and implemented, click the **OK** button at the fifth, fourth, third, second and topmost levels. Clicking the **Cancel** button at any one of the levels cancels any of the changes made at that level and at any lower level. Thus, if you click the **Cancel** button in the topmost level of a form, no changes will be made at any level.

1 Tips for Using the Graphical User Interface

Following are tips for using the graphical user interface:

- Startup Tips appear when you first start the graphical interface. Left clicking anywhere in the window closes the Tip of the Day window. Clicking the **OK** button associated with the tip or clicking the "X" button in the upper right-hand corner of the tip window also closes the Tip window. Use the **Options menu > Show Tips at Startup** command to turn off this feature.

- The graphical user interface works most efficiently with at least a 17-inch monitor and at least 1024x768 resolution.

- Although up to four windows can be open, one or two windows tiled vertically usually works best. Set the number of windows using the **Options menu > Windows** command.

- Customize the graphical user interface display colors to suit your individual preferences using the **Options menu > Colors** command.

- Modify the maximum and minimum graphic font sizes used in the display windows in the graphical user interface using the **Options menu > Preferences > Dimensions/Tolerances** command.

- The menus in the interface are intended to be organized logically so that you can easily locate any menu item. Ask yourself, “What do I want to do?” When creating or modifying a model, you typically edit the model, change the view of the model, define properties or load cases, draw something new in the model, or assign something to the model, such as properties or loads. Thus, one word can describe possible actions, such as edit, view, define, draw, or assign. These action word indicate which menu to use.

- Right clicking on an object brings up a form with information about location, geometry and assignments for that object. This

information is for viewing only. You cannot edit the information in this form.

- Save your model often.
- The Undo feature works for multiple steps, back to the last time you saved your model.
- Do not overlook the extensive online help that is available at the press of the F1 key on your keyboard. If you are in a form when you press the F1 key, context sensitive help related to that form will be displayed.
- Useful information about your model is displayed as text on the left-hand side of the status bar at the bottom of the main window.
- Try a few practice problems to get the hang of the graphical user interface.
- If you have used an earlier version, you may want to import a familiar file to see how it looks in this latest version.

Chapter 2

Overview of a Model

A model developed using this program is different from models produced in many other structural analysis programs for two main reasons:

Note:

There are three types of objects in this program. They are area, line and point objects.

- This program is optimized for modeling building systems. Thus, the modeling procedures and design capabilities are all tailored to buildings.
- This program's model is object-based. It consists of area, line and point objects. You make assignments to these objects to define structural elements such as beams, columns, braces, floors, walls, ramps and springs. You also make assignments to those same objects to define loads.

When an analysis is run, the program automatically converts an object-based model into an element-based model that is used for analysis. We refer to this element-based model as the analysis model. The analysis model consists of joints, frame elements, link elements and shell elements in contrast to the point, line and

area objects in the object-based model that you create. The conversion to the analysis model is internal to the program and essentially transparent (not visible) to the user.

After the analysis is run, the results are reported with respect to the object-based model, not the analysis model. See the section titled "The Analysis Model" later in this chapter for additional information on the analysis model.

The Concept of Objects

Tip:

Understanding area, line and point objects is probably the most important key to successful modeling in this program.

In its simplest form, developing a model requires three basic steps. They are:

- Draw a series of area, line and point objects that represent your building using the Draw menu drawing tools or their corresponding toolbar buttons. See Chapter 8 for documentation of the Draw menu commands and toolbar buttons.

  Note that you could also create these objects directly from one of the built-in templates. See the subsection entitled "Add Structural Objects from a Template" in Chapter 4, and the section entitled "Add to Model From Template Command" in Chapter 5 for more information.

- Assign structural properties and loads to objects using the Assign menu options. See Chapter 10 for documentation of the

A Note about Objects

The concept of objects in a structural model may be new to you. **It is extremely important that you grasp this concept because it is the basis for creating a model in this program.** It is not a difficult concept, but because it is new, it may take some time for you to become comfortable with it. After you understand the concept and have worked with it for a little while, you should recognize the simplicity of the object-based modeling, the ease with which you can create models using objects, and the power of the concept for creating more complex models.

Assign menu commands.

Note that you can also assign structural properties to objects as you draw them using the floating Properties of Object box that appears when you select a Draw menu command. This feature is documented in Chapter 8.

- Manually mesh the area objects if they are not horizontal membrane slab or deck sections that you are letting the program fully, automatically mesh into the analysis model.

Tip:

If you left click any object, the object is selected. If you right click any object while you are in an undeformed shape view, information is displayed in a pop-up form, including exact dimensions, location, and assignments.

An understanding of these three steps is all you need to create a model in this program.

It is extremely important that you become familiar with objects. They are the basic building blocks for your model. If you are new to this program, you should start by studying the area, line and point objects.

Any single area, line or point object can have multiple assignments made to it at the same time. We recommend that you minimize the number of objects in your model by making multiple assignments whenever possible.

For example, suppose you have a beam with three point loads applied to it. You would draw a line object to represent the beam and assign it a frame section (beam) property. You could, although we do not recommend it, draw three point objects located on the beam and assign point loads to those point objects to create the point loads on the beam. Alternatively, you could apply the point loads directly to the line object using the **Assign menu > Frame/Line Loads> Point** command, thereby eliminating the need for the three point objects. We strongly recommend that you assign the point loads directly to the line object.

The Analysis Model

In general, it is not necessary that you concern yourself with the particulars of the analysis model. Nevertheless, it may be helpful

for you to know that when you run an analysis, the program uses the data in your object-based model to create an element-based analysis model (hereafter called the analysis model). All of this happens internally in the program and is essentially transparent (not visible) to the user.

The reason for converting from the object-based model to the analysis model is to create a finite element representation of your model that can be analyzed using standard finite element analysis techniques.

Tip:

If you are a SAP2000 user, you can export your object-based ETABS model to a SAP2000 .s2k file. The SAP2000 model is the same as the element-based analysis model. This is a good way for you to see an ETABS analysis model.

The program transforms your object-based model that is based on area, line and point objects into an analysis model that is based on joints, frame elements and shell elements. In the process of doing this, it internally meshes (divides) some frame elements, as necessary, to provide connectivity to other frame and shell elements in the analysis model.

Also, if your model includes horizontal area objects that are assigned deck or slab properties with membrane behavior only, the program can automatically mesh the floor into the analysis model. You do not need to mesh those types of floors in your object-based model.

All other types of floors, and all ramps and walls must be adequately meshed in your object-based model because the program does not automatically mesh those into the analysis model.

Link elements are never meshed in the analysis model. They always maintain a one-to-one correspondence with the object-based model.

Structural properties are transferred directly from the structural objects to the corresponding elements in the analysis model. Spring stiffnesses for area and line support springs are transferred to spring elements at the analysis joints in a consistent manner based on tributary area.

Loads are transformed from the area, line and point objects in the object-based model onto the frame elements and joints of the analysis model.

After the analysis has been run, the results are typically reported with respect to the objects in the object-based model, not the elements in the analysis model.

Chapter 3

Modeling Tips

This chapters outlines a basic modeling process and then provides some modeling tips.

Modeling Process

It is not necessary, or expected, that you exactly follow the general process for creating a model described in this chapter. Rather, the intent is to guide you in creating a model and running an analysis.

To create a typical model:

1. Set the current units to those that you will want to use most often in your model.

2. Start your model by clicking the **File menu > New Model** command or button and choosing one of the file initialization methods. See the subsection entitled "Initialize a New Model" in Chapter 4 File Menu for more information.

3. Set up your grid lines. See the subsection entitled "Grid Dimensions (Plan) - Define a Grid System" in Chapter 4 File Menu for more information.

4. Define your story levels. See the subsection entitled "Story Dimensions - Define Story Data" in Chapter 4 File Menu for more information.

5. If desired, add structural objects from one of the built-in templates. See the subsection entitled "Add to Model from Template Command" in Chapter 5 Edit Menu for more information. In general, we recommend that you start your model by adding objects from a template whenever possible.

Tip:

In general, we suggest that when starting a model you add structural objects from a template and then edit them as necessary.

6. Use the **Options menu > Preferences** command to modify any of the default preferences if desired. See the section entitled "Preferences" in Chapter 14 Options Menu for more information

7. Use the Define menu to define frame section properties, wall, slab and deck section properties and link properties as required. See Chapter 7 Define Menu for more information.

8. Use the **Define menu > Static Load Cases** command to define your static load cases. See the section entitled "Static Load Cases Command" in Chapter 7 Define Menu for more information.

9. If you are using mass in your model, use the **Define menu > Mass Source** command or the button to specify the source of mass in your model. See the section entitled "Mass Source Command" in Chapter 7 Define Menu for more information.

10. Use the commands available on the Draw menu to draw additional area, line and point objects as needed. See Chapter 8 Draw Menu for documentation of the drawing features.

    As you draw, we recommend that you assign structural properties using the pop-up Properties of Object box.

    The objects you draw may be assigned structural properties, loads or masses. Also, the line objects might be used as guidelines with the Snap To command (see Chapter 8 Draw Menu), the **Trim Lines at or Extend Lines to** options for extending or trimming other lines (see Chapter 5 Edit Menu), or mesh lines for manual meshing.

    Note that mass is required if you are doing a modal analysis to determine mode shapes. It is also required for the non-iterative method of considering P-Delta. It is also required to convert static nonlinear force-deformation results into the capacity spectrum ADRS format.

11. Use the Edit menu commands to modify, and in some cases, tweak the geometry of your model as needed. See Chapter 5 Edit Menu for documentation of the various Edit menu features.

Tip:

The similar stories feature is a useful and powerful tool that you can use when drawing, selecting and making assignments to objects in plan view.

12. Use the Assign menu commands to revise properties in your template model, if necessary, and to make additional assignments to template members as well as to any other members you might have drawn. See Chapter 10 Assign Menu for documentation of the various Assign menu features.

    The types of assignments you make include section properties, loads, masses, moment releases, partial fixity, and so forth.

    To make an assignment to an object, first select the object, then click the appropriate Assign menu command.

13. Use the **Display menu > Show Loads** command and the **Display menu > Set Input Table Mode** command or button to review input. Both of those commands are documented in Chapter 12 Display Menu.

    Another way to review your input is to right click on any object. This brings up a form where you can review all assignments to the object.

    You can also use the **View menu > Set Building View Options** command, or the **Set Building View Options** button to toggle on the display of various input items. Some examples are section properties, member end releases, nonlinear hinges, diaphragm extent and the like.

14. If desired, use the **File menu > Print Tables > Input** command to print input data to a file or to the printer. See the section entitled "Print Commands" in Chapter 4 File Menu for more information.

    Alternatively you can use the **File menu > Export > Save Input/Output as Access Database File** command to save the input data in a database file that can be reviewed, modified and printed using Microsoft Access.

Note:

Note that this program can automatically mesh floors that have membrane properties only.

15. Use the **Analyze menu > Set Analysis Options** to specify various analysis parameters such as the building degrees of freedom. See the section entitled "Set Analysis Options Command" in Chapter 11 Analyze Menu for more information.

16. If your model has floors, walls or ramps that require manual meshing, use the manual meshing options available through the **Edit menu > Mesh Areas** command.

    Note that the program can automatically mesh floors that have membrane properties only. All other floors and all walls and ramps must be manually meshed by you. We recommend that you wait until just before you are ready to run the analysis to perform this manual meshing.

Tip:

We recommend that you run large analyses minimized.

17. Use the **Analyze menu > Run Analysis** command or the button to run your analysis. See the section entitled "Run Analysis Command" in Chapter 11 Analyze Menu for more information. When the analysis is complete, scroll through the text in the Analysis Window to check for any warnings or errors that might invalidate your analysis.

18. Use the display features available on the Display menu to display analysis results on your model or on the screen in a tabular format. See Chapter 12 Display Menu for documentation of the Display menu features.

Tip:

Design is an iterative process. Typically you will rerun your analysis and design several times until your last used analysis section properties match the design sections.

19. If desired, use the **File menu > Print Tables > Analysis Output** command to print output to a file or to the printer. See the section entitled "Print Commands" in Chapter 4 File Menu for more information.

    Alternatively you can use the **File menu > Export > Save Input/Output as Access Database File** command to save the analysis output data in a database file that can be reviewed, modified and printed using Microsoft Access.

20. If desired, use the features available on the Design menu to run your building through one or more of the design post-processors.

21. After you have run a design, save your model before exiting the program. Otherwise your design is not saved.

Modeling Tips

1. Create a Default.edb file that contains standard preferences for your company and use it as a starting point for each model. See the subsection entitled "Initialize a New Model" in Chapter 4 File Menu for more information.

2. Save your model often. It is also useful to occasionally save a backup of your model under a different name.

Tip:

Try to keep your models ***less complex*** *rather than more complex.*

3. With the program's powerful graphical interface, you can easily and quickly create large, complex models. This does not mean you should. ***Resist the temptation to model everything in the building.*** Only model those elements that are an essential part of the vertical or lateral load path.

4. You can get an overview of the options available in the program by "surfing" through the various menus and thumbing through this manual.

5. The **Edit menu > Undo** command or button provides powerful capabilities. Feel free to experiment with options because you can always Undo them.

6. When you use auto select lists for your elements, resist the temptation to put every possible steel section in the auto select list. Keep the auto select lists shorter, say 20 to 30 sections long, to significantly speed up the time it takes to design your model.

7. When creating a model, work in plan and elevation view as much as possible. It is much easier to work in these 2D views than it is to work in a 3D view.

   Do not overlook the developed elevation feature. This feature provides a powerful way for you to work on multiple faces of your building in the same 2D view. See the section entitled "Draw Developed Elevation Definition Command" in Chapter 8 Draw Menu for more information.

8. When creating or editing your model in plan view, consider using the similar stories feature. If you use the similar stories feature, however, keep track of when it is enabled (set to Similar Stories or All Stories) or disabled (set to None) to avoid adding or removing elements from the wrong story level(s) or neglecting to add or remove them.

9. If you are working on a large steel frame building for which you plan to use the Composite Beam Design or Steel Frame Design postprocessor to design the floors and you also plan

to design the lateral system of the building, we suggest the following.

a. Create a single story model for each floor level that is different. Optimize the design of the floors in these single story models.

b. Use the **File menu > Import > ETABS8.edb File** command to import the single story floor framing into a model of the complete building. Note that the similar stories feature is available when you do this import.

c. Run the lateral analysis using the full building with the floor levels imported from single story models.

   For example, assume that you have a ten-story building. Further assume that floor levels 3 through 10 are exactly the same, and floor level 2 and the roof are unique. For floor design, create three single story models: one single story model for level 2, one for levels 3 through 10, and one for the roof.

   Perform your floor design in these three models and then import the floors into a ten-story model for the lateral analysis. Note that you can use the similar stories feature when importing floors 3 through 10 so that you only have to import the single story model once.

   Alternatively, model everything in the ten-story building. The disadvantage to this is that when you do your floor design, you will design floors 3 through 10 separately. Thus, you will design 10 different story levels instead of 3.

Tip:

In this program, it is easy for multiple engineers to work separately on different portions of a large model and then later combine those portions into a single model.

10. For a large building, or if you are working on a tight time schedule, it may be advantageous to have multiple engineers creating different story levels of your building. Those story levels can then be combined into one building using the **File menu > Import > ETABS8.edb File** command.

11. If you are working with a large multistory model, and you want to concentrate on just one story level of that model, use the **File menu > Export > Save Story as ETABS8.edb File** command to export a story to another file as a single story model.

12. The aerial view feature can be useful when you are doing a lot of zooming into and out of regions of your model. See the section entitled "The Aerial View" in Chapter 1 Graphical User Interface for more information.

Note:

The program easily models structures with sloping floors (ramps), such as parking garages.

13. Your structure can be supported at any level. There is no need for "dummy" levels to model nonstandard support conditions.

14. Parking garages with sloping floors can be easily modeled using ramp objects. Create your story levels as you would for any building and connect them where appropriate with ramp objects.

15. You can easily model flexible diaphragms. To do this, assign slab or deck properties to the area object that represents the floor. The membrane properties of the slab or deck model the in-plane diaphragm flexibility. The program automatically lumps the floor mass at the shell element corner points on a tributary area basis.

16. If desired, you can use the **Define menu > Mass Source** command to indicate that the mass of your building is to be determined based on a specified load combination. See the section entitled "Mass Source Command" in Chapter 7 Define Menu for more information.

17. When creating a model, save the manual meshing of area objects as the last thing you do before you run the analysis. This allows you to take advantage of working with fewer objects as you create your model.

18. The program automatically generates static lateral seismic and wind loads based on building code requirements. See the

section entitled "Static Load Cases Command" in Chapter 7 Define Menu for more information.

19. We recommend that as you draw objects, you assign structural properties to them using the floating Properties of Object box. See the sections entitle "Draw Line Object Command" and "Draw Area Object Command" in Chapter 8 Draw Menu for more information.

Tip:

To help draw objects in your model accurately, use the snap options. Reference planes and reference lines can assist you when snapping. See the section entitled "Edit Reference Plans and Reference Lines" in Chapter 5 Edit Menu for more information.

20. It is important that you draw your model accurately. The snap options can help you do this. See the subsection entitled "Snap To Commands" in Chapter 8 Draw Menu for more information.

    If you do not draw your model accurately, the program may not interpret the member connectivity in the way you intend. For example, assume you draw a beam framing into a girder but you stop the beam slightly short of the girder because you did not have the snap options turned on. In that case, depending on the tolerances set and how far the end of the beam is from the girder, the program may not interpret the beam as connecting to the girder. You can avoid this problem by using the snap options so that when the line object representing the beam is drawn, it snaps onto the line object representing the girder. The snap option that would do this is the Snap to Lines and Edges option.

    If you have already drawn objects that are slightly mislocated, use Align features to fix the problem. See the section entitled "Aligning Points/Lines/Edges Command" in Chapter 5 Edit Menu for more information.

Tip:

Do not be shy about defining groups in your model. You will find many uses for them as you create your model, use the design postprocessors and review output results.

21. Groups can be a great benefit when creating a model. You can select elements by groups. Assume you have a braced frame model and that you assign all of your braces to a group. You can then select all of the braces at once by group using the **Select menu > Select by Groups** command. After the braces have been selected, make assignments to them as a group or print input/output tables for them as a group. You can also design elements as a group. In this case, all of the

elements in the design group are given the same section property by the design postprocessor.

See the section entitled "Group Name Command" in Chapter 10 Assign Menu for more information.

22. In general, given the choice, it is better to make rigid diaphragm assignments to horizontal area objects (floors) rather than to point objects. This tip applies to loads assigned to line objects. In general, where possible, we *strongly recommend* that you assign line and point loads to line objects *with frame section properties* (i.e., columns, beams and braces assigned to them) rather than simply assigning the loads to null-type line objects located somewhere on an area object. The transformation of loads from your object-based model into the element-based analysis model is more easily predicted when the loads are assigned to a frame element.

23. Carefully review your input both graphically and in tables to make sure you have modeled what you meant to model. One of the most common problem areas is the frame member end releases; check them carefully.

24. After you run your analysis and before clicking the **OK** button in the Analysis Window, scroll through the messages in the Analysis Window checking for any warnings or error messages that might invalidate your analysis.

25. Carefully review your analysis output results to make sure that your model is behaving as you expect. If it is not, investigate to find out why.

Chapter 4

File Menu

General

The File menu provides basic file operations for creating new models, opening existing models and saving models. It also provides options for printing input and output data as well as controls for other miscellaneous features. This chapter describes the features available on the File menu.

New Model Command

The **File menu > New Model** command is used to start/create a new model. Alternatively, you can click the **New Model** button on the toolbar or use Ctrl+N (hold down the control key and then type N). There are four distinct stages in creating a new model:

- Initialize the model.

- Define a grid system.
- Define story data.
- Add structural objects from a template.

Default values are provided for each of the stages, so with little more than a few clicks of some **OK** buttons, you can create a complete model using default dimensions and properties. More typically, you will modify the default values provided for each of the stages to specify the particular characteristics of your model. The following four subsections describe each of the four stages of creating a new model.

Initialize a New Model

Tip:

You can initialize a new model based on any .edb file.

When you execute the **File menu > New Model** command, a form will display asking if you want to initialize your new model using definitions and preferences from an existing .edb file. (Note: An .edb file is an ETABS model file.) You may select one of three button: the Choose.edb button, the Default.edb button, or the No button. An explanation of these buttons follows.

- **Choose.edb Button.** If you select the Choose.edb button, you will be asked to specify an ETABS file that has an .edb extension in any subdirectory on your computer system. After you have specified an .edb file, the program will start your new model with the definitions and preferences from that .edb file. In that case, the program essentially imports the entire specified .edb file into your new .edb file, except for the following items:
  - ✓ Grid lines
  - ✓ Story data
  - ✓ Objects
  - ✓ Assignments to objects

✓ Information on the number of windows and what is showing in the windows.

Tip:

We recommend that you create your own custom Default.edb file and place it in the directory that contains the Etabs.exe file.

- **Default.edb Button.** If you select the Default.edb button, the program will start your new model using definitions and preferences that are specified in the Default.edb file that is in the same directory as the ETABS.exe file. If no Default.edb file exists in the same directory as the ETABS.exe file, the program will use built-in values for all of the definitions and preferences in your new model.
- **No Button.** If you select the No button, the program will use built-in values for all of the definitions and preferences in your new model.

Note:

The Default.edb file is simply a typical ETABS .edb file that has been renamed Default.edb.

Creating .edb Files for Initializing Your Models

Create an .edb file for initializing your model by renaming an existing .ebd (or model) file.

When creating a Default.edb file, it is most logical to select an existing .edb file that uses most of the common practices of your office with respect to how models should be set up (e.g., units, section properties and the like). Store this file in the same directory as the ETABS.exe file.

If necessary, create other ETABS .edb files that may in certain circumstances be useful for initializing your model. You may want to store those .edbs in a location where all engineers have access to them, or you may want them to be more personalized and have limited access. Either way is functional for ETABS. It works best if the files have names other than Default.edb so that you do not confuse the files (i.e., you could have multiple Default.edb files stored in separate subdirectories and use the Choose.edb button to select a file name Default.edb, but this can be confusing).

Building Plan Grid System and Story Definition Form

After you have used the **New Model** command to initialize your model, the Building Plan Grid System and Story Definition form will appear. This form has four areas:

- **Grid Dimensions (Plan)**, which is used to define a grid system.
- **Story Dimensions**, which is used to define story data.
- **Units**, which is used to define the units to be used in the model.
- **Add Structural Objects**, which is used to add structural objects from program templates.

With the exception of the Units area, which is assumed to be self-explanatory, these areas are described in the following subsections. The form also has **OK** and **Cancel** buttons, which are used to accept or cancel the selections made on the form. You must select the **OK** button for any changes you make to this form to be accepted. Selecting the **Cancel** button cancels any changes.

Grid Dimensions (Plan) - Define a Grid System

Tip:

The custom grid spacing item that is available when you start a new model allows you to immediately specify nonuniformly spaced grid lines.

In the Grid Dimensions (Plan) area of the form, you can define a grid line system. You can select from two options for defining the grid line system:

- **Uniform Grid Spacing:** For this option, you specify the number of grid lines in the X and Y directions and a uniform spacing for those lines. Note that the uniform spacing in the X and Y directions can be different. This option defines a grid system for the global coordinate system only. If necessary, you can later edit this information using the **Edit menu > Edit Grid Data** command. Refer to the section entitled "Edit Grid Data Command" in Chapter 5 for more information. As noted in Chapter 5, the default global coordinate/grid system is a Cartesian (rectangular) coordinate system. Thus, you will likely use

the **Edit > Edit Grid Data > Edit Cartesian/Cylindrical System** command to modify the grid system.

- **Custom Grid Spacing:** This option allows you to label grid lines and to define nonuniformly spaced grid lines in the X and Y directions for the global coordinate system. After you have chosen this option, click the Grid Label button to label grid lines and click the **Edit Grid** button to edit the grid system. If you want to make subsequent changes to your model grid system, please refer to the explanation in Chapter 5 in the "Edit Grid Data Command" section.

There are several reasons for defining a grid system for your model, including:

- Default elevation views in the model occur at each defined primary grid line in your model.
- Structural objects added to the model from a template are added based on the grid line definitions in the model.
- Objects snap to grid lines when drawn in the model.
- Objects mesh at their intersections with grid lines.
- The grid lines in the model can be defined with the same names as are used on the building plans. This may allow for easier identification of specific locations in the model.

Story Dimensions - Define Story Data

Story data is defined from the Story Dimensions area of the Building Plan Grid System and Story Definition form. You can selected from two options for defining the story data:

- **Simple Story Data:** With this option, you can define the number of stories and a typical story height that is used for all story levels. The program provides default names for each story level and assumptions for story level similarity. (You can later edit all of this information using the **Edit menu > Edit Story Data > Edit** command. Refer to the section entitled

"Edit Story Data Command" in Chapter 5 for more information.)

- **Custom Story Data:** This option allows you to define your own story names, story levels of non-uniform height and customized story similarity. After you have chosen this option, click the **Edit Story Data** button and the Story Data form will appear. This form is the same one that appears when you execute the **Edit menu > Edit Story Data > Edit** command. Refer to the section entitled "Edit Story Data Command" in Chapter 5 for more information.

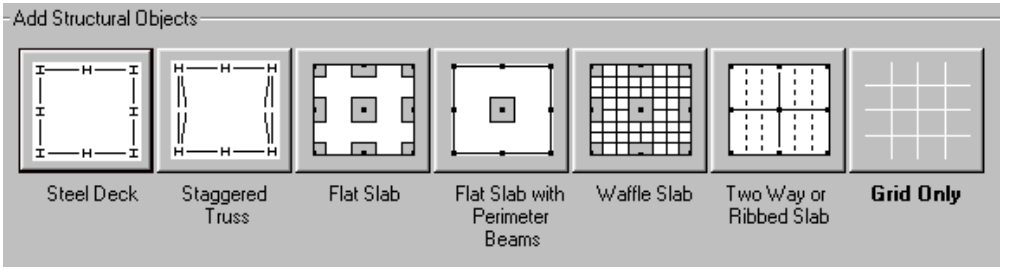
Note:

You can create steel and concrete building models using built-in templates. These models can then be edited on screen to satisfy specific situations.

Add Structural Objects

While in the Building Plan Grid System and Story Definition form, you can add structural objects to your model from one of several built-in templates. It is not necessary that you add the structural objects from a template. You can always draw, import, copy or replicate structural objects later (see Chapter 5 Edit Menu and Chapter 8 Draw Menu for more information). However, in many cases it is simplest, most convenient and quickest to start your model with structural objects added from a template.

The Add Structural Objects area of the Building Plan Grid System and Story Definition form is reproduced herein for reference:



Note that there is one steel building template called Steel Deck, five concrete building templates, and a button for grids only

where no structural objects are added to the model from a template. You can always tell which option (button) has been selected in the Add Structural Objects area because its name will be highlighted. When the Building Plan Grid System and Story Definition form opens, the Grid Only selection is highlighted, thus indicating that unless you select another template, your model will have only a grid system.

You can choose any of the templates by left clicking its associated button. When you choose one of the template buttons, a form for that template will appear where you specify various types of data for the template. Notice that each of the template buttons has a pattern that suggests the type of template. This pattern is repeated in a small area of the form that comes up after the button has been selected. The six template forms (one steel and five concrete) are described in the subsections that follow.

When you have finished specifying data for a template, click the **OK** button to return to the Building Plan Grid System and Story Definition form. As indicated previously, the button name for the template that you just specified should be highlighted. If you then decide you defined the wrong type of template, click another template button and define data for it. When the program creates the model, it adds structural objects based on the last button clicked in the Add Structural Objects area; that is, the highlighted button. You can select only one template on the Building Plan Grid System and Story Definition form.

Note that when using concrete building templates in this program, beams and slab ribs (joists) are normally modeled with depths equal to the dimension from the top of the slab (not bottom of slab) to the bottom of the beam or slab rib. Also note that beams are modeled as line elements in this program. Thus, slabs with out-of-plane bending capability span from center-of-beam to center-of-beam in the program model.

Steel Deck Button - Steel Floor System Form

Selecting the **Steel Deck** button on the Building Plan Grid System and Story Definition form brings up the Steel Floor System form, which allows you to define a steel floor system. Following are descriptions of each of the items on the Steel Floor System form:

- **Overhangs:** These are the distances from the perimeter grid line to the edge of the slab/deck. These distances must be greater than or equal to zero. They cannot be negative. Figure 4-1 illustrates the overhangs of a steel floor system.

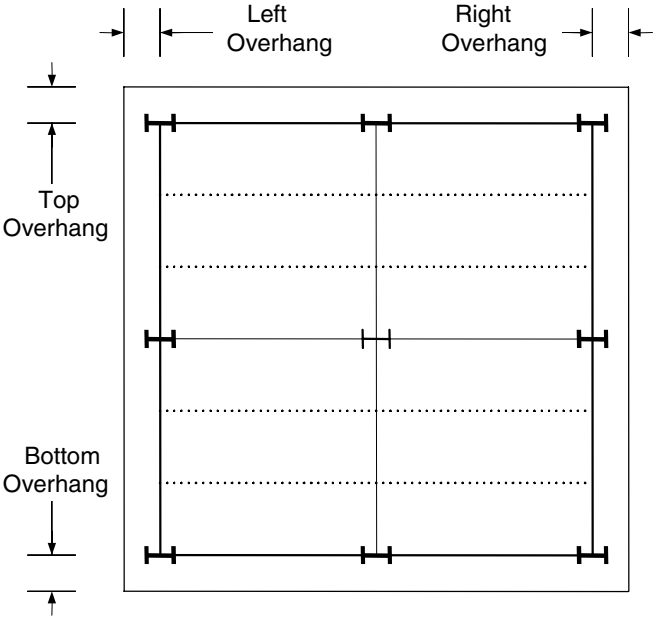


Figure 4-1:
Steel Floor System

- **Secondary Beams:** Selecting the Secondary Beams check box on the Steel Floor System form will include secondary (infill) beams. If the check box is not checked, no secondary beams are included. Secondary beams are the beams that do not frame into columns. Secondary beams are shown in Figure 4-1 as dotted lines. The direction of the beams can be X or Y. Direction X means the span of the beams is parallel to the X-axis.

Direction Y means the span of the beams is parallel to the Y-axis.

Tip:

By default secondary steel beams are pinned.

You can specify the number of secondary beams using one of two methods. You can either specify a maximum spacing, in which case the program determines how many beams fit in the bay, or you can specify a number of equally spaced beams.

Note that by default, moment is released at the ends of all secondary steel beams; that is, they are pinned.

- **Structural System Type:** There are three possible selections for structural system type:
  - ✓ **No Moment Frame:** Moment is released at all beam-to-column connections; that is, those connections are pinned.
  - ✓ **Perimeter Moment Frame:** Moment is *not* released at beam-to-column connections for perimeter columns. Moment is released at beam-to-column connections for all interior columns. In other words, beam column connections are moment resistant at perimeter columns and pinned at interior columns.
  - ✓ **Intersecting Moment Frame:** Moment is *not* released at any beam-to-column connections; that is, all beam-to-column connections are fully moment resisting.
- **Restraints at Bottom:** You can specify no restraints (supports) at the bottom of all columns, pinned restraints (UX, UY and UZ restrained and RX, RY and RZ free), or fixed restraints (UX, UY, UZ, RX, RY and RZ restrained).
- **Structural System Properties:** Here you specify frame section properties to be assigned to columns and beams and a deck section property for the deck/slab. The items in this area include the following:
  - ✓ **Lateral Column:** Column where the beam-to-column connections are fully moment resisting.

- ✓ **Lateral Beam:** Beam where the beam-to-column connections are fully moment resisting.
- ✓ **Gravity Column:** Column where the beam-to-column connections are pinned; that is, *not* fully moment resisting.
- ✓ **Gravity Beam:** Beam where the beam-to-column connections are pinned; that is, *not* fully moment resisting.
- ✓ **Secondary Beam:** All secondary beams.
- ✓ **Deck/Floor:** The deck/slab.

- **Load:** The Dead Load Case drop down box lists all defined static loads that are type Dead. You can choose any one of these load cases in the drop-down box (in most cases there will probably be only one) and then define additional uniformly distributed dead load for that case in the Dead Load (Additional) box. The word additional is used to indicate that it is in addition to the self weight you specify using the self weight multiplier when you define the static load case (see "Static Load Cases Command" in Chapter 7 Define Menu for more information).

  The Live Load Case drop-down box lists all defined static loads that are type Live. You can choose any one of the load cases in the drop-down box and then define uniformly distributed live load for that case in the Live Load box.

- **Create Rigid Floor Diaphragm:** Checking the Create Rigid Floor Diaphragm check box applies a rigid diaphragm constraint to the area object representing the slab/deck.

Staggered Truss Button - Stagger Truss System Form

Selecting the **Staggered Truss** button on the Building Plan Grid System and Story Definition form brings up the Stagger Truss System form, which allows you to define a staggered truss system. This form consists of the following areas:

- **Overhangs:** These are the distances from the perimeter grid line to the edge of the slab/deck in the X and Y directions. These distances must be greater than or equal to zero. They cannot be negative.

- **Truss:** Here the user can assign the truss in the X or the Y direction as well as define the Typical Size of the Panel. Sections can be assigned to Cords, Diagonals and Vertical Truss members as desired.

- **Restraints at Bottom:** You can specify no restraints (supports), pinned restraints (UX, UY and UZ restrained and RX, RY and RZ free), or fixed restraints (UX, UY, UZ, RX, RY and RZ restrained) at the bottom of all columns.

- **Structural System Properties:** Here you specify section properties to be assigned to the framing members and a deck section property for the deck/slab. The items in this area include the following:

  - ✓ **Column:** This column supports the truss; that is, the column is beneath the truss.

  - ✓ **Spandrel:** This beam spans between trusses.

  - ✓ **Brace (bottom):** This is the horizontal brace for the truss bottom chord, where required.

  - ✓ **Hanger:** This hanger is the vertical tension member that supports the level below the hanger.

  - ✓ **Post:** This post is the vertical compression member that supports the level above the post.

- **Load:** The Dead Load Case drop-down box lists all defined static loads that are type Dead. You can choose any one of these load cases in the drop-down box (in most cases there will probably be only one) and then define additional uniformly distributed dead load for that case in the Dead Load (Additional) box. The word additional is used to indicate that it is in addition to the self weight you specify using the self weight

multiplier when you define the static load case (see "Static Load Cases Command" in Chapter 7 Define Menu for more information).

The Live Load Case drop-down box lists all defined static loads that are type Live. You can choose any one of the load cases in the drop-down box and then define uniformly distributed live load for that case in the Live Load box.

- **Create Rigid Floor Diaphragm:** Checking the Create Rigid Floor Diaphragm check box applies a rigid diaphragm constraint to the area object representing the slab/deck.

Flat Slab Button - Flat Slab Form

Selecting the **Flat Slab** button on the Building Plan Grid System and Story Definition form brings up the Flat Slab form, which allows you to define a concrete flat slab floor system with drop panels. No beams are included in this floor system. The form includes the following areas:

- **Overhangs:** These are the distances from the perimeter grid line to the edge of the slab. These distances must be greater than or equal to zero. They cannot be negative.

- **Drop Panels:** Checking the Drop Panels check box adds drop panels to the model. If the check box is unchecked, no drop panels are included. Figure 4-2 illustrates a flat slab system with a drop panel.

  The drop panels are typically assumed to be square and centered on the columns, which are located at all grid line intersections. The Size item for the drop panels is the length of one side of the drop panel. If the drop panel occurs at a perimeter column and the edge distance at that location is less than one-half the drop panel size, the drop panel is truncated at the edge of the slab.

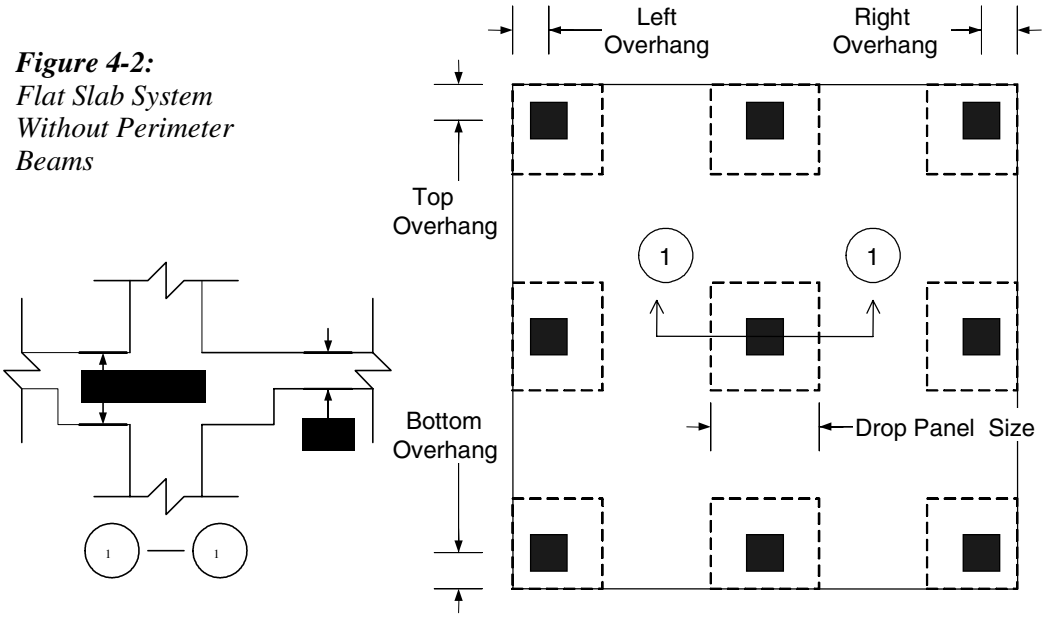


Figure 4-2:
Flat Slab System Without Perimeter Beams

Note that the thickness (depth) of the drop panel is controlled by the section property assigned to it in the Structural System Properties area of the form.

- **Restraints at Bottom:** You can specify no restraints (supports), pinned restraints (UX, UY and UZ restrained and RX, RY and RZ free), or fixed restraints (UX, UY, UZ, RX, RY and RZ restrained) at the bottom of all columns.

- **Structural System Properties:** Here you specify a frame section property to be assigned to the columns and slab section properties to be assigned to the slab and drop panels. The items in this area include the following:

  - ✓ **Column:** Specify frame section properties for all of the columns in the template model.

  - ✓ **Slab:** Specify the slab section property for the floor slab, excluding the drop panels.

  - ✓ **Drop:** Specify the slab section properties for all of the drop panels in the template model.

- **Load:** The Dead Load Case drop down box lists all defined static loads that are type Dead. You can choose any one of these load cases in the drop-down box (in most cases there will probably be only one) and then define additional uniformly distributed dead load for that case in the Dead Load (Additional) box. The word additional is used to indicate that it is in addition to the self weight you specify using the self weight multiplier when you define the static load case (see "Static Load Cases Command" in Chapter 7 Define Menu for more information).

  The Live Load Case drop-down box lists all defined static loads that are type Live. You can choose any one of the load cases in the drop-down box and then define uniformly distributed live load for that case in the Live Load box.

- **Create Rigid Floor Diaphragm:** Checking the Create Rigid Floor Diaphragm check box applies a rigid diaphragm constraint to the area object representing the slab and drop panels.

- **Mesh Area for Analysis:** Checking the Mesh Area for Analysis check box directs the program to mesh the slab immediately. Because you are just beginning to develop your model, it may be desirable to leave this check box unchecked, at least in larger models. Meshing causes the program to slow down, which on larger models, can mean make a significant difference to your schedule. We recommend that you finish all modeling input and then apply automatic meshing just before running an analysis. Automatic meshing is described the "Mesh Areas Command" section in Chapter 5 Edit Menu.

Flat Slab with Perimeter Beams Button - Flat Slab with Perimeter Beams Form

Selecting the **Flat Slab with Perimeter Beams** button on the Building Plan Grid System and Story Definition form brings up the Flat Slab with Perimeter Beams form, which allows you to define a concrete flat slab floor system with drop panels and perimeter beams. The only difference between this template and the Flat Slab template is that this template includes beams fram-

ing between the perimeter columns. Note that the connection between the beams and the columns is modeled as fully moment resistant, as one would typically expect for a concrete structure. The Flat Slab with Perimeter Beams form includes the following areas:

- **Overhangs:** These are the distances from the perimeter grid line to the edge of the slab. These distances must be greater than or equal to zero. They cannot be negative.

- **Drop Panels:** Checking the Drop Panels check box includes drop panels in the model. If the check box is not checked no drop panels are included. Figure 4-3 illustrates a flat slab with drop panels and perimeter beams.

  The drop panels are typically assumed to be square and centered on the columns, which are located at all grid line intersections. The Size item for drop panels is the length of one side of the drop panel. If the drop panel occurs at a perimeter column and the edge distance at that location is less than one-half the drop panel size, the drop panel is truncated at the edge of the slab.

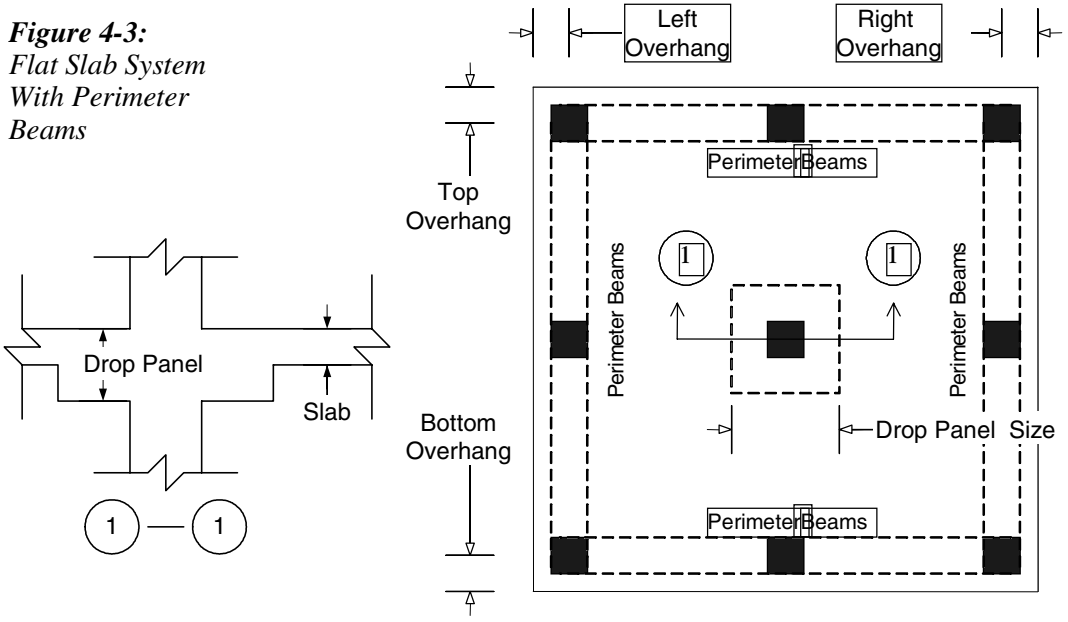


Figure 4-3: *Flat Slab System With Perimeter Beams*

Note that the thickness (depth) of the drop panel is controlled by the section property assigned to it in the Structural System Properties area of the form.

- **Restraints at Bottom:** You can specify no restraints (supports), pinned restraints (UX, UY and UZ restrained and RX, RY and RZ free), or fixed restraints (UX, UY, UZ, RX, RY and RZ restrained) at the bottom of all columns.

- **Structural System Properties:** Here you specify frame section properties to be assigned to the columns and perimeter beams and you specify slab section properties to be assigned to the slab and drop panels. The items in this area are:

  - ✓ **Column:** Specify frame section properties for all of the columns in the template model.

  - ✓ **Beam:** Specify frame section properties for all perimeter beams in the template model.

  - ✓ **Slab:** Specify the slab section property for the floor slab, excluding the drop panels.

  - ✓ **Drop:** Specify the slab section properties for all of the drop panels in the template model.

- **Load:** The Dead Load Case drop-down box lists all defined static loads that are type Dead. You can choose any one of these load cases in the drop-down box (in most cases there will probably be only one) and then define additional uniformly distributed dead load for that case in the Dead Load (Additional) box. The word additional is used to indicate that it is in addition to the self weight you specify using the self weight multiplier when you define the static load case (see "Static Load Cases Command" in Chapter 7 Define Menu for more information).

  The Live Load Case drop-down box lists all defined static loads that are type Live. You can choose any one of the load

cases in the drop-down box and then define uniformly distributed live load for that case in the Live Load box.

- **Create Rigid Floor Diaphragm:** Checking the Create Rigid Floor Diaphragm check box applies a rigid diaphragm constraint to the area object representing the slab and drop panels.

- **Mesh Area for Analysis:** Checking the Mesh Area for Analysis check box directs the program to mesh the slab immediately. Because you are just beginning to develop your model, it may be desirable to leave this check box unchecked, at least in larger models. Meshing causes the program to slow down, which on larger models, can mean make a significant difference to speed of model development. We recommend that you finish all modeling input and then apply automatic meshing just before running an analysis. Automatic meshing is described the "Mesh Areas Command" section in Chapter 5 Edit Menu.

Waffle Slab Button - Waffle Slab Form

Note:

In waffle slabs, the program does not consider the rectangular space between the centerlines of four adjacent ribs (joists) to be filled with a drop panel unless the drop panel size specified fully fills that space.

Clicking the **Waffle Slab** button on the Building Plan Grid System and Story Definition form brings up the Waffle Slab form, which allows you to define a concrete waffle slab floor system with drop panels (solid column heads) and perimeter beams. Note that the connections between the ribs and either other ribs or the columns are modeled as fully moment resistant, as one would typically expect for a concrete structure. The Waffle Slab form includes the following areas:

- **Overhangs:** These are the distances from the perimeter grid line to the edge of the slab. These distances must be greater than or equal to zero. They cannot be negative.

- **Drop Panels and Ribs:** Checking the Drop Panels check box means to include drop panels (solid heads) in the model. If the check box is not checked, no drop panels will be included in the model.

4

The drop panels are typically assumed to be square and centered on the columns, which are located at all grid line intersections. The Size item for drop panels is the length of one side of the drop panel. If the drop panel occurs at a perimeter column and the edge distance at that location is less than one-half the drop panel size, the drop panel is truncated at the edge of the slab. Figure 4-4 illustrates a waffle slab system.

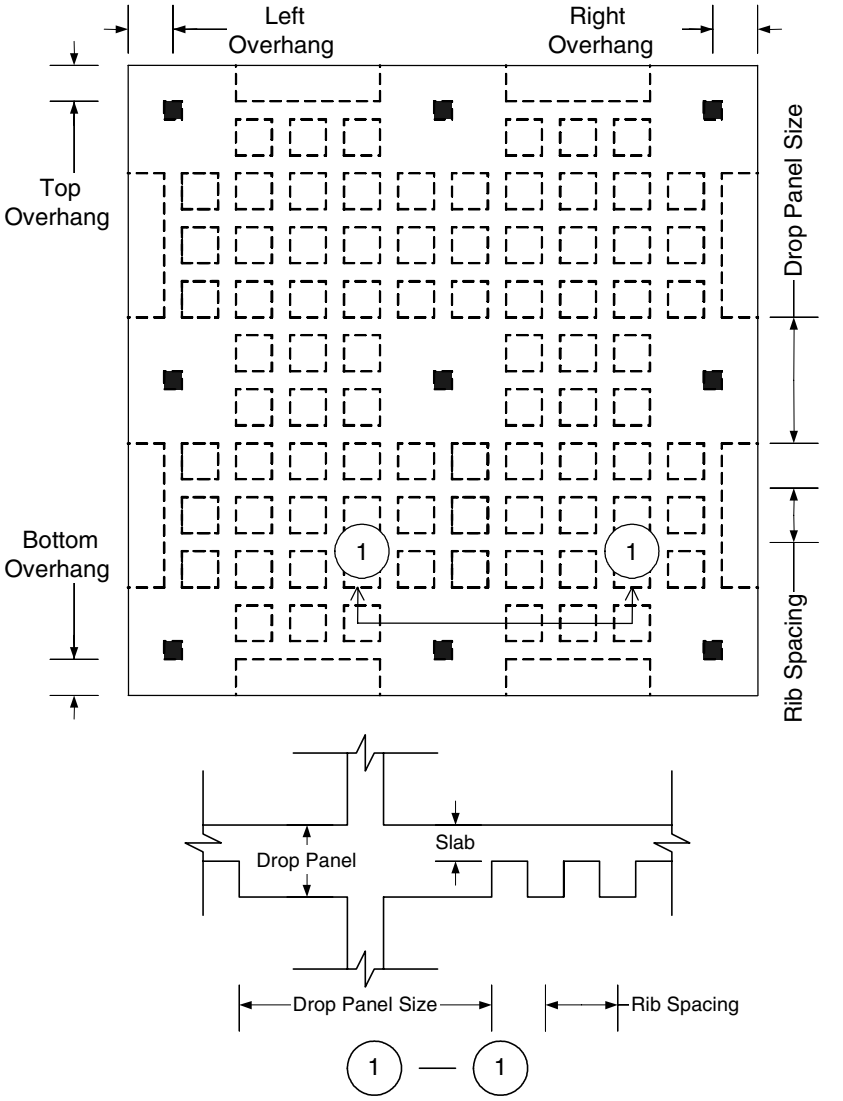


Figure 4-4: *Waffle Slab System*

The actual size of a drop panel included in a model may be less than was input into the template. This happens because in waffle slabs, the program does not consider the rectangular space between the centerlines of four adjacent ribs (joists) to be filled with a drop panel unless the drop panel size specified *fully* fills that space. The drop panel is ignored in any rectangular space between adjacent ribs that it does not fully fill.

Note that the thickness (depth) of the drop panel is controlled by the section property assigned to it in the Structural System Properties area of the form.

Checking the Ribs check box will include waffle slab ribs in the model. If the check box is unchecked, no slab ribs will be included.

Ribs are always provided interconnecting the columns. The rib spacing specified is the typical center-of-rib to center-of-rib spacing that applies to each bay of the structure. When the specified rib spacing is not an exact multiple of the bay width, the ribs are still typically spaced at the specified rib spacing. Any required uneven spacing will occur between the ribs on the grid lines interconnecting the columns and the first adjacent rib. This uneven space is always larger than the specified rib spacing. We assume that you will manually adjust the width of the ribs (beams) on the grid lines if necessary to maintain a constant form size for your waffle slab.

An example of rib spacing is shown in the next subsection for a one-way ribbed slab.

- **Restraints at Bottom:** You can specify no restraints (supports), pinned restraints (UX, UY and UZ restrained and RX, RY and RZ free), or fixed restraints (UX, UY, UZ, RX, RY and RZ restrained) at the bottom of all columns.

- **Structural System Properties:** Here you specify frame section properties to be assigned to the columns and ribs and you specify slab section properties to be assigned to the slab and drop panels. The items in this area are:

- ✓ **Column:** Specify frame section properties for all columns in the template model.
- ✓ **Ribs:** Specify frame section properties for all waffle slab ribs (in both the X and Y directions) in the template model.
- ✓ **Slab:** Specify slab section properties for the floor slab, excluding drop panels.
- ✓ **Drop:** Specify slab section properties for all drop panels in the template model.

- **Load:** The Dead Load Case drop down box lists all defined static loads that are type Dead. You can choose any one of these load cases in the drop-down box (in most cases there will probably be only one) and then define additional uniformly distributed dead load for that case in the Dead Load (Additional) box. The word additional is used to indicate that it is in addition to the self weight you specify using the self weight multiplier when you define the static load case (see "Static Load Cases Command" in Chapter 7 Define Menu for more information).

  The Live Load Case drop-down box lists all defined static loads that are type Live. You can choose any one of the load cases in the drop-down box and then define uniformly distributed live load for that case in the Live Load box.

- **Create Rigid Floor Diaphragm:** Checking the Create Rigid Floor Diaphragm check box applies a rigid diaphragm constraint to the area object representing the slab and drop panels.

- **Mesh Area for Analysis:** Checking the Mesh Area for Analysis check box directs the program to mesh the slab immediately. Because you are just beginning to develop your model, it may be desirable to leave this check box unchecked, at least in larger models. Meshing causes the program to slow down, which on larger models, can mean make a significant difference to the speed of model development. We recommend that you finish all modeling input and then apply automatic mesh-

ing just before running an analysis. Automatic meshing is described the "Mesh Areas Command" section in Chapter 5 Edit Menu.

Two-Way or Ribbed Slab Button - Ribbed Slab Form

Clicking the **Two Way or Ribbed Slab** button on the Building Plan Grid System and Story Definition form brings up the Ribbed Slab form, which allows you to define a concrete flat slab floor system with beams interconnecting all of the columns. No drop panels are included in this template. Note that the connection between the beams and the columns is modeled as fully moment resistant, as one would typically expect for a concrete structure. The Ribbed Slab form includes the following areas:

- **Overhangs:** These are the distances from the perimeter grid line to the edge of the slab. These distances must be greater than or equal to zero. They cannot be negative.

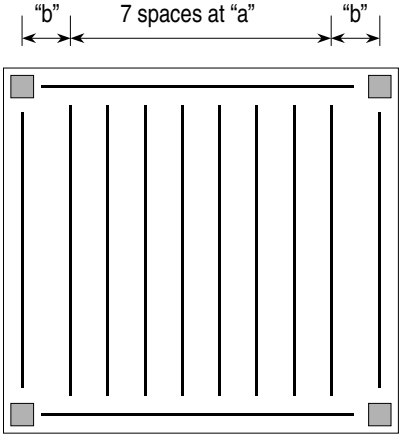
- **Ribs:** Checking the Ribs check box will include the one-way slab ribs (joists) in the model. If the check box is unchecked, no slab ribs are included in the model. The Rib Spacing must be specified by the user. The Direction of Rib can be in either the X or the Y direction. Figure 4-5 illustrates a ribbed slab system. Figure 4-6 illustrates a two way slab system.

- **Restraints at Bottom:** You can specify no restraints (supports), pinned restraints (UX, UY and UZ restrained and RX, RY and RZ free), or fixed restraints (UX, UY, UZ, RX, RY and RZ restrained) at the bottom of all columns.

- **Structural System Properties:** Here you specify frame section properties to be assigned to the columns and beams and you specify slab section properties to be assigned to the slab. The items in this area include the follow:

  - ✓ **Column:** Specific section properties for all columns in the template model.

4



a = specified rib spacing
b = spacing between rib interconnecting columns and first adjacent rib

$a \leq b < 1.5\ a$

Rib Spacing for One-Way Ribbed Slab

Figure 4-5:
Ribbed Slab System

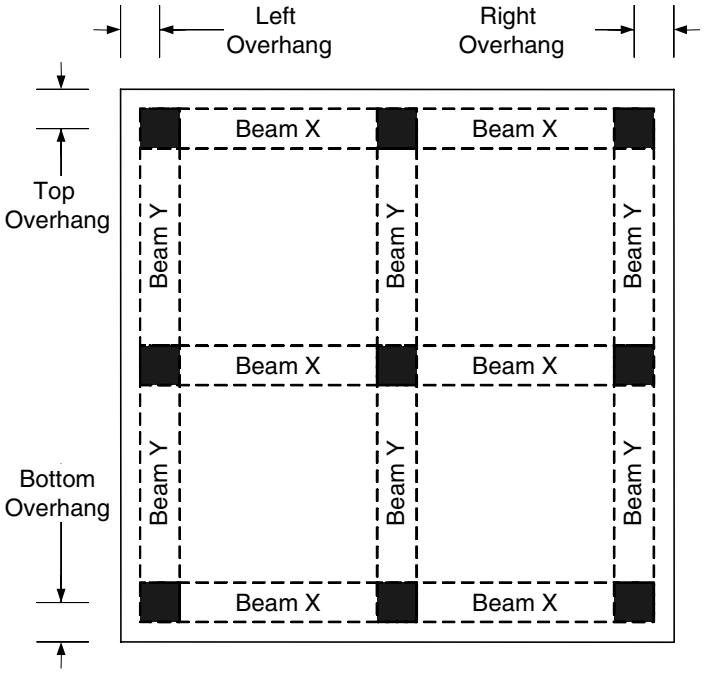


Figure 4-6:
Two Way Slab System

- ✓ **Beam X:** Specific section properties for all beams in the template model that span in a direction parallel to the X-axis.
- ✓ **Beam Y:** Specific section properties for all beams in the template model that span in a direction parallel to the Y-axis.
- ✓ **Slab:** Specify slab section properties for the floor slab.

- **Load:** The Dead Load Case drop down box lists all defined static loads that are type Dead. You can choose any one of these load cases in the drop-down box (in most cases there will probably be only one) and then define additional uniformly distributed dead load for that case in the Dead Load (Additional) box. The word additional is used to indicate that it is in addition to the self weight you specify using the self weight multiplier when you define the static load case (see "Static Load Cases Command" in Chapter 7 Define Menu for more information).

  The Live Load Case drop-down box lists all defined static loads that are type Live. You can choose any one of the load cases in the drop-down box and then define uniformly distributed live load for that case in the Live Load box.

- **Create Rigid Floor Diaphragm:** Checking the Create Rigid Floor Diaphragm check box applies a rigid diaphragm constraint to the area object representing the slab and drop panels.

- **Mesh Area for Analysis:** Checking the Mesh Area for Analysis check box directs the program to mesh the slab immediately. Because you are just beginning to develop your model, it may be desirable to leave this check box unchecked, at least in larger models. Meshing causes the program to slow down, which on larger models, can mean make a significant difference to the speed of model development. We recommend that you finish all modeling input and then apply automatic meshing just before running an analysis. Automatic meshing is de-

scribed the "Mesh Areas Command" section in Chapter 5 Edit Menu.

Grid Only Button

If you select the **Grid Only** button**,** a grid will be added to your model without any structural objects. You can then:

- Use the commands from the Draw menu to draw objects (see Chapter 8 Draw Menu for more information).
- Use the **Edit menu > Add to Model from Template** command to add objects to your model (see the "Add to Model from Template Command" section in Chapter 5 for more information).
- Import stories from a SAFE .f2k file (see the "Import Command" section later in this chapter).
- Import stories from another ETABS .edb file (see the "Import Command" section later in this chapter).
- Copy objects (geometry only) from another ETABS .edb file (see the "Cut, Copy, Paste and Delete Command" section in Chapter 5 for more information).
- Copy objects (geometry only) from a spreadsheet (see the "Cut, Copy, Paste and Delete Command" section in Chapter 5 for more information).
- Use commands from the Edit menu to modify existing objects (see Chapter 5 Edit Menu for more information).

Open Command

To open an existing version 7 or later model, click the **File menu > Open** command or the **Open .EDB File** button located on the toolbar, or hold down the Ctrl key and type O to bring up the Open Model File form. Find the name of the file you want to open in this form and double click on it so that it appears in the

File Name box. Alternatively, you can type the name of the file into the File Name box, including the path if necessary. Then click the **Open** button.

Another way to open an existing model is to select it from the list of recently opened models that is displayed near the bottom of the File menu. This list appears just below the **Delete Analysis Files** command and just above the **Exit** command. If the model you want to open appears in this list, click on its name to open it. Note that this list is maintained in the ETABS.ini file that is kept in your Windows or WinNT directory. If this file is moved or lost, your list of recently used files disappears.

Backup Files

When you open an existing .edb file, the program always immediately creates a backup of that file. The backup file has the same name as your .edb file but it has a .ebk extension. The .ebk file is not undated unless you close the .edb file and then reopen it. The .ebk file is a binary file, not a text file. If your .edb file should somehow become corrupted or lost, you always have a backup of the file in the .ebk file, as it was when you last opened the .edb file. If you need to access the .ebk file, simply change its extension to .edb and use it like a regular .edb file. Thus, in addition to saving your .edb file frequently, it may also be advisable to close and reopen it from time to time to update the .ebk backup file.

Important Note: There is *not* an AutoSave feature in this program. Thus, it is fully your responsibility to save your model.

Save and Save As Commands

Tip:

Save your file often!

To save your model, click the **File menu > Save** command or the **Save Model** button the toolbar, or hold down the control key and type S. If you have just created your model and this is the first time you are saving it, the Save Model File As form appears where you can specify a location (directory) and name for

your model file. Specify the location and name in the form and click the **Save** button to save your model.

If your model previously existed or has been previously saved, clicking the **File menu > Save** command or the **Save Model** button immediately saves your model in the previously specified location, overwriting any earlier versions of the model.

Important Note: There is *not* an AutoSave feature in this program. Thus, it is fully your responsibility to save your file. We recommend that you make it a habit to save your model file early and often. This helps minimize the lose of work that may result from power failures, computer malfunctions or unforeseen software behavior.

If you want to save your model in a new location or with a new name, use the **File menu > Save As** command. The Save Model File As form appears where you can specify a location (directory) and name for your model file.

If a File Becomes Corrupted

When you save a model file, the program actually saves two different files. First it saves a text file with the same name as your .edb file but with a .$et extension. Then it saves your ETABS database file for your model with a .edb extension.

The text file with the .$et extension is intended as a text backup file of your .edb file, which is a binary file. If something happens to the .edb file of your model, such as the file becomes corrupted or it is otherwise lost, you can restore your model by importing the .$et file. Use the **File menu > Import > ETABS8.e2k Text File** command to import a .$et file. Specify the name of your .$et file, including the .$et extension, in the resulting form.

Note that a .$et file is exactly the same as a .e2k file that you can export using the **File menu > Export > Save Model as ETABS8.e2k Text File** command. The .$et file is created (and the previous .$et file is overwritten) every time you save your model. The .e2k files are only created when you use the **File**

menu > Export command, or when you copy a .$et file and give it a .e2k extension, or when you create one from scratch. We do not recommend that you try to create a text input file for ETABS.

Also note the information about .ebk files (backup files) in the previous "Save and Save As Commands" section.

Import Command

Note:

ETABS6 analysis input files can be imported into ETABS7. ETABS6 Steeler, Conker and Waller input files can not be imported into ETABS7 or later versions.

You can import certain types of files into the program using the **File menu > Import** command. Following are the options available for this command.

- **ETABS8.e2k Text File:** This option is used to import ETABS8 .e2k and .$et text input files. If another model is currently open, this option will close that model (prompting you to save it if necessary) and open a new one for the imported file.
- **ETABS7 Text File:** This option is used to import an ETABS7 text input file. ETABS7 Steeler, Conker and Waller input files are not imported. If another model is currently open, this option will close that model and open a new one for the imported file.
- **DXF File:** This option is used to import a .DXF drawing file.
- **CIS/2 Step File:** This option is used to import a file created using the CIMsteel Integration Standard (CIS). CIS is a set of formal computing specifications used in the steel industry to make software applications mutually compatible. This file type is often used by steel fabricators outside the United States.

Export Command

You can export certain types of files using the **File menu > Export** command. Following are the options available for this command.

Export files for Use in CSI Programs

Note:

You can export the ETABS analysis model to a SAP2000 .s2k file. The element-based analysis model is different from the typical object-based ETABS model. See Chapter 2 for additional information about the differences.

- **Save Model as ETABS8.e2k Text File:** This option saves the model as a .e2k text input file. You can later import this file/model back into ETABS using the **File menu > Import > ETABS8.e2k Text File** command if you wish.

- **Save Model as SAP2000.s2k Text File:** This option saves the analysis model (an element-based model rather than the ETABS object-based model) as a .s2k text input file. You can later import this file/model into SAP2000 if you wish.

  Because the model is converted into an element-based model, you may not recognize the names of the elements. They may be different from the names of the objects in your object-based ETABS model.

  This option can be useful if you want to see exactly what the analysis model looks like for your structure. It is also useful if you want to add certain other special purpose elements to your model that are not available in ETABS, such as Solid elements. Note however that once you export your ETABS model to SAP2000, you cannot then import that SAP2000 model back into ETABS. Thus, you will no longer have access to the specialized design features in ETABS.

- **Save Story as SAFE.f2k Text File:** This command saves the specified story level as a SAFE.f2k text input file. You can later import this file/model into SAFE if you wish.

  The items exported in this case are:

  ✓ Structural objects and their assignments, assuming those assignments are valid in SAFE.

- ✓ Grid line definitions.
- ✓ Loads.
- ✓ Information regarding columns above and below.

This command can be useful if you want to use SAFE to perform a more refined analysis and design of the concrete floor that is in your ETABS model. You **cannot** later import this revised file back into your ETABS model.

- **Save Story as ETABS8 .edb File:** This command saves the specified story level as an ETABS8 .edb. If the story you are exporting is the bottom story of a structure, the restraints are included in the exported one-story model. If the story level exported is not the bottom level of a structure, ETABS fixes the base of all columns and walls in the exported one-story structure.

Export Files for Use in AutoCAD

- **Save as .DXF File:** This command is used to transfer full or selected parts of the model into AutoCAD. It provides easy to use features to customize the drawing before transferring to .DXF by selecting appropriate attributes for various objects/elements and text. A full 3D model, including node symbols and frame members, can be transferred. The program does not export any analysis output results. All items that are exported must be showing in the currently active window when the export operation is executed.

Note:

The .DXF files created by ETABS are compatible with AutoCad 2000.

Tip:

When exporting a plan view to a .DXF file all items that are to be exported must be visible in a plan view in the currently active window.

Note that plan views that occur either at story levels or at reference plane elevation can be exported to a .DXF file. Only the elements that occur in the horizontal plane of the plan view are exported. Other elements that are a part of the story level that is associated with the plan view are not exported.

The types of items listed below can be exported to a .DXF file in a plan view. A layer name can be specified for each of those types of items. You can put several different types of items on

the same layer if you wish. If you do not want to export a particular type of item even though it is showing in the plan view, set its layer name to None.

- ✓ **Grid lines:** These are exported to the .DXF file as lines. This item includes the grid line and the grid ID bubble if it exists, but not the grid ID. The grid ID is exported with the text.
- ✓ **Walls:** These are exported to the .DXF file as polylines so that their width can be graphically shown.
- ✓ **Beams:** These are exported to the .DXF file as lines. You can also specify a beam offset. This is the distance between the end of the beam and its supporting girder or column in the .DXF file.

Note:

ETABS does not export analysis results to .DXF files.

- ✓ **Moment connection block:** This is exported to the .DXF file as a block. It includes the moment connection symbols for the beams. The moment connections will only export if they are visible in the currently active window. Use the Moment Connections check box in the Special Frame Items area of the Set Building View Options form to toggle the display of moment connection symbols on and off. You can use the **View menu > Set Building View Options** command to access this form.
- ✓ **Links:** These are exported to the .DXF file as lines.
- ✓ **Slab/deck perimeter:** This is exported to the .DXF file as a polyline.
- ✓ **Column block:** This is exported to the .DXF file as a block. It shows the shapes of the columns as they appear in the plan view.
- ✓ **Dimension lines:** These are exported to the .DXF file as dimension lines.

- ✓ **Text:** Text is exported to the .DXF file as text. Only text that is visible in the currently active window is exported. Text associated with analysis output is not exported.
- ✓ **Grid lines:** These are exported to the .DXF file as lines. This item includes the grid line but not the grid ID bubble or the grid ID text. Grid lines exported in this way are only shown at the base of the building.
- ✓ **Areas:** Area objects are exported to the .DXF file as polylines.
- ✓ **Lines:** Line objects are exported to the .DXF file as lines. You cannot specify a beam offset for a 3D .DXF file like you can for a plan view .DXF file.

Export Files for Use in Access

- **Save Input/Output as an Access database file:** This command exports all of the model input and analysis output as tables in a Microsoft Access database file (.mdb file) that is compatible with Microsoft Access 97.

Export Files to an Enhanced Metafile

- **Save Graphics as Enhanced Metafile:** This command exports all that is graphically showing in the currently active window to a Windows enhanced metafile (.*.emf file).

Export Files to a CIS/2 Step File

- **CIS/2 STEP File**: This command is used to export a file using the CIMsteel Integration Standards (CIS). CIS is a set of formal computing specifications used in the steel industry to make software applications mutually compatible. This file type is often used by steel fabricators outside the United States.

Export Files to a Steel Detailing Neutral File

- **Steel Detailing Neutral File:** This command is used to export a file using the Steel Detailing Neutral File computing specification standards. This file type is often used by steel fabricators in the United States.

Create Video Command

You can create videos in the program showing the movement of the structure during any time history analysis you have run. You can also create videos showing animations of mode shapes and other deformed shape plots of the structure. The videos are saved as .avi files. They can be played back using the Window's media player.

Time History Animation Command

Tip:

You can create videos of time history response in ETABS (as .avi files) and then play back the videos.

Use the **File menu > Create Video > Time History Animation** command to create videos of time histories. Note the following for time history videos:

- The magnification factor controls how large the deformations appear in the video.
- To record the time history file in real time animation, make sure that the number of frames per second is equal to one over the time increment. If you want to record the time history video in slow or fast motion, the value of number of frames per second may be adjusted up or down to speed the animation up or slow it down.

  The time increment controls how many different pictures (frames) of the deformed shape of the structure are created. For example, a time increment of 0.1 means a picture (frame) of the deformed shape is created for every one-tenth second of the time history. The Frames per Second item controls how fast the time history is played back.

- For best results make sure that all program windows are showing undeformed views before attempting to create a time history video.

Cyclic Animation Command

Use the **File menu > Create Video > Cyclic Animation** command to create videos of animated mode shapes and other deflected shapes. A mode shape or deformed shape must be showing in the active window for this command to be available.

Print Setup Command

Clicking the **File menu > Print Setup** command brings up the Print Page Setup form. This form has two selection areas:

- **Lines Per Page** area has a No Page Ejects check box and Default and User Defined selections. If the No Page Ejects check box is checked, the print out will be continuous. Use this check box if your printer has continuous feed paper, which is often referred to as "computer paper." The Default selection will print 56 lines per page, which is the standard length for 8 1/2 x 11 inch paper. The User Defined selection allows you to enter your own number of lines per page.

- **Titles** area allows you to add a Project Title and a Data Title to your print out. Click in the edit box and type your desired title. These titles will appear at the top of each page of the printout.

A Color Printer (Graphics) check box is also on the form; checking this check box tells the printer to use colors in the print out. Remember to use a printer capable of printing in color to take advantage of this selection.

Click the **OK** button to accept the selection you have made on the form. Clicking the **Cancel** button cancels any selection made.

The Setup button brings up the Print Setup form. This form allows you to select the printer, the paper size and the paper orientation for your print out.

Print Preview for Graphics Command

Clicking the **File menu > Print Preview for Graphics** command brings up the Print Preview form. This form provides a snap shot of how the file will print in graphical format. The form is equipped with zoom tools that enable you to zoom into the snap shot for a closer look at your print out before printing.

Note:

*The **File menu > Print Preview for Graphics** command allows you to preview the printed output for graphics before actually printing it.*

The form has a drop-down box with various selections for sizing your print out. For example, the Print to Page selection will print your graphics to fit the size of paper defined for your printing device. You can also select specific scaling for your printout, such as 1 inch equals 1 foot, 1 inch equals 2 feet and so forth. A User Scale selection is also available that allows you to pick your own scaling for the output. This scaling applies only to the print out and has no effect on the model file itself.

To close the Print Preview form, select either the Close button, which will return you to your model without printing, or the Print button, which will send your print to your printer.

Print Graphics

The **File menu > Print Graphics** command or toolbar button prints whatever graphics are displayed in the active window to the printer that is currently specified as active. The printer used may be either a black and white printer or a color printer. The gray scales or colors used for displaying various objects in the print out are controlled in the Assign Display Colors form, which is accessed using the **Options menu > Colors** command.

The colors (grayscales) used for black and white printers are those displayed in the Assign Display Colors form when the De-

vice Type option is set to Printer. The colors used for color printers are those displayed in the Assign Display Colors form when the Device Type option is set to Color Printer. The colors used for display on the screen are those displayed in the Assign Display Colors form when the Device Type option is set to Screen. If the object display colors are set differently for screen display and color printing, the objects' colors in print will be different from their colors displayed on screen.

Print Tables Command

You can use the **File menu > Print Tables** command to print text tables either to a printer or to a text file. The following types of tables can be printed using this command:

- Analysis input data
- Analysis output data
- Design input and output data for steel frame design, concrete frame design, composite beam design and shear wall design.

When you use the **File menu > Print Tables** command, note the following:

- If you select some objects before executing the **File menu > Print Tables** command, the printed output will be for the selected objects only.
- If you *do not* select some objects before executing the **File menu > Print Tables** command, the printed output will be for all objects in the model.

Input Command

Use the **File menu > Print Tables > Input** command to print tables of analysis input data to a printer or to a text file. This command brings up the Print Input Tables form where you can specify the types of input data that you want to print.

Note that you can use the **Display menu > Set Input Table Mode** command to display similar data in a database format on the screen. Data displayed in this manner cannot be printed.

Analysis Output Command

Use the **File menu > Print Tables > Analysis Output** command to print tables of analysis output data to a printer or to a text file. This command brings up the Print Output Tables form where you can specify the types of output data that you want to print.

Note that you can use the **Display menu > Set Output Table Mode** command to display similar data in a database format on the screen. Data displayed in this manner cannot be printed.

Design Tables Commands

The program has the capability to generate output table files for Steel Frame, Concrete Beam, Composite Beam and Shear Wall. Use the **File menu > Print Tables,** and then the respective option command (**Steel Frame Design** command, **Concrete Frame Design** command, **Composite Beam Design** command, or **Shear Wall Design** command) to print tables of design output data to a printer or to a text file.

User Comments and Session Log Command

The **File menu > User Comments and Session Log** command brings up a text window where you can type in any comments that you want to make. Those comments are saved with your model and can be accessed and added to, modified or deleted at any time using the **File menu > User Comments and Session Log** command. Note that the program also occasionally adds comments to this file. You can modify or delete those comments as well.

Last Analysis Run Log Option

Use the **File> Last Analysis Run Log** option to open and view the analysis log file of the last analysis. The file is saved as plain text with a .$og extension. It can be opened using any text viewer/editor, including Notepad. The log file contains useful overall information about the model, analysis and the output. The information cover the statistics of the model (for example, the total number of elements, nodes, supports, load cases, and other similar data), solution sequences, number of equations, storage space required and available solution status, errors, warnings and successful completion of the analysis as well as other pertinent information. The file also shows the number of iterations and convergences obtained at each step for nonlinear and Eigen value solutions. The user can open this file at any time after the analysis has been run to identify and understand any analysis problem such as instability, numerical errors, or the like.

Display Input/Output Text Files Command

The **File menu > Display Input/Output Text Files** command provides a convenient way for you to view input and output text files associated with this program. The input text files you might want to view include those with the .$et and .e2k extensions and perhaps any time history or response spectrum text files that you are using. The output files that you might want to view are those where you have printed output to a file rather than to a printer. Typically, those files have a .txt extension.

When you click the **File menu > Display Input/Output Text Files** command and select a file to be displayed, the program opens the text file in the Windows WordPad program.

Delete Analysis Files Command

You can use the **File menu > Delete Analysis File** command to delete temporary analysis files associated with the current model.

After obtaining the results, use of this command will "free up" hard drive capacity.

4 Exit Command

Use the **File menu > Exit** command to exit the program. Also exit by clicking the X in the upper right-hand corner of the window, or right clicking the program button on your Windows taskbar and choosing **Close** from the popup menu. If the model has changed since it was last saved, the program will prompt you to save your model before exiting.

Note that using these ways to exit the program not only closes the model, but also closes the entire program. To close one model and open another (or start a new one), click on the appropriate command in the File menu to open another model or to start a new one. If you have made changes to your model since you last saved it, the program will prompt you to save your model before beginning work on the next file.

Chapter 5

Edit Menu

General

The Edit menu provides basic tools for editing (modifying) the geometry of your model. This chapter describes many of those tools.

Undo and Redo Commands

Clicking the Undo option or button or holding down the Ctrl key and typing Z deletes the last performed action. The Redo option, button, or Ctrl Y allows the user to restore the last step that was undone. Some commands, such as **Auto Relabel All**, cannot be undone.

Cut, Copy and Paste Commands

In general, the cut, copy and paste commands are similar to the standard cut, copy and paste Windows commands. However, some behavior of the commands is specific to this program.

The cut, copy and paste commands are only active when the currently active window is in plan or plan perspective view.

Tip:

The Cut and Copy commands copy geometry and property names only. Use the ***Edit menu > Replicate*** *command to copy an object and its assignments.*

- The **Cut** command, button, or Ctrl X deletes the selected objects of the story level shown in plan view of the active window. When the objects are deleted, all of their assignments are also deleted. The geometry of the objects are copied to the Windows clipboard. Additionally, the names of any frame section properties assigned to selected line objects or the names of any wall/slab/deck section properties assigned to selected area objects are copied to the clipboard along with the geometry. No other assignments to the object are copied to the clipboard. The geometry and section property names associated with the cut objects can be pasted back into the program, or they can be copied to a spreadsheet, such as Microsoft Excel, in a text format. Additional description of the spreadsheet option is provided later in this section.

- The **Copy** command, button, or Ctrl C copies to the Windows clipboard; this command copies the geometry of the selected objects of the story level shown in plan view of the active window. The names of any frame section properties assigned to selected line objects or the names of any wall/slab/deck section properties assigned to selected area objects are copied to the clipboard along with the geometry. The geometry and section property names associated with the copied objects can be pasted back into the program, or they can be copied to a spreadsheet, such as Microsoft Excel, in a text format. Additional description of the spreadsheet option is provided later in this section.

- The **Paste** command, button, or Ctrl V copies geometry and section property names from the Windows clipboard into

Note:

You can edit geometry in a spreadsheet and then copy and paste it into the program.

the model on the story level shown in plan or plan perspective view in the currently active window. The geometry and section property names that are on the Windows clipboard may have been copied to the clipboard from this program or from a spreadsheet.

It is important to note that the **Cut** and **Copy** commands only copy the geometry and property name of the selected object to the Windows clipboard. Other assignments made to the selected object are not copied using these commands.

Using a Spreadsheet to Create or Modify Model Geometry and Section Properties

You can edit geometry in a spreadsheet and then copy and paste it into the program. Again note that you can only create and/or modify geometry and some section properties in this fashion. You cannot make assignments (loads, supports, end offsets, or the like) through spreadsheet input.

You can see the text format used when the program geometry is copied to or from a spreadsheet by selecting a portion of a model, clicking **Edit menu > Copy** to copy the selected geometry to the clipboard, and then opening a spreadsheet and using the **Paste** command in the spreadsheet to paste the geometry data into the spreadsheet. The appropriate toolbar buttons or the Ctrl key strokes can also be used for these actions.

In the spreadsheet, each object is described in one line. Following are descriptions of the column headings for each of the three object types (point, line and area).

Point Object

The point object data copied from the model and into a spreadsheet or copied from the spreadsheet into the model using the Copy command is organized in the spreadsheet under the following column headings:

Heading	Description - Point Objects
Type:	This is always POINT for point objects.
X:	This is the global X coordinate of the point object.
Y:	This is the global Y coordinate of the point object.
DZ:	This is the Z-direction distance from the story level associated with the plan view where the object is pasted into the program to the point object. DZ is 0 if the point object is at the associated story level. Otherwise, it is a *positive* number measured from the story level down. Note that if the currently active plan view is showing a reference plane, a DZ value of zero pastes the object in at the story level associated with the reference plane, not at the reference plane level. See the section entitled "Edit Reference Planes and Edit Reference Lines Commands" later in this chapter for a description of reference planes.

Line Objects

The line object data copied from the model and into a spreadsheet or copied from the spreadsheet into the model using the Copy command is organized in the spreadsheet under the following column headings:

Heading	Description - Line Objects
Type:	This is always LINE for line objects.
Section:	This is the name of the frame section property assigned to the line object. If no frame section property is assigned to the line object, it is NONE. If a line object has a frame section property name in a spreadsheet and the frame section property

Heading	Description - Line Objects
	has not already been defined in the model, when spreadsheet data is pasted into the model, the program sets the frame section property assignment for that object to NONE. For example, assume a line object has a frame section property name of COL1 in a spreadsheet. Also assume that no previously defined frame section property named COL1 is included in the model. In that case, when the spreadsheet data is pasted into the model, the program sets the property assignment for that line object to NONE.
The following items are provided for each end point of the line object:	
XI (XJ):	This is the global X coordinate of the considered end point of the line object.
YI (YJ):	This is the global Y coordinate of the considered end point of the line object.
DZI (DZJ):	This is the Z-direction distance from the story level of the plan view where the line object is pasted into the program at the considered end point of the line object. DZ is 0 if the end point is at the associated story level. Otherwise, it is a *positive* number measured from the story level down. Note that if the currently active plan view is showing a reference plane, a DZ value of zero pastes the object at the story level associated with the reference plane, not at the reference plane level. See the section entitled "Edit Reference Planes and Edit Reference Lines Commands" later in this chapter for a description of reference planes.
BelowI	This is a flag that indicates if the considered end point of the line object is at the story level below the

Heading	Description - Line Objects
(BelowJ):	story level associated with the plan view where the line object is to be pasted into the model. This item can either be Y for Yes or N for No. Y means it is at the story level below where it is pasted; N means it is not. When the item is Y, any value input for DZ is ignored and the end point is simply placed at the story level below where it is pasted into the model. An example where this is used is for a column. The bottom end point of a column typically has this flag set to Y. This way a column can be pasted into a story of any height and it will always span from story level to story level.

Area Objects

The area data copied from the model and into a spreadsheet or copied from the spreadsheet into the model using the Copy command is organized in the spreadsheet under the following column headings:

Heading	Description - Area Objects
Type:	This is always AREA for area objects.
Section:	This is the name of the wall, slab or deck section property assigned to the area object. If no section property is assigned to the area object, it is NONE. If an area object has a property name in a spread sheet and the property has not already been defined in the model, when the spreadsheet data is pasted into the model, the program sets the property assignment for that object to NONE. For example, assume that an area object has a property name of WALL1. Also assume that no previously defined wall, slab or deck section property named

Heading	Description - Area Objects
	WALL1 is included in the model. In that case, when the spreadsheet data is pasted into the model, the program sets the property assignment for that area object to NONE.
Points:	This is the number of corner points in the area object.
The following items are provided for each corner point, n, of the area object where n represents a number 1 through the number of corner points in the area object:	
X-n:	This is the global X coordinate of the considered corner point of the area object.
Y-n:	This is the global Y coordinate of the considered corner point of the area object.
DZ-n:	This is the Z-direction distance from the story level associated with the plan view where the area object is pasted into the model to the considered corner point of the area object. DZ is 0 if the corner point object is at the associated story level. Otherwise, it is a *positive* number measured from the story level down. Note that if the active plan view is showing a reference plane, a DZ value of zero pastes the object in at the story level associated with the reference plane, not at the reference plane level. See the section entitled "Edit Reference Planes and Edit Reference Lines Command" later in this chapter for description of reference planes.
Below-N:	This is a flag that indicates if the considered corner point of the area object is at the story level below the story level associated with the plan view where the area object is pasted into the model. This item

Heading	Description - Area Objects
	can either be Y for Yes or N for No. Y means it is at the story level below where it is pasted; N means it is not. When the item is Y, any value input for DZ is ignored and the corner point is simply placed at the story level below where it is pasted into the model. A wall is an example of where this feature is used. The bottom corner points of a wall typically have this flag set to Y. This way a wall can be pasted into a story of any height and it will always span from story level to story level.

Delete Command

In general, the **Delete** command or toolbar button in this program works like the standard Windows delete command. This command deletes the selected object(s) and all of its assignments (loads, properties, supports and the like). Alternatively, select the object(s) and press the Delete key on the keyboard to delete the selected object(s).

Add to Model from Template Command

Use the **Edit menu > Add to Model from Template** command to add two-dimensional and three-dimensional frames to your model.

The two-dimensional option can be used to locate planar frames throughout a model. The three-dimensional option can assist in modeling conditions where several towers rest on the same base structure.

Add to Model From Template > Add 2D Frame Command

Click the **Add to Model From Template > Add 2D Frame** command to add a two-dimensional frame or wall to your model. When the 2D Frame form appears, select wall or frame using the **Click Picture of Choice** buttons. This brings up the Portal Fame form, in which you specify the following data:

- Number of stories. The two-dimensional frames are always assumed to start at the base of the building and extend upward. This is different from the three-dimensional frame option.
- Number of bays and typical bay width.
- Typical properties for columns and beams in a frame or typical property for a wall.
- Location in plan. This is the location of one end of the frame in global X and Y coordinates. You also specify a plan orientation angle for the frame in degrees. The angle is measured in the global X-Y plane from the positive global X-axis with positive angles counterclockwise when you look down on the model.
- The base restraints are specified as pinned, fixed or none.

Add to Model From Template > Add 3D Frame Command

Click the **Add to Model From Template > Add 3D Frame** command to add a three-dimensional frame to your model. When the Structural Floor System Type form appears, select the frame type to be added to your model by clicking the appropriate button in the Click Picture of Choice area of the form. You may recognize the buttons because they are the same as the buttons used when you start a model from a template. With the exception of the location data, which is described here for this command, please refer to the subsection entitled "Add Structural Objects" under the "New Model Command" section of Chapter 4 File Menu for a description of these buttons and their functions. The

following information is provided to locate the three-dimensional frame:

- A coordinate system (grid system) is specified. By default, the added three-dimensional frame fills all of the specified bays and story levels in this coordinate system (grid system). If you want to fill only some of the bays or story levels in the specified coordinate system (grid system), click the **Advanced** button and fill in the starting and ending grid line IDs and the story levels at top and bottom of the frame.

Note that unlike an added two-dimensional frame, an added three-dimensional frame can start at a story level above the base level.

Replicate Command

Note:

Replication copies objects and their assignments.

To replicate one or more objects and most of their assignments, select the object(s) and use the **Edit menu > Replicate** command or toolbar button to specify the desired replication option. The selected objects, including their assignments, will be replicated (copied) as specified.

Important note: When using the replication feature, if an object to be replicated is targeted to occupy exactly the same location as an existing object, the replication is not completed at that location; the existing object remains. However, if other locations are targeted and those locations do not contain objects, the replication is completed at those other locations.

Clicking the **Edit menu > Replicate** command or the toolbar button brings up the Replicate form. This form has a tab for each type of replication that is available: Linear, Radial, Mirror, and Story. Each of these replication types is subsequently described in separate subsections.

Clicking the **Options** button on any of the tabs brings up the Replicating Objects Assignment form, which consists of a series of check boxes for each object type (point, line, and area). Use

this form to control which assignments you want to replicate by checking or unchecking the boxes. By default all assignments are replicated, except for the following, which are never replicated:

- Rigid diaphragm assignments to point objects and area objects.
- Pier label assignments to line and area objects.
- Spandrel label assignments to line and area objects.

For area and line objects, some assignments are always replicated and you have no control over them. Table 5-1 lists those assignments.

Table 5-1: Area and Line Object Assignments That Are Always Replicated

Line Object Assignments	Area Object Assignments
Frame section property	Section property
End releases (not partial fixity)	Opening
Output stations	Local axes
Local axes	Automatic mesh/no mesh
Automatic mesh/no mesh	

Table 5-2 lists the area, line and point object assignments for which you can control replication.

Table 5-2: Object Assignments with Replication That Can be Controlled

Point Object Assignments	Line Object Assignments	Area Object Assignments
Panel zones	Additional masses	Additional masses
Restraints (supports)	Line springs	Area springs
Additional masses	Partial fixities	Stiffness modifiers
Point springs	End and joint offsets	Uniform loads

Table 5-2: Object Assignments with Replication That Can be Controlled

Point Object Assignments	Line Object Assignments	Area Object Assignments
Link properties	Link properties	Temperature loads
Forces	Nonlinear hinges (pushover)	
Ground displacements	Property modifiers	
Temperatures	Point loads	
	Distributed loads	
	Temperature loads	

Linear Tab

For linear replication, specify a dx, dy and a number of times the object is to be replicated on the Linear Tab of the Replicate form. The object and its assignments are then copied the specified number of times, incrementing the global X and Y coordinates by the specified dx and dy.

Radial Tab

For radial replication, specify on the Radial Tab a point to rotate about (the rotation is in the global X-Y plane about the global Z-axis), a rotation angle, and a number of times the object is to be replicated. The object and its assignments are then copied the specified number of times, incrementing the location of the objects by the specified rotation angle.

Two Rotate About Point options are available: rotate about the center of the selected objects, or specify global X and Y coordinates of a specific point to rotate about.

When the rotation occurs about the center of the selected objects, the program calculates the location of that point as follows. ETABS determines the maximum and minimum global X-coordinate of all selected objects. The global X-coordinate of the center of the selected objects is determined as the average of the coordinates of the maximum and minimum X coordinates. The

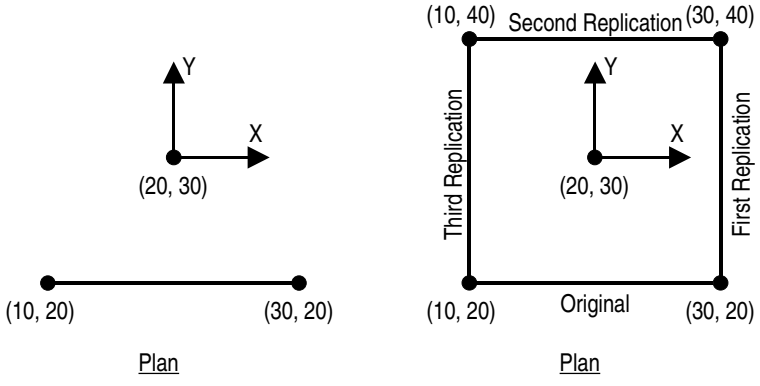


Figure 5-1: *Example of radial replication*

a) Original Objects **b) Replicated Objects**

global Y-coordinate of the center of the selected objects is determined in a similar manner.

The rotation angle is input in degrees. Angles are measured from the positive global X-axis. Positive angles appear counterclockwise when you view them from above.

Figure 5-1 shows an example of radial replication. Figure 5-1a shows a plan view of a frame that extends from the point (10, 20) to the point (30, 20) where the coordinates are given in the global coordinate system. The frame is selected and radial replication is specified about the point (20, 30). The angle is set to 90 degrees and the number is set to 3. Figure 5-1b shows the result of the replication.

Mirror Tab

For mirror replication, specify on the Mirror Tab a line in the global X-Y plane to mirror about, or if you prefer, think of it as a vertical plane to mirror about. The vertical plane is defined by the specified line in the global X-Y plane and vertical line, parallel to the global Z-axis, that intersects the specified line.

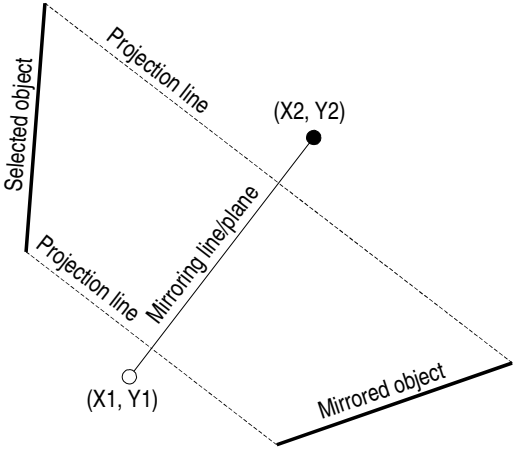


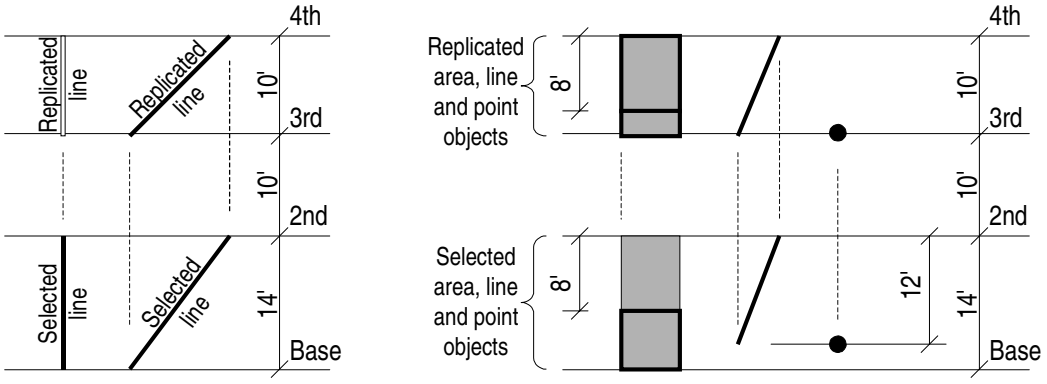
Figure 5-2:
Example of mirror replication

You specify the line in the X-Y plane by specifying two points (X1, Y1) and (X2, Y2) in global coordinates. The program replicates the selected objects by mirroring the objects and their assignments about the specified line/plane. Figure 5-2 illustrates the mirroring process. Note that the projection lines used in the mirroring process (shown dashed in the figure) are perpendicular to the specified mirroring line/plane.

Story Tab

For story replication, specify on the Story Tab a story that the selected objects are to be replicated on. The object and its assignments are then copied to that story level. If the story level where you select the objects and the story level to which you replicate the objects have different story heights, be aware of the following:

- Elements that extend from one story level to the next still extend from one story to the next when they are replicated, even if the story heights are different. Figure 5-3a shows some examples.
- Distances are measured from the top of the story down. If an object that is to be replicated to story level X is below the



a) Elevation **b) Elevation**

(Above)
Figure 5-3:
Examples of story replication

bottom of story level X, that object is placed at the bottom story level X; that is, it is placed at story level (X - 1). This can happen when you are replicating from a story level that is taller than the story level you are replicating to.

Figure 5-3b shows some examples. Note that the height of the lower replicated area object on the left side of Figure 5-3b is reduced from 6 feet to 2 feet when it is replicated to fit into the 4th story level. Also note that the line object is taken to the bottom of the story level because the 12-foot dimension that it is to be replicated to exceeds the story height. Notice that the plan location of the bottom of the line object remains the same at each level; thus the slope of the line object is different at the two levels. Similar to the line object, the point object is placed at the bottom of the 4th story level.

Delete Original Check Box

If the Delete Original check box is checked, the program will delete the object selected for replication as part of the replication operation, leaving only the replicated object(s).

Edit Grid Data Command

The **Edit menu > Edit Grid Data** command is used to define new coordinate systems, modify existing coordinate systems and edit the grid line data associated with the coordinate systems. The default global coordinate/grid system is a Cartesian (rectangular) coordinate system. Additional coordinate/grid systems can be defined that are either Cartesian or Cylindrical. Cylindrical coordinate systems are based on a set of radial and circumferential grid lines. The following four options, described individually, are available when you select the **Edit menu > Edit Grid Data** command:

Edit Grid Data > Edit Cartesian/Cylindrical System Command

Note:

In ETABS you can have Cartesian (rectangular) and/or cylindrical coordinate/grid systems.

Clicking the **Edit menu > Edit Grid Data > Edit Cartesian/cylindrical System** command or the toolbar button brings up the Coordinate System form, where you can add, modify, or delete a coordinate system.

Add New System Button

Clicking the **Add New System** button on the Coordinate System form will open the Coordinate System Definition form. There are several areas in this form. The first frame offers a field for the name of the system. Initially, a default name is given, which can be changed.

You can specify the grid type as Cartesian or Cylindrical. If the Cartesian option is selected, specify the number of grid lines and the line spacing in the X and Y directions. The grids will be drawn using this spacing in a uniform manner. If the Cylindrical option is selected, input the grid spacing and number of grids in the radial and theta directions. Here too the grids will be drawn with uniform spacing.

Edit Grid Button. To specify non-uniform spacing or change other default values, click the **Edit Grid** button. In the Coordi-

nate System form that appears, change the spacing in the X or Y directions (or Radius and Theta) by first clicking the X Grid or Y grid option (or Radius and Theta) in the Display area. The table on the left-hand side of the form will display the selected grids with their coordinates. If you choose to view the spacing rather than the coordinates, select the Spacing option under the Display Grid as area. The values that appear in the table are editable.

Add Primary or Secondary grid lines by selecting the appropriate option under the area labeled Grid Line # 1. Also change display settings of the grid such as color, bubble location and the like in this area. The bubble size can also be edited. Each of these selections is described as follows:

- ✓ **Grid ID:** This is an identifier for the grid line. It can be left blank if desired.
- ✓ **Coordinate/Spacing:** This is the location of the grid line in the specified coordinate system. Grid line locations can be specified by their coordinate or by their spacing.
- ✓ **Primary/Secondary Grid Lines:** You can specify a grid line as either a primary or a secondary line. Primary grid lines are intended to represent the main architectural grid lines of the building. Secondary grid lines are intended as temporary reference lines for modeling. They do not have a bubble assigned to them for the grid ID. You can use the **View menu > Set Display Options command** to collectively hide the secondary grid lines from view.
- ✓ **Hide Grid Line:** Checking this box for either primary or secondary grid lines marks them as hidden and they are not displayed, regardless of the setting specified in the **View menu > Set Display Options** feature.
- ✓ **Switch Bubble Location:** Checking this box moves the bubble location to the other end of the grid line.
- ✓ **Color:** Clicking on the Color box allows you to change the color of the grid line.

- ✓ **Bubble Size:** The grid line bubbles are drawn as hexagons. The bubble size indicates the distance between two opposing faces of the hexagon measured in the current units of the model. This size applies to all bubbles in the coordinate/grid system.

- ✓ **Hide All Grid Lines:** If this box is checked, all grid lines (primary and secondary) *in this coordinate/grid system only* are hidden. If you want to hide grid lines in all coordinate systems, use the **View menu > Set Building View Options command**. When using the **View menu > Set Building View Options command,** be sure to turn off both primary and secondary grid lines.

- ✓ **Reset to Default Color:** Clicking this button sets the color of *all* grid lines in this coordinate/grid system back to their default color. Note that you can use the **Option menu > Colors command** to set the default color for grid lines.

- ✓ **Locate System Origin:** Clicking this button brings up a form that allows you to specify the location of the origin of the coordinate system relative to the global coordinate system. Note that this button is not visible if you are editing the global coordinate/grid system. In the Locate System Origin form you specify the following:

  - ❑ **Global X:** This is the global X coordinate of the origin of the coordinate/grid system.

  - ❑ **Global Y:** This is the global Y coordinate of the origin of the coordinate/grid system.

  - ❑ **Rotation (deg):** This angle, input in degrees, specifies the orientation of the positive X-axis (or theta equals 0 degrees radial line in a cylindrical system) of the coordinate system relative to the positive global X-axis. The angle is measured from the positive global X-axis to the coordinate system X-axis. Positive angles appear counterclockwise when viewed from above.

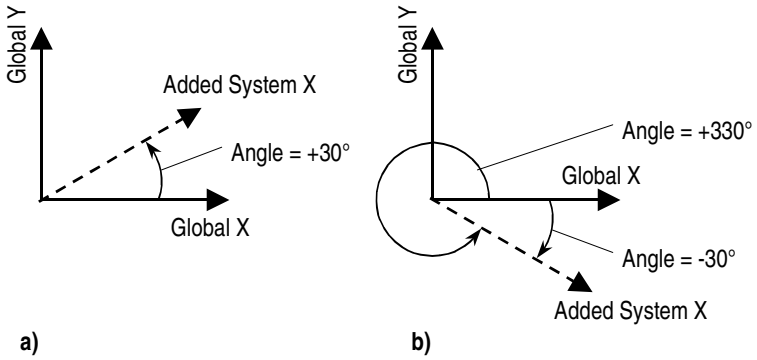


Figure 5-4: *Example of rotation angle used to specify orientation of added coordinate/grid systems relative to the global coordinate/grid system*

Figure 5-4 shows some examples. The figure shows the global X-axis and the orientation of the X-axis for the added system. Figure 5-4a shows an example where the rotation angle is specified as 30 degrees. Figure 5-4b shows an example where the rotation angle could be specified as either 330 degrees or -30 degrees.

Grid Labels Button. Clicking on the **Grid Labels** button of the Coordinate System Definition form will open the Grid Labeling Options form. All of the features on this form, which include the starting ID of the grid in either direction and the direction of labeling (left to right, right to left, top to bottom or bottom to top), are self-explanatory.

Modify/Show System Button

Clicking the **Modify/Show System** Button on the Coordinate System form will bring up the **Coordinate System** form. Change the spacing in the X or Y directions (or Radius and Theta) by first clicking the X Grid or Y grid option (Radius or Theta) in the Display area. The table on the left-hand side of the form will display the selected grids with their coordinates. If you choose to view the spacing rather than the coordinates, select the Spacing option under the Display Grid as area. The values that appear on the table are editable. You can also add Primary or Secondary grid lines by selecting the appropriate option under the area labeled Grid Line # 1. You can also change display settings of the

grid, such as color, bubble location, and the like in this area. The bubble size can also be edited here. All of the options available on this form are the same as those described in the previous section entitled "Add New System Button."

Delete System Button

Any system, except the Global system, can be deleted by selecting the system and clicking the Delete System button.

Edit Grid Data > Edit General System Command

Clicking the **Edit menu > Edit Grid Data > Edit General System** command brings up the Grid System form. In the Grid Systems form, you can add a new system, add a copy of a system, modify and show the list of systems, and delete a system by selecting the appropriate button. Each of the buttons is described s follows:

Add New System Button

Clicking the **Add New System** button on the Grid System form will open the Grid System Data form. The form has the following area:

- ✓ **Grid System Name:** Specifies the name of the grid system. Accept the default name or specify a different one.

- ✓ **Click to:** Adds a new grid line or modifies or deletes an existing grid line. Clicking the **Add New Grid Line** button (or **Modify/Show Grid Line** button if a grid line already exists) will bring up the Grid Line form. Here you can specify the Grid Line ID, Type of Grid (Straight or Arc) and its Plan Coordinates. You can also select the story level at which the grid line is to be included. Checking the Apply to All check box will include the change in all the levels that lie between the top and bottom story level selected. Other options available include:

- **Primary/Secondary Grid Lines:** Specify a grid line as either a primary or a secondary line. Primary grid lines are intended to represent the main architectural grid lines of the building. Secondary grid lines are intended as temporary reference lines for modeling. They do not have a bubble assigned to them for the grid ID.

- **Hide Grid Line:** Hides the primary or secondary grid lines depending on which check box is checked.

- **Switch Bubble Location:** Moves the bubble location to the other end of the grid line.

- **Color:** Changes the color of the grid line.

- **Bubble Size**: The grid line bubbles are drawn as hexagons. The bubble size indicates the distance between two opposing faces of the hexagon measured in the current units of the model. This size applies to all bubbles in the coordinate/grid system.

✓ **System Origin**: The System Origin portion of the Grid System Data form allows you to specify the location of the origin of the coordinate system relative to the global coordinate system. Specify the following using the System Origin form:

- **Global X:** This is the global X coordinate of the origin of the coordinate/grid system.

- **Global Y:** This is the global Y coordinate of the origin of the coordinate/grid system.

- **Rotation (deg):** This angle, input in degrees, specifies the orientation of the positive X-axis (or theta equals 0 degrees radial line in a cylindrical system) of the coordinate system relative to the positive global X-axis. The angle is measured from the positive global X-axis to the coordinate system X-axis. Positive angles appear counterclockwise when viewed from above.

Add Copy to System Button

Before clicking the **Add Copy to System** button, select the system you wish to copy from the Systems list box on the left-hand side of the form. Clicking the **Add Copy to System** button brings up the Grid System Data form. This form has options that allow you to add a copy of the selected grid system to the model without changing it (click the **OK** button), or to modify all or portions of the system, such as adding to the system (click the **Add New Grid Line** button) modifying grid lines (click the **Modify/Show Grid Line** button), or deleting grid lines from the system (click the **Delete Grid Line** button), before adding the copy to your model.

The **Add New Grid Line** button works as described in the previous section entitled "Add New Grid Line Button." The **Modify/Show Grid Line** button works as described in the following section entitled "Modify/Show System Button." The **Delete Grid Line Button** works as described in the subsequent section entitled "Delete Grid Line Button."

Modify/Show System Button

Before clicking the **Modify/Show System** button, select the system you wish to modify or show from the Systems list box on the left-hand side of the form. Clicking the **Modify/Show System** button brings up the Grid System Data form. This form has the following area:

- ✓ **Grid System Name:** Specifies the name of the grid system. Accept the default name or specify a different one.

- ✓ **Click to:** Adds a new grid line or modifies or deletes an existing grid line. Clicking the **Add New Grid Line** button (or **Modify/Show Grid Line** button if a grid line already exists) brings up the Grid Line form. Here you can specify the Grid Line ID, Type of Grid (Straight or Arc) and its Plan Coordinates. You can also select the story level at which the grid line is to be included. Checking the Apply to All check box will include the change in all the levels that lie between the

top and bottom story level selected. Other options available include:

- **Primary/Secondary Grid Lines:** Specify a grid line as either a primary or a secondary line. Primary grid lines are intended to represent the main architectural grid lines of the building. Secondary grid lines are intended as temporary reference lines for modeling. They do not have a bubble assigned to them for the grid ID.
- **Hide Grid Line:** Hides the primary or secondary grid lines depending on which check box is checked.
- **Switch Bubble Location:** Moves the bubble location to the other end of the grid line.
- **Color:** Changes the color of the grid line.
- **Bubble Size**: The grid line bubbles are drawn as hexagons. The bubble size indicates the distance between two opposing faces of the hexagon measured in the current units of the model. This size applies to all bubbles in the coordinate/grid system.

✓ **System Origin**: The System Origin portion of the Grid System Data form allows you to specify the location of the origin of the coordinate system relative to the global coordinate system. Specify the following using the System Origin form:

- **Global X:** This is the global X coordinate of the origin of the coordinate/grid system.
- **Global Y:** This is the global Y coordinate of the origin of the coordinate/grid system.
- **Rotation (deg):** This angle, input in degrees, specifies the orientation of the positive X-axis (or theta equals 0 degrees radial line in a cylindrical system) of the coordinate system relative to the positive global X-axis. The angle is measured from the positive global X-axis to the

coordinate system X-axis. Positive angles appear counterclockwise when viewed from above.

Delete System Button

Clicking the Delete System Button deletes the selected system. If only one system is defined, this option is inactive.

Edit Grid Data > Convert System to General Command

Clicking the **Edit Menu > Edit Grid Data > Grid Systems** command brings up the Convert Cartesian/Cylindrical System to General form. This form enables you to convert previously defined Cartesian/Cylindrical systems to a single General System. Converting to the General System groups all previously defined systems into one system.

Edit Grid Data > Glue Points to Grid Lines Command

The **Edit Menu > Edit Grid Data > Glue Points to Grid Lines** command is a toggle switch. As the name suggests, when enabled, this command allows you to "glue" point objects that fall directly on grid lines to those grid lines. When a point object is glued to a grid line and the grid line is moved, the point object moves with the grid line. Line and area objects that are attached to the point object when it is moved remain attached to the point object and thus move or resize as appropriate.

Edit Story Data Command

Use the **Edit menu > Edit Story Data** command or toolbar button to edit story information and to insert new story levels or delete existing story levels. Clicking the **Edit menu > Edit Story Data** command brings up three submenus: Edit, Insert Story and Delete Story.

Edit Story Data > Edit Command

Clicking the **Edit menu > Edit Story Data > Edit** command brings up the Story Data form. The items in this form are described as follows.

- **Label:** Identifies the story level. Default values for this label are STORY1, STORY2, and so forth. You can change the label for any story level; for example, you may want to label your story levels 1st, 2nd, and so on. Note that the bottom of the building is identified as story level BASE; you cannot change this.

- **Height:** Defines the interstory height of the story level. It is the distance from the considered story level to the story level below. Note that by default, the story height of the BASE level is zero; you cannot change this.

- **Elevation:** Identifies the elevation of the story level relative to the base elevation. Note that you can specify the BASE level elevation (the default is 0). The program automatically calculates all other elevations and you cannot change them. These story level elevations are provided for informational purposes.

- **Similar To:** This is tag indicates that a story level is similar to another story level for drawing, assignment and selection purposes when working in plan view. See the section entitled "Similar Story Levels" in Chapter 8 for more information.

- **Splice Point:** A Yes in this column indicates that the columns will be spliced at this floor level. A No in this column indicates that the columns at this floor level are not to be spliced.

- **Splice Height:** If this floor level has been specified as a level where the columns will be spliced, the dimension here indicates the height above the top of steel at that level where the column splice will occur.

Cautionary Note: Once you change something in the story data and close the form by clicking the **OK** button, you cannot undo the change.

Edit Story Data > Insert Story Command

Clicking the **Edit menu > Edit Story Data > Insert Story** command brings up the Insert New Story form. Specify the following items in this form:

- **Story ID:** Specify the name of the new story level.
- **Story Height:** Specify the inter-story height (not elevation above the base level) of the new story level.
- **Number of Stories:** Specify the number of stories to be inserted.
- **Insert Above Level:** Specify the story level above which the new story level is to be inserted. It can be any story level that currently exists in the model.

Note:

When a new story level is inserted, all story levels above it are automatically moved up.

- **Replicate New Story:** Specify an existing story level from which the new story level is to be replicated. If you specify an existing story level instead of None, all of the framing and all of the assignments in the specified existing story level will be copied to the new story level.

  In the Replicate New Story area of the Insert New Story option, note that specifying an existing story level (say Story Level X) instead of None is equivalent to the following process:

  1. Create a new story level (call it Story Level Y) using the None option in the Replicate New Story area of the Insert New Story form.
  2. Select all of the objects on Story Level X and click the **Edit menu > Replicate** command to open the Replicate form.

3. Select the Story tab and highlighting Story Level Y in the Replicate on Stories area of the form.

4. Click the **OK** button.

Edit Story Data > Delete Story Command

Clicking the **Edit menu > Edit Story Data > Delete Story** command brings up the Select Story to Delete form where you can specify the story to be deleted. When a story level is deleted, all story levels above it are automatically moved down.

Edit Reference Planes and Edit Reference Lines Command

You can use the Edit menu to create, modify and delete reference planes and reference lines. Use the **Edit menu > Edit Reference Planes** and **Edit menu > Edit Reference Lines** commands to complete these actions.

Note:

When drawing objects, you can snap to reference planes and lines.

Reference planes are horizontal planes at user-specified Z-ordinates. The main purpose of these planes is to provide a horizontal plane/line that you can snap to when drawing objects in elevation views. You can also view reference planes in a plan view. This option can be useful for adding mezzanine-type framing when you have not specified the mezzanine as a story level in the story data.

Note the following about reference planes:

Note:

You can view and draw on reference planes in plan view.

- If you draw vertical line objects or vertical area objects (columns or walls) in plan on a reference plane level, the program inserts one object from the reference plane to the story level below and another object from the reference plane to the story level above.

- When you draw vertical line or area objects (columns or walls) in plan on a story level and there is a reference plane in

that story level, the program does *not* break the vertical object at the reference plane. A single object is drawn from the story level to the story level below.

Reference lines are vertical lines at user-specified global X and Y coordinates. The main purpose of those lines is to be available for snapping when drawing objects in elevation or plan view. Reference lines appear as points in plan view.

Merge Points Command

Use the **Edit menu > Merge Points** command or toolbar button to merge any points in the model at any time. Note that you can specify a tolerance distance for automatically merging points using the Auto Merge Tolerance edit box on the Dimensions/Tolerances form, which is accessed using the **Options menu > Preferences > Dimensions/Tolerances** command. Note that when points are created (drawn, moved, copied, or replicated) such that the distance between them is smaller than the specified Auto Merge Tolerance, the program will automatically merge the new point into the existing point.

To use the **Edit menu > Merge Points** command, first select the points to be merged. Then execute the command and specify a merge tolerance. The program uses the following logic to merge the points:

1. The program organizes the selected points based first on the number of grid lines that pass through them and then on the order in which they were drawn.

2. The program merges all selected points that are within the specified merge tolerance of the first point in the sorted list (if any) with the first point in the selected list.

3. The sorted list is updated by deleting any point that has been merged to the first point on the sorted list and by deleting the first point on the sorted list. This creates a new first point on the sorted list.

4. Steps 2 and 3 are repeated until all points have been deleted from the sorted list.

A couple of special cases exist for merging points, as follows:

- Suppose one point is exactly at a story level (call it Story Level X) and a second point is a very short distance above the first point. Thus, the second point is associated with Story Level (X+1). In that case, the point above *always* merges into the point located at Story Level X assuming, of course, that the distance between the points is within the specified merge tolerance.
- If one point is located just below a story level and another point is located just above the same story level, and thus is actually associated with the story level above, those two points will *never* merge. This is true regardless of the specified merge tolerance.

Align Points/Lines/Edges Command

Tip:

*Use the **Edit menu > Align Points/Lines/Edges** command to align objects in your model and to trim or extend line objects.*

The **Edit menu > Align Points/Lines/Edges** command or toolbar button provides some powerful tools for aligning objects in your model. To use this command, select the objects to be aligned and then specify a coordinate system, align option and align tolerance. These items are described in the subsections that follow.

Important: Note the following about aligning points, lines and edges:

- If a point object is moved using the **Edit menu > Align Points/Lines/Edges** command, all objects connected to the point object are reoriented. For example, if the point object at the top of a column-type line object is aligned (moved) in plan, the top of the column-type line object moves with the point object. The bottom of the column-type line object does not move. Note that in that case, because the column-type line

object is no longer vertical, the program automatically changes it to a brace-type line object.

- Suppose a line object is selected but the points at the end of the line object are not selected. Next suppose that the **Edit menu > Align Points/Lines/Edges** command is used to align (move) this line object. In that case, the line object moves but the point objects at the end of the line object do not move. New point objects are created at the ends of the line object in its new position if necessary. Any other objects that were connected to the point objects at the ends of the line object in its original location remain where they were; they do not move in any way. Similarly any assignments to the point objects at the ends of the line object in its original location remain where they were. If no other objects are connected to the point objects at the ends of the line object in its original location, and if there are no assignments made to these point objects, the program deletes them after the line object has been moved.

- When the program aligns an edge of an area object, only the edge of the area object being aligned actually moves. All other edges of the area object remain in their original location. Thus, when an area object is aligned, its shape changes.

- Suppose an area object is selected but the points at the corners of the area object are not selected. Next suppose that the **Edit menu > Align Points/Lines/Edges** command is used to align (move) an edge of this area object. In that case, the edge of the area object moves but the point objects at the ends of the edge of the area object do not move. New point objects are created at the ends of the edge of the area object in its new position if necessary. Any other objects that were connected to the point objects at the ends of the edge of the area object in its original location remain where they were; they do not move in any way. Similarly any assignments to the point objects at the ends of the edge of the area object in its original location remain where they were. If no other objects are connected to the point objects at the ends of the edge of the area

object in its original location and if there are no assignments made to these point objects, the program deletes them after the edge of the area object has been moved.

Align Points/Lines/Edges Form

Clicking the **Edit menu > Align Points/Lines/Edges** command brings up the Align Points/Lines/Edges form. The Coordinate System specified on this form indicates which coordinate/grid system is to be considered for the following align options:

- Align to X-coordinate
- Align to Y-coordinate
- Align to Z-coordinate
- Align to X grid lines
- Align to Y grid lines

Align Options to Selected Objects

Eight Align Options to Selected Objects are available on the Align Selected Lines/Edges/Points form:

- Align Points to X-Coordinate
- Align Points to Y-Coordinate
- Align Points to Z-Coordinate
- Align Points to X Grid Line.
- Align Points to Y Grid Lines
- Trim Lines at
- Extend Lines to
- Align Points to

These options are described in the subsections that follows.

Align to X, Y or Z Coordinate

For these options, specify the location of the coordinate that you want to align to. Then if the appropriate coordinate of the selected object is within the maximum move allowed of the specified coordinate, the appropriate coordinate of the selected object is changed to the specified coordinate.

For example, assume that you choose to align to an X-coordinate of 4 and assume that your maximum move allowed is 0.2. In that case:

- Any selected point object that has an X-coordinate between 3.8 and 4.2 (in the specified coordinate system) is moved such that it has an X-coordinate of 4. Selected point objects with X-coordinates outside of the 3.8 to 4.2 range are not moved.

- Any selected line object where each of the ends of the line has an X-coordinate between 3.8 and 4.2 (in the specified coordinate system) is moved such that each of its ends has an X-coordinate of 4. Selected line objects with the X-coordinates of one or both ends outside of the 3.8 to 4.2 range are not moved.

- Any selected area object where two adjacent corner points each have an X-coordinate between 3.8 and 4.2 (in the specified coordinate system) is resized such that two adjacent corner points have an X-coordinate of 4. Selected area objects where there are not two adjacent corner points each having an X-coordinate between 3.8 and 4.2 are not resized. Note that the effect of changing the coordinates of two adjacent corner points of the area object is to move one of its edges.

Align to X or Y Grid Lines

For these options, you specify which grid line you want to align to. Then if the appropriate coordinate of the selected object is within the maximum move allowed of the specified grid line, the appropriate coordinate of the selected object is changed to be the same as the specified grid line.

When you are aligning to X grid lines, the X-coordinate of the selected object is examined, and if it is in the appropriate range, it is modified. The Y and Z coordinates are not affected.

When you are aligning to Y grid lines, the Y-coordinate of the selected object is examined, and if it is in the appropriate range, it is modified. The X and Z coordinates are not affected.

For example, assume that you choose to align to an X grid line at a coordinate of 4 and assume that your maximum move allowed is 0.2. In that case:

- Any selected point object that has an X-coordinate between 3.8 and 4.2 (in the specified coordinate system) is moved such that it has an X-coordinate of 4 to match the grid line. Selected point objects with X-coordinates outside of the 3.8 to 4.2 range are not moved.

- Any selected line object where each of the ends of the line has an X-coordinate between 3.8 and 4.2 (in the specified coordinate system) is moved such that each of its ends has an X-coordinate of 4 to match the grid line. Selected line objects with the X-coordinates of one or both ends outside of the 3.8 to 4.2 range are not moved.

- Any selected area object where two adjacent corner points each have an X-coordinate between 3.8 and 4.2 (in the specified coordinate system) is resized such that two adjacent corner points have an X-coordinate of 4 to match the grid line. Selected area objects where there are not two adjacent corner points each having an X-coordinate between 3.8 and 4.2 are not resized. Note that the effect of

changing the coordinates of two adjacent corner points of the area object is to move one of its edges.

Trim or Extend Selected Lines

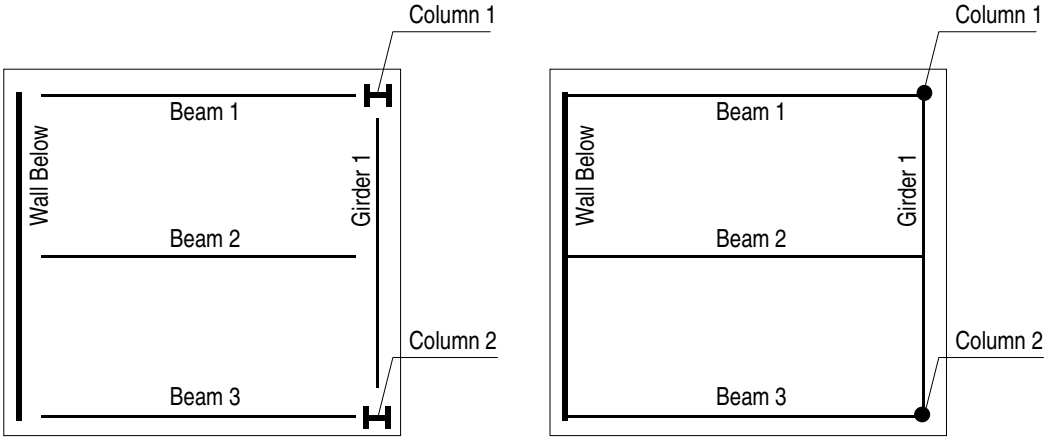
These options allow you to trim or extend line objects. Recall that in the program, to ensure that a beam is connected to a girder, the end of the beam should be exactly on the girder, not some distance away, as it might be drawn in the building plans. If you import a floor plan from a *.DXF file, it is likely that the beams will be drawn such that they stop short of the girders. Similarly, the girders will be drawn such that they stop short of the columns. These commands allow you to fix such things in your ETABS model.

For these options, specify at what point you want the selected lines to be trimmed (or extended). Then, if the end of the selected line object is within the maximum move allowed of the specified trim (or extend) item, the line option is trimmed (or extended) to that item.

Four different choices are available for the trim (extend) item. They are:

- Any line object that has frame section properties assigned to it.
- Any line object or any edge of an area object.
- Any line object.
- Any edge of an area object.

Note that the specified trim (extend) item need not be in the same plane as the selected line objects. For example if you choose line objects as your extend item and you select a girder as the object to be extended, the girder can be extended to the line object representing the column even though the girder is in a horizontal plane and the column is in a vertical plane.



a) Floor Plan as Drawn in Building Plans **b) Floor Plan in ETABS Model**

(Above)
Figure 5-5:
Example of extending selected lines

Consider the example shown in Figure 5-5. Figure 5-5a shows a floor plan as it might be drawn on your building plans. Note that the beams stop short of the girder and the beams and girders stop short of the columns. Also, in this example the beams stop short of the wall.

If you select Beams 1, 2 and 3 and Girder 1 and execute the **Edit menu > Align Points/Lines/Edges** command, you could then use the "Extend selected lines to" align option to specify that the lines be extended to "Line or Edge." This would give you the model shown in Figure 5-5b as long as the maximum required line extension is less than your specified maximum move allowed. This would give your model the correct connectivity between the various elements.

The left side of Beams 1, 2 and 3 are extended to meet the top edge of the vertical area object that represents the wall below. The right side of Beam 1 is extended to meet the vertical line object that represents Column 1. The right side of Beam 2 is extended to meet the horizontal line object that represents Girder 1. The right side of Beam 3 is extended to meet the vertical line object that represents Column 2. The top side of Girder 1 is extended to meet the vertical line object that represents Column 1.

The bottom side of Girder 1 is extended to meet the vertical line object that represents Column 2.

If you specified that the lines are to be extended to "Edge," only the left side of Beams 1, 2 and 3 would be extended to meet the top edge of the vertical area object that represents the wall below. No other line extensions would be accomplished in that case.

Assume that the beams, girder and column are all assigned frame section properties. If you specified that the lines are to be extended to "Frame Sections," all of the extensions shown in Figure 5-5b would be accomplished, except that the left side of Beams 1, 2 and 3 would not be extended to meet the top edge of the vertical area object that represents the wall below.

Note the following about trimming and extending selected lines:

- If two or more specified trim (extend) items are within the maximum allowed move distance to the end of the line, the trim (extend) is done to the item that is closer to the end of the line.

  For example, assume that in Figure 5-5 the right end of Beam 2 is within the specified maximum allowed move distance to both Girder 1 and the edge of the slab. If you select Beam 2 and specify that the lines are to be extended to "Line or Edge," the right end of Beam 2 would be extended to Girder 1, not the edge of the slab, because the right end of the beam is closer to Girder 1 than it is to the edge of the slab.

- Line objects are always trimmed (extended) along their longitudinal axis.

- Specified trim (extend) items (frame sections, line objects or edges of area objects) are only considered if they are visible in the active window. You can use this feature together with the **View menu > Show Selection Only** command to get additional control of the trimming (extending) of line objects.

Align Points To

You can align selected points to the following items:

- Any line object that has frame section properties assigned to it.
- Any line object or any edge of an area object.
- Any line object.
- Any edge of an area object.

When you use this option, any selected point is aligned with the closest specified item if that item is within the specified maximum move allowed. Specified items (frame sections, line objects or edges of area objects) are only considered if they are visible in the active window. You can use this feature together with the **View menu > Show Selection Only** command to get additional control for aligning points.

When checking for the closest specified item to a selected point, the program measures the perpendicular distance to line and edges of area objects. When a point is moved to align with an item, it is either moved perpendicular to the line object or edge of the area object.

Align Tolerance

The Align Tolerance area of the Align Selected Lines/Edges/Points form displays the Maximum Move Allowed. This is a distance that is you can specify in the current units. If the selected object is within the align tolerance distance of whatever it is specified to be aligned with, the selected object is moved. If it is not within the align tolerance distance, it is not moved.

When aligning line objects to an X, Y or Z coordinate or to an X or Y grid line, the line object is only moved if both of the end points of the line object are within the specified maximum move allowed. If you have instead, or in addition, selected the point objects at the ends of the line object, it is possible to have one

end of the line object move (align) and not the other. This happens if the alignment is based on the point object because the line object connected to the point object is reoriented when the point object moves.

Similarly, when aligning the edge of an area object to an X, Y or Z coordinate or to an X or Y grid line, the edge of the area object is only moved if both of its end points are within the specified maximum move allowed. If you have instead, or in addition, selected the point objects at the ends of the edge of the area object, it is possible to have just one of the point objects move (align).

Move Point/Line/Areas Command

Selected objects can be moved in any direction by clicking the **Edit menu > Move Points/Lines/Areas** command or toolbar button, which brings up the Move Points/Lines/Area form. In this form, specify the distances that the object is to be moved in the global X, Y and Z directions. One restriction on the movement is that when you move objects in the Z direction, they cannot cross a story level. See the subsequent subsection entitled "Moving Objects in the Z Direction" for more information.

Tip:

Objects moved in the Z direction cannot cross a story level.

When you move a point object, all line and area objects attached to the point are reoriented or resized to account for the movement. For example, if you move a point object at the top of a column, the column will become sloped. (Note that the program would then consider this column to be a brace).

When you move a line object, the line object moves but the point objects at the end of the line object do not move. New point objects are created at the ends of the line object in its new position if necessary. Any other objects that were connected to the point objects at the ends of the line object in its original location remain where they were; they do not move in any way. Similarly any assignments to the point objects at the ends of the line object in its original location remain where they were. If no other objects are connected to the point objects at the ends of the line

object in its original location and if there are no assignments made to these point objects, the program deletes them after the line object has been moved.

Similarly, when you move an area object, the area object moves but the point objects at the corners of the area object do not move. New point objects are created at the corners of the area object in its new position if necessary. Any other objects that were connected to the point objects at the corners of the area object in its original location remain where they were; they do not move in any way. Similarly any assignments to the point objects at the corners of the area object in its original location remain where they were. If no other objects are connected to the point objects at the corners of the area object in its original location and if there are no assignments made to those point objects, the program deletes them after the area object has been moved.

Moving Objects in the Z Direction

You can only move objects in the Z-direction within their own story level or to the story level below. You can not specify a delta Z dimension that requires an object to move across a story level.

For example, suppose you have a four-story building with 10-foot high story heights at all levels. Thus the first story level is at an elevation of 10 feet, the second story level is at 20 feet, the third story level is at 30 feet and the fourth story level is at 40 feet. Further suppose that you are moving an area object corner point that occurs at the mid-height of the third story level, that is, at an elevation of 25 feet.

You can specify a delta Z dimension for this corner point between -5 feet and +5 feet inclusive, that is between the distances of the second and third story levels. If you specify a delta Z dimension less than -5 feet, ETABS moves the point to the second story level elevation. If you specify a Z coordinate greater than +5 feet, ETABS moves the point to the third story level eleva-

tion. If you specify a Z coordinate between -5 feet and +5 feet, inclusive, then ETABS moves the point to the specified location.

Important Note: In some cases, moving a point object in the Z direction would cause line and/or area objects that are attached to the point object to cross story levels. In such cases ETABS does not allow the move to take place.

Expand/Shrink Areas Command

Select an area object and use the **Edit menu > Expand/Shrink Areas** command or toolbar button to expand or shrink an area object. When you specify an offset value in the Expand/Shrink Areas form, each edge of the area object is moved that amount in a direction perpendicular to the edge.

Note:

Positive offset values expand an area object and negative offset values shrink it.

Positive offset values cause the edges to move away from the interior of the area object; that is, they expand the object. Negative offset values cause the edges to move toward the exterior of the area object; that is, they shrink the object.

If you specify a negative offset value that causes the area object to collapse on itself, the command is ignored and the object is not shrunk. For example, assume that a rectangular area object has dimensions of 40 inches by 60 inches. If you specify an offset value of -20 inches or less (e.g., -25 inches), the area object would collapse on itself because when each of the sides that are 40 inches apart move toward each other by 20 inches, the two sides are in the same location and the area object is invalid. If the two sides that are 40 inches apart are each moved toward the other more than 20 inches, the two sides would have to cross (overlap) each other. This is not allowed.

Useful Feature: Typically, when you select an area object and use the **Edit menu > Expand/Shrink Areas** command, all sides of the area object are moved. It is possible to select one or more sides of an area object individually and have the **Edit menu >**

Expand/Shrink Areas command apply only to the selected sides of the area object. Do this as follows:

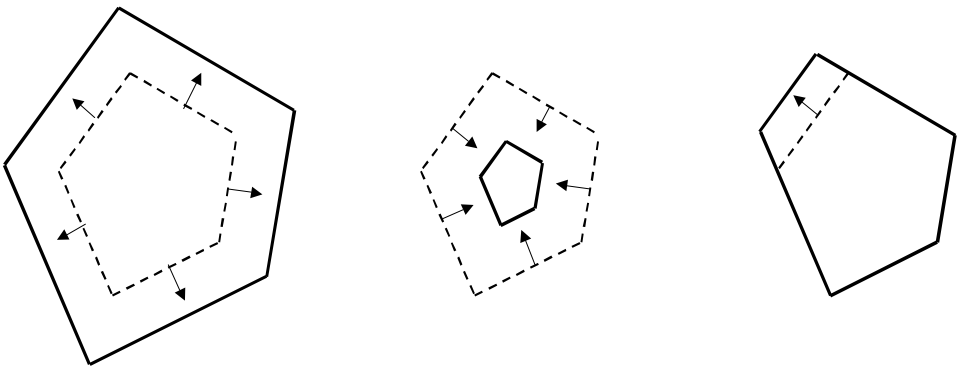
- Press and release the E key on your keyboard to enter the edge select mode. In this mode, when you click on the edge of an area object, that edge is selected. You are not able to select entire area objects by clicking inside them in this mode. Tap the space bar to discontinue edge select mode.
- Click on the area object edge(s) that you want to select.
- Use the **Edit menu > Expand/Shrink Areas** command in the normal fashion.
- When you have finished selecting area objects edges, press and release the space bar on your keyboard to return to the normal area object select mode where you select entire area objects by clicking inside of them. Note that you do not automatically return to the normal area object select mode. You must specifically press and release the space bar or the Esc key on your keyboard to return to normal select mode.

Tip:

You can expand or shrink selected edges of area objects.

Figure 5-6 shows some examples of expanded and shrunken area objects. In the figure, the dashed line represents the original area object and the solid line represents the final area object after it has expanded or shrunk. Note in Figure 5-6c that because only one edge of the area object was moved, the solid line lies on top of the dashed line for all other edges.

Figure 5-6:
Example of expanded and shrunken area objects



a) Expanded Area Object **b) Shrunken Area Object** **c) Area Object with One Side Expanded**

Merge Areas Command

Selecting two area objects that have a common edge or overlap and using the **Edit menu > Merge Areas** command or tool-bar button will merge the two area objects into one area object. You cannot merge more than two area objects at a time in the same command.

When you merge two area objects, the new area object takes on the properties and assignments of the area object with the larger area. If the two area objects have exactly the same area, the property and assignments come from the first drawn area object. Because you may not remember which area object was drawn first, carefully check the assignments of the newly combined area object. Figure 5-7 shows some examples of merged area objects.

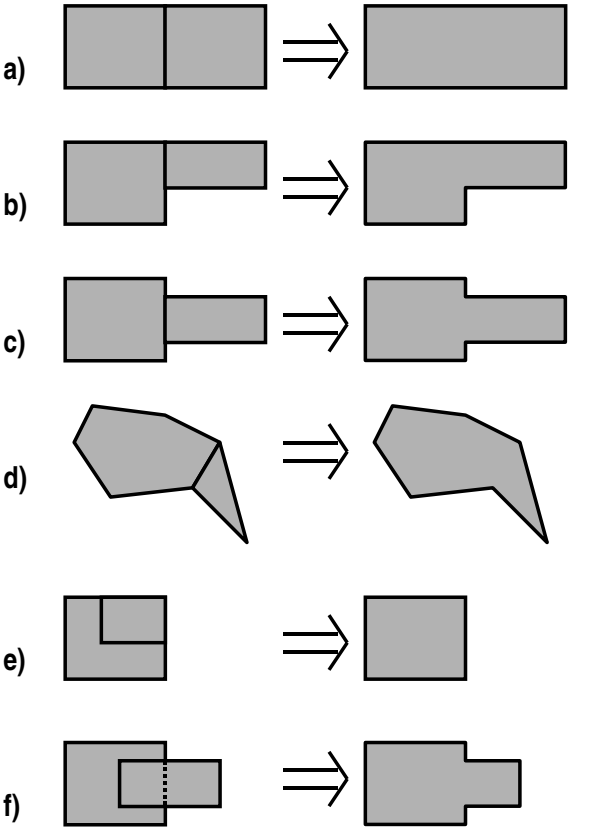


Figure 5-7: *Examples of merged area objects*

Mesh Areas Command

Selected areas can be meshed using the **Edit>Mesh Areas** command or toolbar button. Several options are available in the Mesh Selected Areas form:

- **Auto Mesh Area (Horiz):** This option meshes the selected area into smaller areas. The smaller areas are three-sided or four-sided and must have beams on all sides.

- **Cookie Cut at Selected Line Object (Horiz):** This option meshes the selected area at the selected lines. Select one or multiple lines. If the selected line passes through more than one area, all of the areas will be meshed. Note that this and the Auto Mesh Area option only work in plan view.

- **Cookie Cut at Selected Point at [Specified] Angle:** Use this option to mesh areas at a specified point and angle. The angle will be measured in the counterclockwise direction for the x- and y-axis. If the point lies in the overlapping region of two areas, both of the areas will be meshed at the given angle.

- **Mesh Quads/Triangles into [Specified Number] by [Specified Number] Areas:** This option meshes the selected area in the number of areas specified by the user. For example, specifying a meshing of 2 by 8 means that the selected area will be meshed into 2 areas along the x-axis and 8 areas along the y-axis. The size of the meshed areas will be uniform along a given direction. Only quads and triangles can be meshed using this option.

- **Mesh Quads/Triangles at:**

  - ✓ **Intersections with Visible Grid Lines:** This option meshes each selected area at any location where it intersects a visible grid line, regardless of the coordinate system associated with the grid line.

  - ✓ **Selected Point Objects on Edges:** Selecting this option will mesh the area (horizontally and vertically) using the

selected point at the edge as reference. One more points can be selected for this type of meshing.

- ✓ **Interactions with Selected Line Objects:** The areas selected are meshed with the line intersecting the area. More than one line can be selected to mesh a desired area.

Note the following about Meshing Area Objects.

- The property assignments to meshed area objects are the same as the original area object.
- Load and mass assignments on the original area object are appropriately broken up onto the meshed area objects.
- When this menu item is clicked, all edges of the currently selected area will be split at their mid-points. If clicked again for the same selected area, they will be divided in half again, and so on.

Split Area Edge Command

The **Edit menu > Split Area Edge** command adds point objects at the mid-point of each edge of an area object. To use this command, select the area object and then click the command.

Join Lines Command

Select two or more collinear line objects with common end points and the same type of property (frame section, link or none) and use the **Edit menu > Join Lines** command or toolbar button to combine the line objects into a single line object. Note the following about combined line objects.

- Combined line objects must be collinear.
- Combined line objects must have a common end point.

- Combined line objects must all have the same type of property. In other words, they must all have frame section properties, or they must all have link properties or they must all have no properties.
- When line objects with frame section properties are joined, the section property assigned to the combined line object is the one associated with the largest area. If two of the combined frame sections have the same area, the property of the first drawn object is used.
- Load and mass assignments from the unjoined line objects are combined on the joined object.
- Assignments to the unjoined line objects that would be illegal in the middle of the joined line object are ignored. For example, frame member end releases, rigid end zones and joint offsets that would occur in the center of joined frame members are ignored.

Figure 5-8 shows some examples of joined line objects. Figure 5-8(a) shows that two collinear line objects with a common end point (and the same property type assignment) are joined into one line object. Figure 5-8b shows that five collinear line objects can be joined at the same time. Figures 5-8c and 5-8d show that two sets of collinear line objects can be joined simultaneously. The two sets of line objects can have different property type assignments but all of the property type assignments within either set of line objects must be the same.

Figure 5-8e illustrates that the collinear line objects must have a common end point, otherwise they are not joined. To join the example shown in Figure 5-8e, move one of the center joints so that it is coincident with the other center joint and then perform the join.

Figure 5-8f illustrates that assignments to the unjoined line objects that would be illegal in middle of the joined line object are ignored. In that case, a moment release that is in the center of the

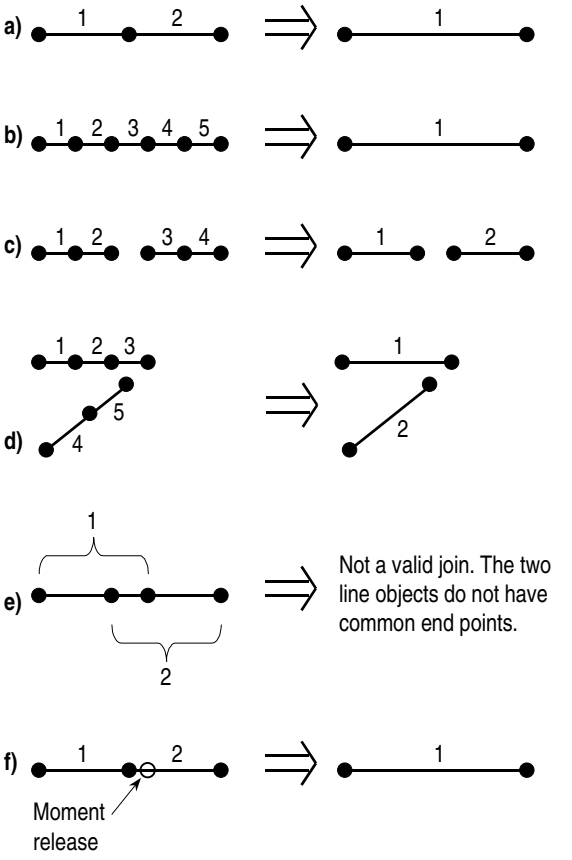


Figure 5-8: *Examples of joined line objects*

combined beam is ignored. If you want this moment release to remain, you should not join the line objects.

Divide Lines Command

Select one or more line objects and use the **Edit menu > Divide Lines** command or toolbar button to divide the line object into multiple line objects. Several options are available for dividing the line objects:

- **Divide into [Specified Number of] Objects:** This option divides the selected line object(s) into the specified number of line objects. The divided line objects are all the same length.

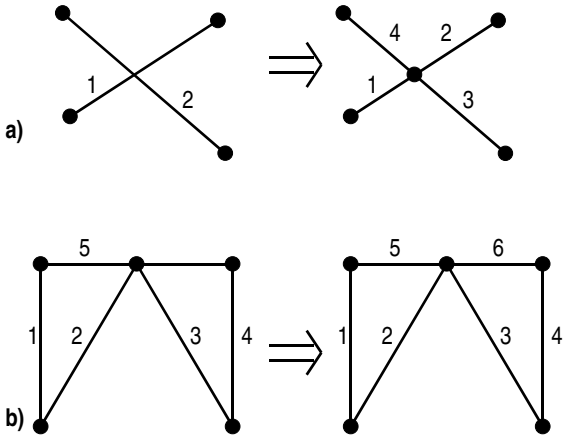


Figure 5-9: *Examples of breaking line objects at intersections with selected lines and points*

- **Break at Intersections with Selected Lines and Points:** This option breaks each selected line at any point where it intersects another *selected* line or point. Figure 5-9 shows some examples.

  Figure 5-9a shows two crossing line objects. Initially the line objects are not connected at their intersection. When the two line objects are selected and the Break at Intersections with Selected Lines and Points option is used to divide the lines, each of the lines is broken into two objects at the intersection point.

  Figure 5-9b shows a common situation in a chevron-braced frame. Notice that the line object representing the top beam (labeled 5) spans from one column to the other and is not broken at the intersection with the braces. To break this beam at the intersection with the braces, select the beam and one of the braces (say you select the line objects labeled 2 and 5) and use the Break at Intersections with Selected Lines and Points option. Alternatively, select the point at the top of the braces and the top beam (line object labeled 5) to achieve the same result.

  In either of the case shown in Figure 5-9, the program would provide connectivity at the intersection points, regardless of whether the line objects are divided or not, unless the inter-

secting line objects are indicated not to be meshed using the **Assign menu > Frame/Line > Automatic Frame Mesh/No Mesh > Don't Mesh It** command (refer to Chapter 10 Assign Menu). In most cases, manual breaking up of intersecting line objects as shown in Figure 5-9 is not necessary unless different properties are to be assigned or a different label is required. The example in Figure 5-9 is merely intended to illustrate the Break at Intersections with Selected Lines and Points option for dividing lines.

- **Break at Intersections with Visible Grid Lines:** This option breaks each selected line at any location where it intersects a visible grid line, regardless of the coordinate system associated with the grid line.

Note the following about divided line objects.

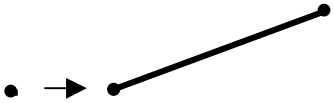
- The property assignments to divided line objects are the same as the original line object.
- Load and mass assignments on the original line object are appropriately broken up onto the divided line objects.
- Assignments that occur at the ends of the original line object, such as releases and rigid end zones, occur at the appropriate ends of the two end line objects when the original object is divided.

Extrude Points to Lines Command and Extrude Lines to Areas Command

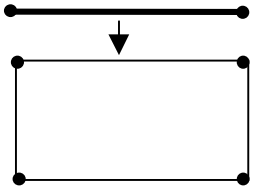
In many situations during model creation, it is very convenient to generate objects of one kind by some transformation of another type of object. These commands are useful in creating line objects from points and area objects from line objects. Figure 5-10 illustrates examples of object extrusion. Figure 5-11 illustrates the Extrude Lines to Areas form that comes up when you click the **Extrude Lines to Area** command or the toolbar button.

Figure 5-10: *Examples of object extrusion (linear)*

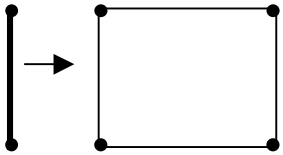
(a) Generation of a single horizontal line object from a point object



(b) Generation of a single inclined line object from a point object

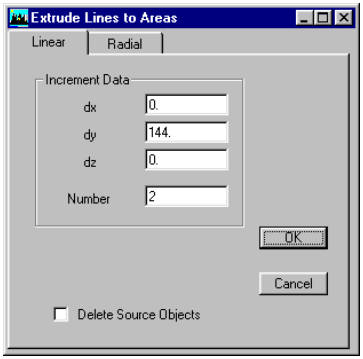


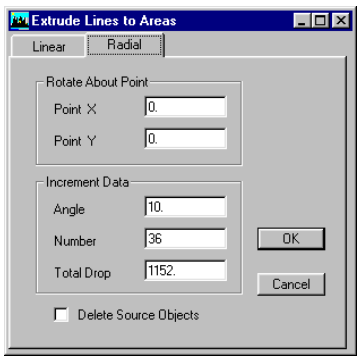
(c) Generation of an area object from vertical extrusion of an line object



(d) Generation of an area object from horizontal extrusion of an line object

Figure 5-11: *Input for linear and radial extrusion*





The process of extrusion increases the dimensional space of an existing object by one. In other words, line objects are of one dimension that can be generated from a dimensionless object, the point object. In a similar manner, a two-dimensional object, area or plate/shell can be generated from a one-dimensional object, the line object. This feature is especially suited to creating

plate/shell elements from beams and beams/columns from point/nodes.

The object can be extruded linearly or radially. The same two options are available on the Extrude Points to Lines form and the Extrude Lines to Areas form. They Linear and Radial and are described as follows:

Linear Extrusion

For linear extrusion you must specify the increment distance along the X-axis (dx), the increment distance along the Y-axis (dy), and the number of times the object is to be extruded. The object is then extruded the specified number of times, each time incrementing the global X and Y coordinates by the specified dx and dy.

A Delete Source check box is provided on the Extrude Lines to Areas form. Checking this check box will delete the lines used to create the area. Leaving the check box unchecked will keep the lines as well as the area.

Radial Extrusion

A typical use of this type of extrusion is to generate a cylindrical surface from radial extrusion of a single line about the central axis of the cylinder. For radial extrusion, specify a point to rotate about (the rotation is in the global X-Y plane about the global Z-axis), a rotation angle, and a number of times the object is to be extruded. The object is then extruded the specified number of times, each time incrementing the location of the objects by the specified rotation angle. The rotation angle is input in degrees. Angles are measured from the positive global X-axis. Positive angles appear counterclockwise when you view them from above.

A Delete Source check box is provided on the Extrude Lines to Areas form. Checking this check box will delete the lines used to

create the area. Leaving the check box unchecked will keep the lines as well as the area.

Auto Relabel All Command

You can relabel all objects of the current model using the **Edit > Auto Relabel All** command. This is a one-way command and cannot be undone. You cannot selectively relabel objects. Unlike other commands on the Edit menu, you do not need to select the objects before using this command.

The program relabels the objects in the following order. working the global coordinate system, the objects are first sorted by their global delta Z from their story level, then by their global Y location, and finally by their global X location. The advantage of relabeling is so that with this relabeling scheme if you find area object labeled W2, for example, you will know that you can typically find the area object labeled W3 nearby. Without relabeling, the object labeling is in the order in which the objects were defined in the model, which often times is not necessarily an orderly progression.

Typically, we recommend that you use the **Auto Relabel All** command after you have finished creating your model to get optimum labeling for the model. Keep in mind that if you add or subtract objects from your model after you have used the **Auto Relabel All** command and then use the command again, what was once object W3, for example, may not longer be area object W3.

Nudge Feature

Tip:

You can nudge dimension lines.

The program includes a nudge feature that allows you to modify the geometry of your model in a plan view. To use the nudge feature, select the item(s) that you want to nudge and press the Ctrl key and one of the arrow keys on your keyboard simultaneously. Note the following about the nudge feature:

- The nudge feature only works in plan view.

- You can nudge any selected point, line or area object. You can also select dimensions lines and nudge them.
- Pressing the Ctrl key plus the right arrow key nudges the object in the positive global X direction.
- Pressing the Ctrl key plus the left arrow key nudges the object in the negative global X direction.
- Pressing the Ctrl key plus the up arrow key nudges the object in the positive global Y direction.
- Pressing the Ctrl key plus the down arrow key nudges the object in the negative global Y direction.
- The distance that the object(s) is nudged (moved) when you press the Ctrl and arrow keys is specified in the Dimension/Tolerance Preferences, which can be accessed using the **Option menu > Preferences > Dimensions/Tolerances** command. The name of the item that controls the movement is Plan Nudge Value.
- You cannot nudge objects in the Z direction.
- Similar to the **Edit menu > Move Points/Lines/Areas** command, when you nudge an area object without having selected the corner points of the object, the area object moves but the point objects at the corners of the area object do not move. New point objects are created at the corners of the area object in its new position if necessary. Any other objects that were connected to the point objects at the corners of the area object in its original location remain where they were; they do not move in any way. In other words, the area object is disconnected from other objects when it is nudged.
- Similar to the **Edit menu > Move Points/Lines/Areas** command, when you nudge a line object without having selected the end points of the object, the line object moves but the point objects at the ends of the line object do not move. New point objects are created at the ends of the line object in its

new position if necessary. Any other objects that were connected to the point objects at the ends of the line object in its original location remain where they were; they do not move in any way. In other words, the line object is disconnected from other objects when it is nudged.

- Similar to the **Edit menu > Move Points/Lines/Areas** command, when you nudge a point object all of the objects connected to the point move too; they are either reoriented or resized, or both. Unlike area and line objects, the point object does not disconnect from the objects it is attached to when it is nudged.

Chapter 6

View Menu

General

The View menu provides basic options and tools for viewing a model. This chapter describes those options and tools in the order displayed in the menu drop-down box.

The viewing options available on the View menu should not be confused with the display options available on the Display menu. The View menu items control the type of view and the visibility of objects. The Display menu items control the display of input and output items. The Display menu is described in Chapter 12.

Set 3D View Command

Set a window to a three-dimensional (3D) view using the **View menu > Set 3D View** command or by clicking the 3-d toolbar button. Use of the **View menu > Set 3D View** command will

open the Set 3D View form where you can define the view direction by specifying a plan angle, elevation angle and an aperture angle. Use of the 3-d toolbar button gives you the default 3D view, with the plan, elevation and aperture angle as specified in Figure 6-1.

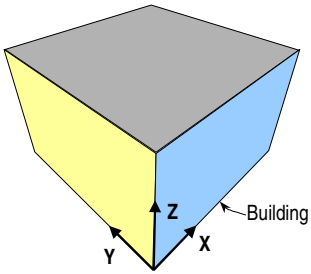
All angles are specified in degrees. The view direction defines the location where you would be standing if you were viewing the building from the outside. Figure 6-1a shows a three-dimensional view of a building using the default view direction of plan angle = 225 degrees, elevation angle = 35 degrees and aperture angle = 60 degrees. Figures 6-1b, 6-1c and 6-1d illustrate how the plan, elevation and aperture angles are defined within the program. Following are explanations of the terms used in Figure 6-1.

- **Eye point:** This is the location from which you are viewing the building.
- **Target point:** This is the geometric center of the building.
- **View direction:** This is defined by a line drawn from the eye point to the target point.
- **Plan angle:** This is the angle (in degrees) from the positive global X-axis to the line defining the view direction measured in the horizontal global XY plane. A positive angle appears counterclockwise as you look down on the model. Any value between -360 degrees and +360 degrees, inclusive, is allowed for the plan angle.
- **Elevation angle:** This is the angle (in degrees) from the global XY plane to the line defining the view direction. A positive angle starts from the global XY plane and proceeds toward the positive global Z-axis. A negative angle starts from the global XY plane and proceeds toward the negative global Z-axis. Any value between -360 degrees and +360 degrees, inclusive, is allowed for the elevation angle.

Note:

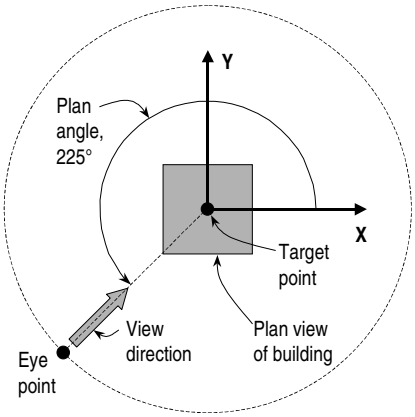
The plan and elevation angles together control the direction from the eye point to the target point. The aperture angle controls the distance from the eye point to the target point.

Figure 6-1: *Illustration of plan, elevation and aperture angles used to define a 3D view*

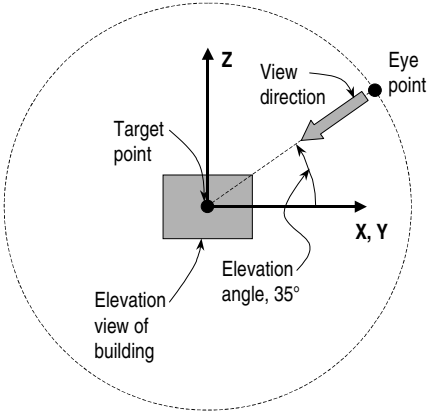


Plan angle = 225°
Elevation angle = 35°
Aperture angle = 60°

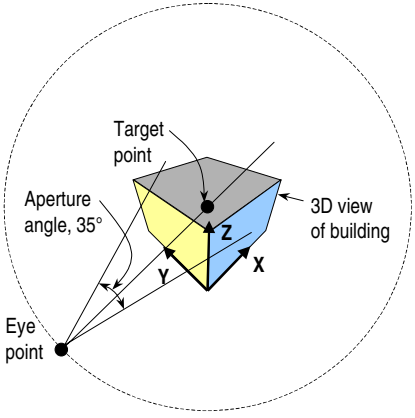
a) Default 3D View



b) Plan Angle



c) Elevation Angle



d) Aperture Angle

- **Aperture angle:** The plan angle and the elevation angle together define the direction from the eye point to the target point. The aperture angle sets the distance from the eye point to the target point. This distance is set as follows:

  - ✓ The program constructs the view direction line from the eye point to the target point.
  - ✓ The program constructs a 3D bounding rectangular box that just encloses the model.
  - ✓ The program passes a plane through the target point and perpendicular to the view direction line.
  - ✓ The program projects the eight corner points of the bounding box onto the plane. Those points are the "projected corner points."
  - ✓ The program constructs lines from the eye point to the projected corner points of the bounding box. Eight such lines are constructed because there are eight corner points. Those eight lines are the "corner point lines."
  - ✓ The program locates the eye point along the view direction line such that the largest angle between the view direction line and any of the eight corner point lines is equal to one-half of the specified aperture angle. This ensures that the entire structure is included in the view.

  Note that the program does not allow the eye point to be located inside the structure.

Set 3D View Form

The 3D View form has four Fast View buttons labeled 3-d, xy, xz and yz. The fast view buttons automatically set the plan, elevation and aperture angle to give you the specified 3D view. The fast view 3D view is as shown in Figure 6-1. The other fast views give you 3D perspective views of the specified planes (i.e., xy, xz, yz).

Adjust the 3D View

When the active window is showing a 3D view, you can adjust the 3D view using the **Rotate 3D View** toolbar button. First click the button. Then left click the mouse in the window with the 3D view; hold down the mouse left button while you drag the mouse to adjust the view direction. Note that as soon as you left click the mouse in the window, a bounding box (dashed lines enclosing the model) appears, and as you drag the mouse, the orientation of the bounding box changes, showing you the new orientation of the model. When you release the left mouse button, the entire model is redrawn in the new view direction.

Refer to the subsection in this chapter entitled "Perspective Views" for additional information on three-dimensional views.

Set Plan View Command

Note:

You can create plan views of story levels and reference planes.

Set a window to a plan view using the **View menu > Set Plan View** command or the **Plan View** toolbar button. Using the menu item or clicking the toolbar button brings up the Select Plan Level form. From this form, select the story level or reference plane that you want to show in plan view. Refer to the section entitled "Set Reference Planes and Set Reference Lines Commands" in Chapter 5 Edit Menu for additional information on reference planes. Alternatively, use the **Move Up in List** and the **Move Down in List** toolbar button to quickly change views.

The following objects are visible in plan view if they have been specified as visible in the Set Building View Options form, which is accessed through the **View menu > Set Building View Options** command (see subsequent section entitled "Set Building View Options Command"):

- All area, line and point objects that lie in the horizontal plane of the plan view. This horizontal plane occurs at the story level

elevation for plan views of story levels and at the reference plane elevation for plan views of reference planes.

- All column-type line objects that have an end point in the considered plan view or pass through the considered plan view. Note that column-type line objects cannot pass through plan views at story levels. They can only pass through plan views at reference plane levels.
- All wall-type area objects that have corner points in the considered plan view or pass through the considered plan view. Note that wall-type area objects cannot pass through plan views at story levels. They can only pass through plan views at reference plane levels.

Note that braces and ramps are never visible in a plan view. Braces are visible in elevation view and in any three-dimensional view, including all perspective views. Ramps are visible in any three-dimensional view, including all perspective views.

Refer to the subsection in this chapter entitled "Perspective Views" for additional information on plan views. See Chapter 9 Select Menu for special selection rules when windowing in plan view ("windowing" means to draw a window around one or more objects to select them).

Set Elevation View Command

By default the program defines elevation views along each of the defined primary grid lines in the model. Use the **View menu > Set Elevation View** command or click the **Elevation View** toolbar button to display the Set Elevation View form. This form allows you to add additional elevation views, modify or delete existing elevation views, and select which elevation view to display.

Developed elevation views can be displayed but not defined from the Set Elevation View form. Refer to the section entitled "Draw Developed Elevation Definition Command" in Chapter 8

for an explanation of developed elevations and information on defining developed elevation views.

Note:

Elevation views are automatically created along primary grid lines. They are not automatically created along secondary grid lines.

The Set Elevation View form includes a list of the defined elevation names (including the names of defined developed elevations, if any) and several command buttons. To select an elevation for display, highlight the appropriate elevation name in the Set Elevation View form and click the **OK** button. To delete a defined elevation view, highlight the elevation in the Set Elevation View form and click the **Delete Elevation Name** button.

To define a new elevation view, click the **Add New Elevation** button in the Set Elevation View form. (Note: The **Add New Elevation** button is not used to define developed elevations. Use the **Draw menu > Draw Developed Elevation Definition** command to define a developed elevation.)

To modify an existing elevation view, highlight the elevation in the Elevations list box on the Set Elevation View form and click the **Modify/Show Elevation** button. Note that you cannot modify the default elevations along the grid lines that are created by the program; however, you can delete them. Also you cannot modify user-defined developed elevations; however, you can delete them.

Both the **Add New Elevation** and the **Modify/Show Elevation** commands bring up the Elevation Data form. This form allows you to specify a name, coordinate system and location for the elevation. The location is either an X or Y ordinate in the specified coordinate system. If you specify an X ordinate, the elevation is a view of the YZ plane in the specified coordinate system at the specified X ordinate. Similarly, if you specify a Y ordinate, the elevation is a view of the XZ plane in the specified coordinate system at the specified Y ordinate.

With the active window in elevation view, use the **Move Up in List** button and the **Move Down in List** button to move quickly through the available elevation views.

Refer to the subsection in this chapter entitled "Perspective Views" for additional information on elevation views.

Perspective Toggle Button

The **Perspective Toggle** toolbar button is a useful tool that has slightly different behavior, depending on whether it is used in a plan, elevation or three-dimensional view.

Perspective Toggle in a Plan View

In plan view, clicking the **Perspective Toggle** button switches the view to a 3D perspective view. If the plan view is of a story level, the 3D perspective view only shows the objects that are associated with that story level. Objects associated with other story levels are not shown. If the plan view is of a reference plane, the perspective view shows only the objects of the story level associated with that reference plane.

Note:

A perspective view of a plan view shows only the objects that are a part of the story level associated with the plan view.

While in the perspective view, you can use the **Rotate 3D View** toolbar button to adjust the view direction.

As the name implies, the **Perspective Toggle** button is a toggle switch. Press it once to switch to a perspective view of a story level. Press it again to switch back to your original plan view.

Perspective Toggle in an Elevation View

In an elevation view, clicking the **Perspective Toggle** button switches the view to a 3D perspective view of the entire structure. The initial direction of the 3D perspective view is looking directly at the elevation that was displayed, with the elevation angle set to 0 degrees and the aperture angle set to 60 degrees. If the elevation displayed is a developed elevation, the initial direction of the 3D perspective view is looking directly at the first segment of the developed elevation.

While in the perspective view, you can use the **Rotate 3D View** main toolbar button to adjust the view direction.

As the name implies, the **Perspective Toggle** button is a toggle switch. Press it once in an elevation view to switch to a perspective view of a structure. Press it again to switch back to your original elevation view.

Perspective Toggle in a Three-Dimensional View

Clicking the **Perspective Toggle** button in a three-dimensional window switches the view from a perspective view to an isometric view. In other words, the aperture angle is toggled to 0 degrees.

As the name implies, the **Perspective Toggle** button is a toggle switch. Press it once in a 3D view to switch to an isometric view of structure. Press it again to switch back to your original perspective view.

Set Building View Limits Command

Setting building view limits allows only objects that are entirely inside the view limits to be displayed. Use the **View menu > Set Building View Limits** command and the Set Building View Limits form to set the limits for a view. Note that view limits affect only objects; the limits have no effect on the coordinate/grid systems, which will display in their entirety.

The Set Limits form allows you to specify X-axis, Y-axis and story level (Z-axis) limits. The story level limits are set by specifying a top story level and a bottom story level.

Two methods are available for specifying the X- and Y-axis limits. For the first method, type in the minimum and maximum X and Y coordinates. For the second method, graphically set the limits in the small plan view located in the Plan Limits area of the form.

With respect to the second method, in the Plan Limits area, a dashed box with selection handles on the four sides is superimposed over a plan view of the structure. The dashed box defines the view limits. Modify the size and location of the dashed box as follows:

- Left click inside the dashed box, and while holding down the left mouse button, drag the box to a new location.
- Left click on one of the selection handles on the sides of the dashed box, and while holding down the left mouse button, drag the mouse to resize the box.

Set Building View Options Command

The program allows you to enhance the display of a model in the active window by changing colors, special effects and various other aspects. These building view options are controlled using the **View menu > Set Building View Options** command or by clicking the **Set Building View Options** toolbar button. Clicking either the command or the button brings up the Set Building View Options form. ***Note that the settings made in this form affect only the active window.***

Note:

The settings made in the Set Building View Options form affect only the currently active window.

The option categories on this form include the following:

- View by Colors of
- Special Effects
- Object Present in View
- Object View Options
- Piers and Spandrels
- Visible in View
- Special Frame Items
- Other Special Items

Each of the building display option categories is briefly described in the following subsections.

View by Colors of

You can view your model by the colors of the following items:

- **Objects:** This option displays the model by the colors of the objects as defined in the Assign Display Colors form which you can access using the **Options menu > Colors** command.
- **Sections:** This option displays the model by the colors of the frame and wall/slab/deck section properties. Any object that is not assigned frame section properties or wall/slab/deck section properties is displayed in a color that is the opposite of the background color. Note that you assign display colors to frame section properties and to wall/slab/deck section properties when you define the section properties.
- **Materials:** This option displays the model by the colors of the material properties assigned to the frame and wall/slab/deck section properties. Any object that is not assigned frame section properties or wall/slab/deck section properties is displayed in a color that is the opposite of the background color. Note that you assign display colors to material properties when you define the material properties.
- **Groups:** This option displays the model by the colors of one or more selected groups. When you select this option be sure to click the associated Select button and select the groups. Any objects that are not part of any of the specified groups are displayed in a color that is the opposite of the background color. When an object is part of more than one specified group it is displayed in the color of the first defined group that it is a part of.
- **Design Type:** This option displays the model with a different color for each different member design type. For example, all members specified as LATCOLS will be one color, all members specified as LATBMS will be another color, etc.

- **Typical Members:** This option displays all members on a level specified as “Typical” in one color.

- **White Background Black Objects:** This option displays all model objects in black on a white background. This option can sometimes be useful when you are cutting and pasting screen shots into a report that is done in black and white.

Special Effects

Four special effects features are available. They are:

- **Object Shrink:** This feature shrinks line objects and area objects. It is useful when you are trying to determine connectivity in your model. Also if you actually want to see dots at point object locations then you must shrink the area and line objects using this feature.

  Line objects are shrunk by a percentage that is controlled by the Shrink Factor in the Preferences form that you access using the **Options menu > Preferences > Dimensions/Tolerances** command. See the subsection titled "Dimensions and Tolerances" under the section titled "Preferences" in Chapter 14 for additional information.

  The object shrink feature can also be toggled on and off using the **Object Shrink Toggle** button, , that is available on the main (top) toolbar.

- **Object Fill:** This feature fills the area objects, that is draws them solid. The color used is controlled in the Assign display Colors form that is accessed using the **Options menu > Colors > Display** command. See the Object Edge item below for additional information.

- **Object Edge:** This feature displays the edges (outline) of the area objects. The color used is controlled in the Assign display Colors form that is accessed using the **Options menu > Colors > Display** command.

Note if neither the object fill or the object edge feature is active you will not be able to see the area objects, however, if you click in the location where they are supposed to be you will select them. If you want the area objects to be invisible and not selectable then you should uncheck the appropriate boxes in the Object Visibility area of the Set Building View Options form.

Tip:

Showing extrusions is a convenient way of checking the local axes orientation of frame members.

- **Extrusion:** This feature shows the extruded shape of all line objects with frame section properties assigned to them. Line objects that do not have frame section properties assigned are shown non-extruded. Only line objects are displayed when the extrusions feature is activated. Area objects and point objects are not displayed when extrusions are shown.

  When line objects are assigned auto select list frame section properties ETABS displays the extruded shape of the current analysis section. Note that before you have run the first analysis the current analysis section is the median (by weight) beam in the auto select list.

  Showing extruded shapes is a very powerful tool for verifying the local axes orientation for frame members.

Object Present in View

Note:

If an object does not have its corresponding check box checked in the Object Visibility area of the Set Building View Options form, then you can not see or select the object.

The Object Present in View area of the Set Building View Options form provides controls for the visibility of area, line and point objects. When a check box in this area is checked, the objects of that type are visible; when the box is not checked, objects are not visible. Note that when objects are not visible because the appropriate check box in the Object Visibility area is not checked, you cannot select the object. Contrast this with the information in the second paragraph describing the Object Edge feature in the subsection above entitled "Special Effects."

Following are the items for which you can control the object visibility:

- **Floor (Area):** All floor-type area objects, that is, all horizontal area objects with wall/slab/deck section property assignments.
- **Wall (Area):** All wall-type area objects, that is, all vertical area objects with wall/slab/deck section property assignments.
- **Ramp (Area):** All ramp-type area objects, that is, all sloped area objects (not vertical or horizontal area objects) with wall/slab/deck section property assignments.
- **Openings (Area):** All area objects that are designated as openings. Note that these are a subset of all null areas.
- **All Null Areas:** All null area objects, that is, all area objects that do not have wall/slab/deck section property assignments.
- **Column (Line):** All column-type line objects. By default column-type line objects are those with frame section property assignments that are oriented vertically (length is oriented parallel to the Z-axis).
- **Beam (Line):** All beam-type line objects. By default beam-type line objects are those with frame section property assignments that are oriented horizontally (fall in the XY plane).
- **Brace (Line):** All brace-type line objects. By default brace-type line objects are those with frame section property assignments that are sloped (not oriented vertically or horizontally).
- **Link (Line):** All line objects with link property assignments. Note that it is possible for a line object to have both a frame section property assignment and a link property assignment simultaneously. In this case ETABS creates a frame member and a link element in the same location in the analysis model. The following two paragraphs describe how the ETABS object visibility options affect this type of line object.

  If a line object has a link property assignment and no frame section assignment then it is classified as a null-type line object. This line object is visible if either the Links (Line) box is

checked or the All Null Lines box is checked, or both boxes are checked.

If a line object simultaneously has a frame section property assignment and a link property assignment then the line object type is either column, beam or brace depending on its orientation. Thus, for example, a *vertical* line object with both a frame section property assignment and a link property assignment (column-type) is visible if either the Column (Line) box is checked or the Links (Line) box is checked, or both boxes or checked. However it is not visible if the Column (Line) box and the Links (Line) box are unchecked even if the All Null Lines box is checked.

Note that this box does not control the visibility of zero-length links that are assigned to point objects.

- **All Null Lines:** All null line objects, that is, all line objects that do not have frame section property assignments. See the discussion above for the Link (Line) check box for additional information.

- **Point Objects:** This check box controls the visibility of point objects. Note the following about this feature:

  - ✓ When the Point Objects check box is not checked, point objects are not visible in the model and point objects cannot be selected in the model.

  - ✓ When the Points Objects check box is checked and the associated Invisible check box is unchecked and the Object Shrink check box is checked (see next bullet), point objects are visible and you can select them by clicking on them or by windowing them. If the Invisible check box is checked while the Points Objects check box is also checked, the point objects are invisible in the model, but you can select them by clicking on them or windowing them.

- ✓ This option works in conjunction with the Object Shrink feature. That is, you only see a dot representing a point object when point objects are visible (see previous bullet item) and the Object Shrink feature is on (i.e., the Object Shrink check box is checked in the Special Effects area of the Set Building View Options form or you click the **Object Shrink Toggle** toolbar button).

- ✓ This option also works in conjunction with the Links (Point) check box, the Supports check box, and the Springs check box. The Point Objects check box must be checked in order for links assigned to point objects, supports and grounded springs to be visible. In other words, if the Links (Point) check box is checked, the links assigned to point objects are still not graphically visible unless the Point Objects check box is also checked. Similarly, if the Supports check box is checked, the supports are still not graphically visible unless the Point Objects check box is also checked and if the Springs check box is checked, the grounded point springs are still not graphically visible unless the Point Objects check box is also checked.

- **Link (Point):** All point objects with link property assignments. Note that this box does not control the visibility of links that are assigned to line objects. Also as noted previously, the check box works in conjunction with the Point Objects check box.

Object View Options

The Object View Options area of the Set Building View Options form allows you to toggle the display of object labels, section properties and local axes on and off. Following is a list of the specific items you control in this area of the form:

- **Area Labels:** Labels (names) for all types of area objects (floor, wall, ramp and null).

- **Line Labels:** Labels (names) for all types of line objects (column, beam, brace and null).
- **Point Labels:** Labels (names) for all point objects.
- **Area Sections:** Wall/slab/deck section property names are displayed for all area objects with wall/slab/deck section property assignments.
- **Line Sections:** Frame section property names are displayed for all line objects with frame section property assignments.
- **Link Sections:** Link section property names are displayed for all line objects with link section property assignments.
- **Area Local Axes:** Arrows indicating local axes orientation are displayed for all area objects. Note that the local 1 axis is always shown with a red arrow, the local 2 axis is always shown with a white arrow, and the local three axis is always shown with a blue arrow.
- **Line Local Axes:** Arrows indicating local axes orientation are displayed for all line objects. Note that the local 1 axis is always shown with a red arrow, the local 2 axis is always shown with a white arrow and the local three axis is always shown with a blue arrow.

Tip:

One way to remember the colors associated with the local axes is to think of the American flag which is red, white and blue. Note that local axis 1 is red, local axis 2 is white and local axis 3 is blue.

Note that no control is provided for the point local axes. This is the case because by default in ETABS, the point local axes always correspond to the global local axes. That is, for point objects, local axis 1 is the same as the global X-axis, local axis 2 is the same as the global Y-axis and local axis 3 is the same as the global Z-axis.

Piers and Spandrels

The Piers and Spandrels area of the Set Building View Options form allows you to toggle the display of pier and spandrel labels and local axes on and off. Following is a list of the specific items you control in this area of the form:

- **Pier Labels:** Labels (names) for all specified pier elements. Recall that you specify one or more area objects as a pier by selecting them and then clicking the **Assign menu > Shell/Area > Pier Label** command.

- **Spandrel Labels:** Labels (names) for all specified spandrel elements. Recall that you specify one or more area objects as a spandrel by selecting them and then clicking the **Assign menu > Shell/Area > Spandrel Label** command.

- **Pier Axes:** Arrows indicating local axes orientation are displayed for all specified piers. Note that the local 1 axis is always shown with a red arrow, the local 2 axis is always shown with a white arrow and the local three axis is always shown with a blue arrow.

- **Spandrel Axes:** Arrows indicating local axes orientation are displayed for all specified spandrels. Note that the local 1 axis is always shown with a red arrow, the local 2 axis is always shown with a white arrow and the local three axis is always shown with a blue arrow.

Visible in View

The Visible in View area of the Set Building View Options form allows you to toggle the display of the following items:

- **Story Labels:** This item toggles the labels (names) for story levels on and off in elevation views. Note that the program does not display story level labels in three dimensional views to avoid cluttering the view. Story label names can be edited using the **Edit menu > Edit Story Data > Edit** command.

- **Dimension Lines:** This item toggles the display of dimension lines on and off. Note that dimension lines are only displayed in plan and elevation views, not three-dimensional views.

- **Reference Lines:** This item toggles the display of reference lines on and off. See the section titled "Edit Reference Planes

and Edit Reference Lines Commands" in Chapter 5 Edit Menu for additional information.

- **Reference Planes:** This item toggles the display of reference planes on and off. See the section entitled "Edit Reference Planes and Edit Reference Lines Commands" in Chapter 5 Edit Menu for additional information.

- **Grid Lines:** This item toggles the display of primary grid lines on and off. It does not affect the display of secondary grid lines. Use the **Edit menu > Edit Grid Data** command to control whether a grid line is a primary or secondary grid line.

- **Secondary Grids:** This item toggles the display of secondary grid lines on and off. It does not affect the display of primary grid lines. Use the **Edit menu > Edit Grid Data** command to control whether a grid line is a primary or secondary grid line.

- **Global Axes:** This item toggles the display of global axes on and off. Note that you can use the **View menu > Change Axes Location** command to modify the location of the global axes in a view.

- **Supports:** This item toggles the display of supports (restraints) on and off. ***Note that both this item and the Point Objects box in the Object Visibility area of the Set Building View Options form must be checked for the supports to be visible.*** The four basic graphic symbols used for displaying supports in the ETABS graphical interface are as follows:

- **Springs:** This item toggles the graphical display of grounded point springs (not links) on and off. Note that these are the springs that are assigned using the **Assign menu > Joint/Point > Point Springs** command. Also note that if the Springs check box is checked, the grounded point springs are still not graphi-

cally visible unless the Point Objects check box is also checked.

Special Frame Items

The Special Frame Items area of the Set Building View Options form allows you to toggle the display of various assignments made to line objects. These assignments are only meaningful if the line object is also assigned a frame section property. If the line object is not checked to be visible in the Objects Present in View area of the form, the special frame assignments are not visible even if their box is checked. Following is a list of the specific items you control in this area of the form:

- **End Releases:** This item toggles the display of dots near each end of any line object with frame section properties that has an end release assignment (with or without partial fixity). The color of the dots is based on the default color specified for text in the **Options menu > Colors > Display** command. The end releases are assigned using the **Assign menu > Frame/Line > Frame Releases/Partial Fixity** command.

- **Partial Fixity:** This item toggles the display of text saying "FIX*" without the quotes for any line object with frame section properties that has an end release assignment *with partial fixity specified at one or both ends*. The asterisk has no specific meaning but rather is a convenient method of minimizing any confusion between this text and other labels that may be concurrently displayed. The partial fixity is assigned using the **Assign menu > Frame/Line > Frame Releases/Partial Fixity** command.

- **Moment Connections:** This item toggles the display of triangles at the ends of frame members that are fully fixed. The moment connection symbols are only displayed for beams and braces. They are not displayed for columns.

  The moment connection symbol only appears on beams and braces that have no end releases of any type assigned to them.

If a beam or brace has any type of end release assigned to it (e.g., axial, shear, moment or torsion), the moment connection symbol will not appear for that object.

- **Property Modifiers:** This item toggles the display of text saying "PM*" without the quotes for any line object with frame section properties that is assigned frame property modifiers that are not all ones. The asterisk has no specific meaning but rather is a convenient method of minimizing any confusion between this text and other labels that may be concurrently displayed. The frame property modifiers are assigned using the **Assign menu > Frame/Line > Frame Property Modifiers** command.

- **Nonlinear Hinges:** This item toggles the display of dots together with a text label at the location of each frame nonlinear hinge (pushover) assigned to a line object with frame section properties. The color of the dots is based on the default color specified for text in the **Options menu > Colors > Display** command. The frame nonlinear hinges are assigned using the **Assign menu > Frame/Line > Frame Nonlinear Hinge** command.

- **Panel Zones:** This item toggles the display of a Panel Zone at each joint assigned as a Panel Zone.

Tip:

Do not confuse end offsets and joint offsets.

- **End (Length) Offsets:** This item toggles the display of thickened lines at any end of a line object with frame section properties that has an end offset along the length of the beam assigned to it. The length of the thickened line is scaled to match the specified length of the end offset. The color of the thickened lines is based on the default color specified for text in the **Options menu > Colors > Display** command. The end offset is assigned using the **Assign menu > End (Length) Offsets** command. This command brings up the Assign Frame End Length Offsets form. See Chapter 10 Assign Menu for more information.

- **Joint Offsets:** This item toggles the display of text saying "OFF*" without the quotes for any line object with frame sec-

tion properties that has a frame joint offset assigned to it. The asterisk has no specific meaning but rather is a convenient method of minimizing any confusion between this text and other labels that may be concurrently displayed. The joint offset is assigned using the **Assign menu > Frame/Line > Insertion Point** command. This command brings up the Frame Insertion Point form where you can specify the frame joint offsets from the Cardinal Point.

- **Output Stations:** This item toggles the display of text values reporting either the maximum output station spacing or the minimum number of output stations depending on how the output stations are specified. If the text value is reported in parenthesis, the value represents the minimum number of output stations. If it is not reported in parenthesis, it is the maximum spacing between output stations. The frame output stations are assigned using the **Assign menu > Frame/Line > Frame Output Stations** command.

The graphics or text displayed as a result of checking boxes in this area of the Set Building View Options form alert you that a particular type of assignment is made to a line object with frame section properties, but the display does not tell you the particulars of the assignment because of space limitations. You can always right click on the line object to review its assignments in detail.

Other Special Items

The Other Special Items area of the Set Building View Options form allows you to toggle the display of other miscellaneous items. Following is a list of the specific items you control in this area of the form:

- **Diaphragm Extent:** This item toggles the graphical display of the extent of rigid diaphragms (if any). A large dot is displayed at the center of mass associated with the rigid diaphragm. Dashed lines are drawn from this center of mass point to each point object that is a part of the rigid diaphragm constraint.

Also, text is provided adjacent to the large dot identifying the name of the rigid diaphragm.

Note that the rigid diaphragms are assigned using either the **Assign menu > Joint/Point > Rigid Diaphragm** command or the **Assign menu > Shell/Area > Rigid Diaphragm** command.

- **Auto Floor Mesh:** This item toggles the graphical display of automatic meshing of area objects performed by the program. This feature puts the model in a special display mode where elements are shown rather than objects. In other words, the shell elements in the analysis model are displayed. The model is displayed with the elements shrunken so that you can clearly see the meshing.

  Note that when you select this item, all other items in the Set Building View Options form are grayed out because they will not be displayed on the analysis model.

- **Additional Masses:** This item toggles the display of text values of additional area, line and point masses. The additional area masses are assigned using the **Assign menu > Shell/Area > Additional Area Mass** command. The additional line masses are assigned using the **Assign menu > Frame/Line > Additional Line Mass** command. The additional point masses are assigned using the **Assign menu > Joint/Point > Additional Point Mass** command.

Zoom Commands

There are five zoom features available in this program. Those features allow you to zoom in or out on a view. Zooming in enlarges a smaller area of the model to show more detail, and zooming out decreases detail but shows more of the overall model. All five zoom features are available using the View menu command and the toolbar buttons. The zoom features and their associated toolbar buttons are as follows:

Rubber Band Zoom,

This command allows you to zoom in on the model by windowing. To use the command, depress and hold down the left button on your mouse. While keeping the left button depressed, drag the mouse to "rubber band" a window around the portion of the view that you want to zoom in on. The rubber band window that shows the extent you have dragged the mouse appears as a dashed line on your screen. When you release the mouse left button, the new view is displayed.

Restore Full View,

This command has two uses. First, if you have zoomed in or out from the initial default view of a window, this command returns you to the original default view where the entire structure just fills the window.

The second use for this command occurs if you have used the **View menu > Pan** command to change the view in the window. In that case, the **View menu > Restore Full View** command returns you to the view that you had before you executed the **View menu > Pan** command.

When you use both zoom commands and pan commands together, the program behaves as follows, depending on the order in which you performed the zoom and pan commands:

- ✓ If you first perform a zoom command and then a pan command, clicking the **View menu > Restore Full View** command once returns you to the view you had before executing the pan command. Clicking the **View menu > Restore Full View** command a second time returns you to the original default view for the window where the entire structure just fills the window.

- ✓ If you first perform a pan command and then a zoom command, clicking the **View menu > Restore Full View** command returns you to the original default view for the window where the entire structure just fills the window.

Previous Zoom,

This command takes you back to your immediately previous zoom settings. If you use the **View menu > Previous Zoom** command repeatedly without using other commands to change the zoom in between, the effect is to toggle between two zoom settings. You cannot use the **View menu > Previous Zoom** to go back more than one zoom setting.

The **View menu > Previous Zoom** command has no effect in the following circumstances:

- ✓ Immediately after you first display a view in a window.
- ✓ Immediately after you use the **View menu > Pan** command.

Zoom In One Step,

This command zooms in on the model one step. The size of the step is controlled by the Auto Zoom Step item in the Preferences form, which can be accessed using the **Options menu > Preferences > Dimensions/Tolerances** command.

The program default value for the Auto Zoom Step is 10 percent. This means that when you use the **View menu > Zoom In One Step** command, the program increases the magnification of all objects in the view by 10 percent.

Zoom Out One Step,

This command zooms out on the model one step. The size of the step is controlled by the Auto Zoom Step item in the Preferences form, which can be accessed using the **Options menu > Preferences > Dimensions/Tolerances** command.

The program default value for the Auto Zoom Step is 10 percent. This means that when you use the **View menu > Zoom Out One Step** command, the program decreases the magnification of all objects in the view by 10 percent.

Pan Feature

The pan feature allows you to move a view within the window such that you can see beyond the original edges of the view. The distance you can move beyond the original edge of the view is controlled by the Pan Margin item that is set in the preferences. The **Options menu > Preferences > Dimensions/Tolerances** command gives you access to the Pan Margin preference item. See the subsection entitled "Dimensions and Tolerances Command" under the section entitled "Preferences" in Chapter 14 Options Menu for more information on the Pan Margin item.

Click the **View menu > Pan** command or the **Pan** button to pan a view. After you have used the menu command or clicked the button, click and hold down the left mouse button in the view and drag the mouse (while still holding down the left mouse button) to pan the view.

You must reuse the menu command or re-click the button every time you pan. If you have panned a view one or more times, clicking the **View menu > Restore Full View** command, or its associated button returns you to the view you had before you executed the first **View menu > Pan** command.

Measure Command

Tip:

*You can use the **Draw menu > Draw Dimension Line** command to draw dimension lines that include dimension text (measurements).*

You can use the **View menu > Measure** command to make measurements in your model. You can measure lines, areas and angles. Each of these are described below.

- **Lines:** When you execute the **View menu > Measure > Line** command, left click on two points to define the line you want to measure and the program will report the length of the line in the status bar at the bottom of the program window.
- **Areas:** When you execute the **View menu > Measure > Area** command, left click on the corner points of an area you want to measure and the program will report the area and perimeter of the area in the status bar at the bottom of the program window.

When defining the last point for the area, either double left click, or single left click and press the Enter key (or Esc key) on your keyboard.

- **Angles:** When you execute the **View menu > Measure > Angle** command, left click on three points to define two lines that have one common endpoint. The program reports the angle between these lines in the status bar at the bottom of the program window. The angle is always reported in degrees and it is always less than or equal to 180 degrees.

Note the following about using the **View menu > Measure** command.

- After you have drawn the line, area or angle and reviewed the measurement, click anywhere and the drawn line, area or angle will be deleted.
- The measurements are always reported in the current units.

Change Axes Location Command

By default, the global axes are shown at the global origin. You can use the **View menu > Change Axes Location** to display the Axes Location form that allows you to specify a new location for the global axes. The new location is specified by entering global X, Y and Z coordinates.

Note the following about the global axes:

- A default view for a window is not scaled such that the global axes fit in the window. Thus if you locate the global axes a large distance from the rest of your structure, they may not be visible in a default view. (The default view for a window is scaled based on story levels, grid lines and area, line and point objects.)

- You can control the visibility of the global axes using the **View menu > Set Building View Options** or the corresponding **Set Building View Options** button .

Show Selection Only and Show All Commands

Sometimes you may find that there are too many objects in a view for you to see necessary details. In such cases, you may want to select just a few of the objects in the view and use the **View menu > Show Selection Only** command. This command will refresh the view such that only the selected items are visible in the window. If you change the view type in the window, say from a plan view to a 3D view, the originally selected items continue to be the only ones visible. Use the **View menu > Show All** command to remove the effects of the **View menu > Show Selection Only** command.

Save Custom View and Show Custom View Commands

The **View menu > Save Custom View** command allows you to give any view a name and then save it. You can then later use the **View menu > Show Custom View** command to restore your named custom view.

These commands can be useful if you are going to use a certain view over and over again and it takes you significant time to create the view. For example, if you have a view with special limits set and selected objects only displayed and a special zoom set and a special view angle set, you may want to save it as a custom view so that you can easily recreate it at a later time.

Refresh Window and Review View Commands

The **View menu > Refresh Window** command and the corresponding **Refresh Window** button are used to refresh the

view after drawing or editing objects. This command redraws what is visible on the screen but *does not* rescale it in any way.

The **View menu > Refresh View** command is also used to refresh the view after drawing or editing objects. This command redraws what is visible on the screen and returns the view to its default full view where the entire model is visible. Note that the default view is scaled based on all grid lines, story levels, and objects being included in the view. Thus if any new grid lines, story levels, or objects have been added outside of the original model boundaries, this command rescales the default full view such that all grid lines, story levels, and objects fit into it.

The refresh view and refresh window commands are similar. However, unlike the refresh view command, the refresh window command does not rescale the window or return it to a default view.

Show Rendered View

Tip:

Click the Reset and Refresh View *button in the Model Viewer to display the Rendered View.*

Clicking the **View menu > Show Rendered View** command brings up the Model Viewer window. A rendering of your model is displayed when you click the **Reset and Refresh View** button in the upper left-hand corner of the Model Viewer window. The Model Viewer renders the view that is shown in the active window (i.e., plan, elevation, or 3D) at the time the **View menu > Show Rendered View** command is selected.

This command gives you a realistic, extruded view of the structural objects that you have defined in your model. You can change the angle and position of the view by clicking on the rendering in the Model Viewer window and holding the left button down while you move the mouse. You can also zoom in or out of the view by clicking and holding down the left or right mouse button, respectively, and NOT moving the mouse.

The **View menu > Show Rendered View** command is a powerful tool for verifying the geometry of your model.

Chapter 7

Define Menu

General

The Define menu provides a means of defining section properties and load case definitions. This chapter describes the features available on the Define menu.

Items related to ETABS nonlinear analysis are mentioned in passing but elaboration on those items is beyond the scope of this manual. Consult the Technical Notes for more in-depth descriptions of items related to nonlinear analysis.

Material Properties Command

The material properties in this program are always linear elastic. Use the **Define menu > Material Properties** command or button to define material properties. This command brings up the

Define Materials form where the names of all defined material properties are listed. In this form you can do the following:

- Click the **Add New Material** button to display the Material Property Data form where you can define new material properties.
- Highlight a material property name and click the **Modify/Show Material** button to display the Material Property Data form where you can review and/or modify the material properties for the highlighted material property.
- Click the **Delete Material** button to delete an existing material property. Note that you cannot delete two built-in material properties that are named STEEL and CONC. You also can not delete any material property that is currently specified in the definition of a frame section property or a wall/slab/deck section property. In other words, you cannot delete a material property if it is in use.

Material Property Data Form

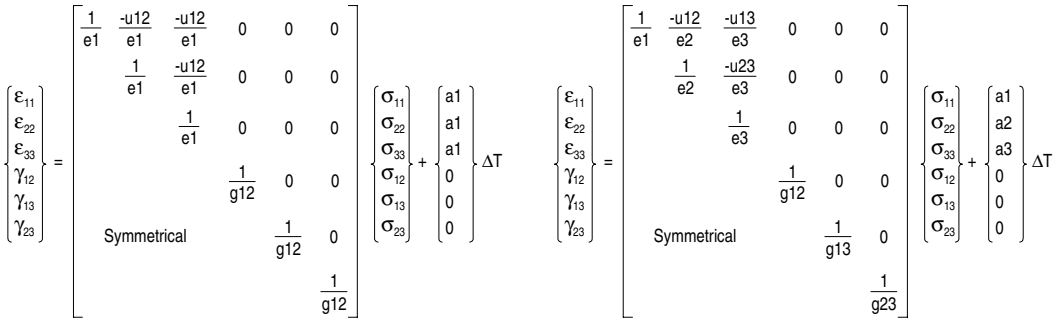
The Material Property Data form consists of six different areas. They are:

Note:

You can specify isotropic or orthotropic material properties in ETABS.

- **Material name:** Specifies or modifies the name of a material property. Note that you cannot change the name of the built-in STEEL and CONC material properties.
- **Type of material:** Specifies the material to be isotropic or orthotropic. The option chosen here affects what is shown in the Analysis Property Data area of the form.

  The behavior of an *isotropic* material is independent of the direction of loading. In addition, the shearing behavior is uncoupled from the extensional behavior and it is not affected by temperature change. Isotropic behavior is usually assumed for steel and concrete, although that is not always the case.



a) Isotropic Material

b) Orthotropic Material

(Above)
Figure 7-1:
Illustration of how mechanical and thermal properties relate strain to stress and temperature change for isotropic and orthotropic materials

The isotropic mechanical and thermal properties relate strain to stress and temperature change as shown in Figure 7-1a. In the figure, **e1** is Young's modulus of elasticity, **u12** is Poisson's ratio, **g12** is the shear modulus and **a1** is the coefficient of thermal expansion.

The shear modulus is not directly specified for an isotropic material. Instead, the program derives it from the specified Young's modulus and Poisson's ratio as shown in Equation 7-1.

$$g12 = \frac{e1}{2\left(1 + u12\right)} \qquad \text{Eqn. 7-1}$$

Note that in this program, Poisson's ratio must satisfy the condition that $0 \leq u12 < 0.5$ and that Young's modulus must be positive.

The behavior of an *orthotropic* material can be different in the three local axis directions. However, like an isotropic material, the shearing behavior is uncoupled from the extensional behavior and it is not affected by temperature change.

The orthotropic mechanical and thermal properties relate strain to stress and temperature change as shown in Figure 7-1b. In the figure **e1, e2** and **e3** are the moduli of elasticity, **u12, u13** and **u23** are the Poisson's ratios, **g12, g13** and **g23** are the shear moduli and **a1, a2** and **a3** are the coefficients of thermal expansion.

Note that in this program, for orthotropic materials, the elastic moduli and the shear moduli must be positive. The Poisson's ratios may take on any values, provided that the upper left 3x3 portion of the stress-strain matrix is positive definite (i.e., has a positive determinant.) The check for this is made at analysis runtime, not when the values are entered.

- **Analysis Property Data:** Specifies the mass per unit volume, weight per unit volume, modulus of elasticity, Poisson's ratio, coefficient of thermal expansion, and if you are specifying an orthotropic material, the shear modulus.

  The mass per unit volume is used in calculating the self-mass of the structure if you have specified that mass is to be determined from element and additional masses. The weight per unit volume is used in calculating the self-weight of the structure.

  For isotropic materials, define one value for the modulus of elasticity, Poisson's ratio and coefficient of thermal expansion. The shear modulus is calculated as previously described for Equation 7-1.

  For orthotropic materials, define three values for the modulus of elasticity (one for each local axis direction), the Poisson's ratio, the coefficient of thermal expansion and the shear modulus.

- **Display Color:** Assigns a color to the material property. If you use the **View menu > Set Building View Options** command or button to display the Set Building View Options form, you can then choose an option to view the model based on the colors associated with the material properties. In that case, each object appears in a color associated with its assigned material property. See the section entitled "Set Building View Options Command" in Chapter 6 View Menu for more information. You can change the color associated with the material by clicking in the color box.

- **Type of Design:** Here you can specify the type of design as Steel, Concrete or None. The option you specify here affects what is shown in the Design Property Data area of the form.

  The Steel Frame Design and Composite Beam Design postprocessors do not design members unless (among other things) the type of design specified for their associated material property is Steel.

  The Concrete Frame Design and Shear Wall Design postprocessors do not design members unless (among other things) the type of design specified for their associated material property is Concrete.

- **Design Property Data:** The data specified in this area depends on the design type specified in the Type of Design area of the form. In general, the Design Property data specified in the Material Property Data form is used only by the design postprocessors. The one exception to this is that for any degree of freedom in the frame nonlinear hinge properties that is specified as default, the program calculates the hinge force-deformation properties based on the Design Property data provided here.

Note:

The shear strength reduction factor multiplies the calculated concrete shear strength. This reduction factor is used for all shear calculations whenever lightweight concrete is specified.

If the type of design is **Steel,** the following items are specified:

- ✓ Minimum yield stress, Fy
- ✓ Minimum tensile stress, Fu
- ✓ Cost per unit weight

The cost per unit weight item is used in the Composite Beam Design postprocessor when the optimum beam size may be determined based on the cost of the beam, connectors and camber, rather than just the area (weight) of the beam.

If the type of design is **Concrete,** the following items are specified:

- ✓ **Specified Concrete Compressive Strength, f'c:** This item is used in all calculations.
- ✓ **Bending Reinforcing Yield Stress, fy:** This is the reinforcing steel yield stress used in the calculations for bending and axial load calculations.
- ✓ **Shear Reinforcing Yield Stress, fys:** This is the reinforcing steel yield stress used in the calculations for shear.
- ✓ **Lightweight Concrete:** Check this check box if you have lightweight concrete. Checking this check box enables the shear strength reduction factor edit box.
- ✓ **Shear Strength Reduction Factor:** If the lightweight concrete check box is checked, then *for all shear calculations,* the calculated concrete shear strength is multiplied by this factor. Typically, this reduction factor is between 0.75 and 0.85.

If the type of design is **None,** nothing is specified in the Design Property Data area.

Frame Sections Command

Use the **Define menu > Frame Sections** command or button to define frame section properties. This command brings up the Define Frame Properties form. The Properties area of this form lists the names of all the currently defined frame section properties. The Click To area of the form allows you to define new frame sections, modify existing frame section definitions and delete existing frame sections. Note that you can only delete frame sections if they are not currently assigned to any line objects in your model and if they are not used to define other frame section properties, such as nonprismatic sections and auto select section lists.

Importing Sections from a Database

Shortcut:

*Use the **Assign menu > Frame/Line > Frame Section** command to simultaneously define frame sections and assign them to selected line objects.*

The drop-down box that initially reads "Import I/Wide Flange" in the Click To area of the Frame Properties form allows you to import many different types of frame section properties from one of several section databases that are included with this program. The types of section properties you can import are as follows:

- I-shaped members, including wide flange sections
- Channels
- Double channels
- Structural tees
- Single angles
- Double angles
- Structural box/tubes
- Pipe sections
- Rectangular sections
- Circular sections
- General sections
- Steel Joists

Note that the Rectangular, General and Circular sections in the above list are not available in the section databases provided with the program but could be in user-created section databases.

Typically, the sections in the database file have both section properties (e.g., area, moment of inertia) and specified dimensions. General sections have section properties associated with them but no dimensions specified for them. Steel joists have section properties specified, or total load and live load capacities are specified for a range of joist spans, but no dimensions are specified.

To import a section from a database, click the **Define menu > Frame Sections** command or button, click on one of the section types in the Import drop-down box and specify a database file from which to choose the section, if necessary.

The default name for the frame sections database file is Sections.pro. Note that the steel joist sections are kept in a separate database file with the default name of Joists.pro. Steel joist section properties are described later in this chapter in the section entitled "Steel Joist Section Properties."

The program then displays a list of all of the sections of the type specified in the database. You can select one or more sections from the list by clicking on them. Following are some possible methods of multiple selection:

- Select one section by left clicking on it and continuing to hold down the mouse button while dragging your mouse up or down to select additional adjacent sections.
- Select one section by left clicking on it. Then hold down the Shift key on your keyboard and select another section. The second section is added to the selection as well as all sections between the first and second section.
- Select one section by left clicking on it. Then hold down the Ctrl key on your keyboard and select another adjacent or non-adjacent section. That section is added to the selection. You can continue holding down the Ctrl key and clicking on other sections to add them to the selection.

Adding User-Defined Frame Section Properties

The drop-down box that initially reads "Add I/Wide Flange" in the Click To area of the Frame Properties form allows you to easily define section properties for many different types of frame sections. The types of section properties you can define are as follows:

- I-shaped members

- Channels
- Double channels
- Structural tees
- Single angles
- Double angles
- Structural tubes
- Pipe sections
- Rectangular sections
- Circular sections
- General sections
- Steel joist sections
- Auto select section list
- Sections defined in the Section Designer utility
- Nonprismatic sections

For all but the last five items in the above list, simply specify dimensions for the section and the program will automatically calculate the section properties. For general sections, simply specify the section properties (e.g., area, moment of inertia, shear area). No dimensions are input for general sections. Figure 7-2 is provided to help you determine the shear area for general sections of various shapes.

Steel joist section properties are described later in this chapter in the section entitled "Steel Joist Section Properties."

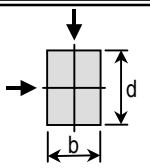
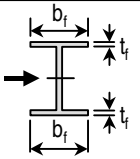
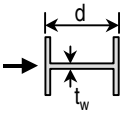
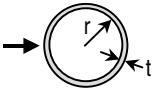
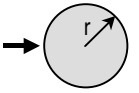
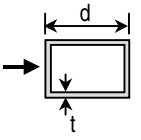
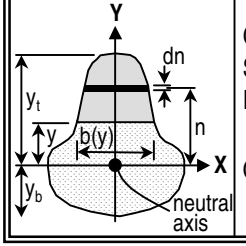
Section	Description	Effective Shear Area
	Rectangular section: Shear forces parallel to the b or d directions	$\frac{5}{6}$ bd
	Wide flange section: Shear forces parallel to flange	$\frac{5}{3}$ t_f b_f
	Wide flange section: Shear forces parallel to web	t_w d
	Thin walled circular tube section: Shear forces from any direction	π r t
	Solid circular section: Shear forces from any direction	0.9 π r^2
	Thin walled rectangular tube section: Shear forces parallel to d-direction	2 t d
	General section: Shear forces parallel to Y-direction I_X = Moment of inertia of section about X-X $Q(y) = \int_y^{y_t} n\, b(n)\, dn$	$\frac{I_X^2}{\int_{y_b}^{y_t} \frac{Q^2(y)}{b(y)} dy}$

Figure 7-2: *Shear Areas for Various Sections*

Auto select section lists are simply lists of previously defined steel sections. These are useful for Steel Frame Design and Composite Floor Design because the program can pick the optimal section for a steel frame element from an Auto Select Section List. There must be at least two steel frame sections defined before you are allowed to define an auto select section list.

Tip:

Use the Section Designer utility to graphically define frame sections. Select the Add SD Section option in the Define Frame Properties form.

You can use the Section Designer utility to graphically define unusual sections. The program then calculates the section properties for that section. See the section entitled "Adding Frame Section Properties using Section Designer" later in this chapter for more information.

You can use the Add Nonprismatic feature to define nonprismatic frame sections where the section properties vary along the length of the frame element. See the section entitled "Nonprismatic Sections" later in this chapter for more information.

When you specify concrete frame sections, you can also specify some of the reinforcing information. See the section entitled "Reinforcing for Concrete Frame Section Properties" later in this chapter for more information.

Steel Joist Section Properties

The database file for joist sections includes the Steel Joist Institute (SJI) K-Series, KCS-Series, LH-Series and DLH-Series joists. The capacities used for these items are those published by the SJI.

The K-, LH- and DLH-Series joist capacities are all defined by total load capacities and live load capacities that produce an L/360 deflection specified for a range of joist spans. This is referenced in the program as Standard design capacities. The KCS-Series joist capacity is specified by a moment capacity, a shear capacity, a moment of inertia, and minimum and maximum spans for which the joist can be used. This is referenced in the program as Envelope design capacities

In addition to the joist section properties that can be imported from the database file, you can also define user joist section properties. The capacities of the user joists can either be defined as Standard design capacities (similar to the K-Series joists) or as Envelope design capacities (similar to the KCS-Series joists).

Note that you can not directly modify the joist section properties that are imported from the joist database file. However, upon import, you can convert those imported sections to user sections that can then be modified.

When you define steel joist properties, there is a moment of inertia term called I33 for Analysis. This moment of inertia is used in the analysis model. It is also used to calculate the joist deflection for joists with Envelope design (e.g., KCS-Series joists). In addition, the Standard Design joists have an I33 specified for each span at which capacity data is specified. This I33 value is only used when displaying the deflection diagram that is available when performing Interactive Joist Design.

Adding Frame Section Properties Using Section Designer

Select the Add SD Section option in the Define Frame Properties form to define a frame section property using the Section Designer feature. This brings up the SD Section Data form. The following areas are in this form:

- **Section Name:** You can specify or modify the name of the frame section.
- **Base Material:** You can specify or modify a defined material property. The options available in the Design Type area of the form will depend on the type of material selected here.
  - ✓ If the Base Material is Concrete, the Design Type options available are No Check/Design and the Concrete Column options.

- ✓ If the Base Material is Other, the Design Type option available is No Check/Design.
- ✓ If the Base Material is Steel, the Design Type options available are No Check/Design and the General Steel Section.

- **Design Type:** You can specify the section type, given the restrictions outlined in the previous bullet. If the Design Type is No Check/Design (not designed), any frame section assigned this property is not designed by any postprocessor. If the Design Type is General Steel Section, any frame section assigned this property is designed by the Steel Frame Design postprocessor as a general section. If the Design Type is Concrete Column, any frame section assigned this property is designed by the Concrete Frame Design postprocessor.
- **Concrete Column Check/Design:** This area is only active if the Concrete Column option is selected in the Design Type area. Here you specify whether the concrete column is to have its specified reinforcing checked or new longitudinal reinforcing designed when it is run through the Concrete Frame Design postprocessor.
- **Define/Edit/Show Section:** After you have appropriately specified items in the other areas of the form, click the **Section Designer** button in this area to go to the Section Designer utility and draw the section. When you exit the Section Designer utility, you return to the SD Section Data form. You can then click the **OK** button to complete the definition of the frame section property.

Nonprismatic Sections

Nonprismatic frame sections may be defined for which the properties vary along the element length. You may specify that the element length be divided into any number of segments; they do not need to be of equal length. Most common situations can be modeled using from one to five segments.

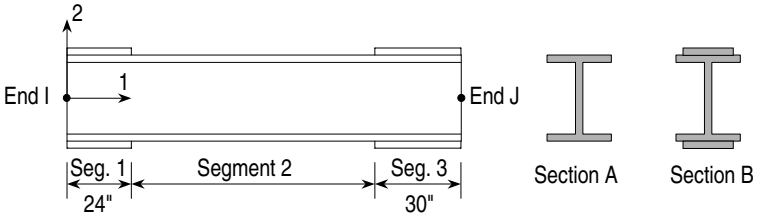
Note:

ETABS analyzes nonprismatic sections. The Steel Frame Design and the Concrete Frame Design postprocessor can design nonprismatic sections. The Composite Beam Design postprocessors do not currently design nonprismatic sections.

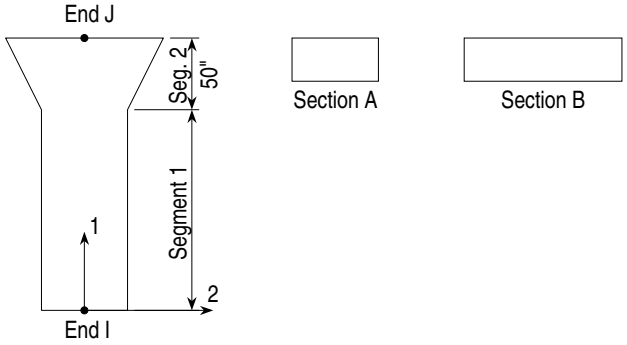
The variation of the bending stiffnesses may be linear, parabolic, or cubic over each segment of length. The axial, shear, torsional, mass, and weight properties all vary linearly over each segment. Section properties may change discontinuously from one segment to the next.

See Figure 7-3 for examples of nonprismatic frame sections. Figure 7-3a shows a steel beam with cover plates at the ends. Section A is the section without cover plates and section B is the section with cover plates. You might define section B by clicking the **Define menu > Frame Sections** command, selecting "Add SD Section" from the Add drop-down box and drawing the section in the Section Designer utility (separate instructional documentation is available for the Section Designer utility).

Figure 7-3: *Nonprismatic frame section examples*



a) Steel Beam with Cover Plates at Ends



b) Concrete Column with Flare at Top

After both section A and section B have been defined, you can define the nonprismatic section using the **Define menu > Frame Sections** command and selecting "Add Nonprismatic" from the

Add drop-down box to display the Nonprismatic Section Definition form. Table 7-1 shows the assignments that would be entered in the form. Note that the variation items grayed out in the table are not used by the program because the start section and the end section are the same.

In Table 7-1 note that segment 2 has a length type of variable and a segment length of 1. See the subsection entitled "Segment Lengths and Segment Type" for an explanation of these parameters.

Table 7-1: Input for Nonprismatic Frame Section Example in Figure 7-3a

Segment	Start Section	End Section	Length	Length Type	EI33 Variation	EI22 Variation
1	B	B	24	Absolute		
2	A	A	1	Variable		
3	B	B	30	Absolute		

Figure 7-3b shows a concrete column with a flare at the top. Section A is the section at the lower portion of the column and section B is the section at the top of the column. You might define both section A and B by clicking the **Define menu > Frame Sections** command, and selecting "Add Rectangular" from the Add drop-down box.

After both section A and section B have been defined, you can define the nonprismatic section using the **Define menu > Frame Sections** command and selecting "Add Nonprismatic" from the Add drop-down box to display the Nonprismatic Section Definition form. Table 7-2 shows the assignments that would be entered into the form. Note that the variation items grayed out in the table are not used by the program because the start section and the end section are the same.

Table 7-2: Input for Nonprismatic Frame Section Example in Figure 7-3b

Segment	Start Section	End Section	Length	Length Type	EI33 Variation	EI22 Variation
1	A	A	1	Variable		
2	A	B	50	Absolute	Cubic	Linear

Nonprismatic Section Definition Form Buttons

The following buttons appear on the Nonprismatic Section Definition form and complete the actions described when used with the Start Section and End Section drop-down boxes and the Length edit box.

- **Add**: This button adds the selected parameters to your segment definition. First, highlight the name of the section where you want to start your segment in the Start Section drop-down box. Then highlight the name of the section where you want your segment to end in the End Section drop-down box. Continue defining your segment by typing in the desired length in the Length edit box; and the desired parameters from the drop-down boxes for length type and variations. Click the **Add** button to complete your definition.

  Note that clicking the **Add** button again quickly repeats the definition. Repeating the definition can be helpful if you want to define a second segment that has only one parameter that needs to be modified, such as the Start Section selection. See the Modify bullet item for an explanation of how to modify a parameter or segment definition.

- **Insert**: This button inserts a selection into your list of defined segments. Assume you have already defined segments X and Y and that you want to insert segment A after segment X and before segment Y. To do this, highlight segment Y, define segment A by selecting from the drop-down boxes and using the edit boxes. When the segment is fully defined, click the **Insert** button. You can repeat the insertion by clicking the **Insert** button.

- **Modify**: To change any parameter, highlight the parameter that you want to change in the list of defined segments. Then click the pull down box and highlight the new selection or type the new parameter in the edit box before clicking the **Modify** button.

- **Delete**: To delete a segment, highlight the segment and then click the **Delete** button.

Starting and Ending Sections

The properties for a segment of a nonprismatic section are defined using the drop-down boxes on the Nonprismatic Section Definition form. From these drop-down boxes, select the following:

- The section name of a previously defined prismatic section that defines the properties at the start of the segment, i.e., at the end closest to joint i.

- The section name of a previously defined prismatic section that defines the properties at the end of the segment, i.e., at the end closest to joint j. The starting and ending sections may be the same if the properties are constant over the length of the segment.

The material would normally be the same for both the starting and ending sections and only the geometric properties would differ, but this is not required.

Segment Lengths and Length Type

When a nonprismatic frame section is assigned to an element, the actual lengths of each segment for that element are determined as follows:

- The clear length of the element, Lc, is first calculated as the total length minus the end offsets: Lc = L - (ioff + joff). In this equation L is the full length of the frame element and ioff and

joff are the lengths of the end offsets along the length of the frame element at the i and j ends of the element respectively.

- If the sum of the absolute lengths of the segments exceeds the clear length, they are scaled down proportionately so that the sum equals the clear length. Otherwise, the absolute lengths are used as specified.

1st Segment:

$$\frac{vl1}{vl1 + vl2} = \frac{1}{1+2} = \frac{1}{3}$$

2nd Segment:

$$\frac{vl2}{vl1 + vl2} = \frac{2}{1+2} = \frac{2}{3}$$

- The remaining length (the clear length minus the sum of the absolute lengths) is divided among the segments having variable lengths in the same proportion as their specified lengths. For example, for two segments with variable lengths specified as vl1 = 1 and vl2 = 2, one-third of the remaining length goes to the first segment, and two-thirds to the second segment. See the calculations to the left.

The length of a nonprismatic segment may be specified as either a variable length or an absolute length.

Variation of Properties

Nonprismatic column/beam/brace section properties are interpolated along the length of each segment from the values at the two ends. The variation of the bending stiffnesses, EI_{33} and EI_{22}, along the length of the segment is specified as linear, parabolic, or cubic; the variation type can be changed using the drop-down box on the Nonprismatic Section Definition form.

Specifically, the linear, parabolic or cubic variation for EI_{33} is calculated by the program as follows:

- **Linear:** The value EI_{33} varies linearly along the length of the segment.
- **Parabolic:** The value $\sqrt[2]{EI_{33}}$ varies linearly along the length of the segment.
- **Cubic:** The value $\sqrt[3]{EI_{33}}$ varies linearly along the length of the segment.

This usually corresponds to a linear variation in one of the section dimensions. For example, a linear variation in the width of a rectangular shape yields a linear variation for EI_{33}. A linear variation in the depth of a rectangular shape yields a cubic variation for EI_{33}. Finally, a linear variation in the depth of an I-shape yields a parabolic variation for EI_{33}.

The interpolation of the bending stiffness in the 1-2 plane, EI_{22}, is defined in the same manner to that for the 1-3 plane.

The remaining stiffness properties, other than EI_{33} and EI_{22}, are always assumed to vary linearly between the ends of each segment. Similarly, the mass and weight densities are always assumed to vary linearly between the ends of each segment.

If a shear area is zero at either end, it is taken to be zero along the full segment, thus eliminating all shear deformation in the corresponding bending plane for that segment.

Effect upon End Offsets Along the Length of Frame Elements

Frame section properties vary only along the clear length of the element. Section properties within the longitudinal end offset at the i-end of the element are constant using the starting section of the first segment. Section properties within the end offset at the j-end of the element are constant using the ending section of the last segment. Note that if a longitudinal end offset rigidity factor is specified, the specified part of the end offset is rigid and the rest has the section property described above.

Reinforcing for Concrete Frame Section Properties

When you specify frame section properties for rectangular or circular concrete members you can also specify some of the reinforcing information for that member. When you use the Add Rectangle or Add Circle option in the Click To area of the Define Frame Properties form, the appropriate Rectangular Section

or Circular Section form will appear. If the Material is set to CONC (concrete) on the form, a **Reinforcement** button will appear. Clicking the **Reinforcement** button brings up the Reinforcement Data form. The Design Type on this form can be set to Column or Beam.

Reinforcing Information for Columns

Note:

You can specify reinforcing information for rectangular, T-shaped and L-shaped concrete beam sections and for circular and rectangular concrete column sections. Reinforcing for other column sections can be specified using the Section Designer utility.

For columns the following areas are provided in the Reinforcement Data form:

- **Configuration of Reinforcement:** Specifies rectangular or circular reinforcement. You can, if desired, put circular reinforcement in a rectangular column or put rectangular reinforcement in a circular column.
- **Lateral Reinforcement:** If the Configuration of Reinforcement is Rectangular, the Lateral Reinforcement can only be Ties. If the Configuration of Reinforcement is Circular, the Lateral Reinforcement can be Ties or Spiral (transverse).
- **Rectangular Reinforcement:** When the Configuration of Reinforcement is Rectangular, the following options are available in this area.
  - ✓ **Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar.

    In the special case of rectangular reinforcement in a circular column, the cover is taken to be the minimum distance from the edge of the column to the center of a corner bar of the rectangular reinforcement pattern.
  - ✓ **Number of bars in 3-dir:** This is the number of longitudinal reinforcing bars (including corner rebar) on the two faces of the column that are parallel to the local 3-axis of the section.
  - ✓ **Number of bars in 2-dir:** This is the number of longitudinal reinforcing bars (including corner rebar) on the two

Note:

Cover is typically measured from the nearest edge of the concrete section to the centroid of the reinforcing steel.

faces of the column that are parallel to the local 2-axis of the section.

- ✓ **Bar size:** This is the specified size of reinforcing steel for the section. You can only specify one bar size for a given concrete frame section property.

- **Circular Reinforcement:** This area is visible if you have chosen a circular configuration of reinforcement. The following options are available in this area.

  - ✓ **Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar.

    In the special case of circular reinforcement in a rectangular column, the cover is taken to be the minimum distance from the edge of the column to a circle drawn through the center of all the rebar in the circular reinforcement pattern.

  - ✓ **Number of bars:** This is the number of longitudinal reinforcing bars in the section.

  - ✓ **Bar size:** This is the specified size of reinforcing steel for the section. You can only specify one bar size for a given concrete frame section property.

- **Check/Design:** In this area you specify Reinforcement to be Checked or Reinforcement to be Designed when a member with this frame section property is run through the Concrete Frame Design postprocessor. All information in the Reinforcement Data form is used in checking reinforcement. If the reinforcement is to be *designed,* all information in the Reinforcement Data form is used, except the bar size, which is ignored, and the total required steel area is calculated. For design, the configuration of reinforcement, lateral reinforcement and cover are used.

If you specify reinforcing in a concrete column frame section property that is assigned using the Section Designer utility, the Concrete Frame Design postprocessor either checks the column

for the specified reinforcing or designs new reinforcing, depending on the option you selected when you specified the section.

Reinforcing Information for Beams

For concrete beams, you specify two types of reinforcing information: Concrete Cover to Rebar Center and Reinforcement Overrides for Ductile Beams.

Rebar cover is specified at the top and bottom of the beam. The top cover is measured from the top of the beam to the centroid of the top longitudinal reinforcing. The bottom cover is measured from the bottom of the beam to the centroid of the bottom longitudinal reinforcing.

Note:

The reinforcing data specified for concrete frame sections is used by the Concrete Frame Design postprocessor. It is also used to determine default nonlinear hinge (pushover) properties for concrete members. It is not used to modify the analysis properties of the section. They are based on the gross section properties.

The reinforcement overrides are specified areas of longitudinal reinforcing steel that occur at the top and bottom of the left and right ends of the beam. These overrides are used by the program as follows:

- In the Concrete Frame Design postprocessor, when the design shear in a concrete beam is to be based on provided longitudinal reinforcement (that is, the shear design is based on the moment capacity of the beam), the program compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity on which the shear design is based.
- In the Concrete Frame Design postprocessor, when the minimum reinforcing in the middle of a beam is to be based on some percentage of the reinforcing at the ends of the beam, the program compares the calculated required reinforcement at the ends of the beam with that specified in the reinforcement overrides and uses the larger value to determine the minimum reinforcing in the middle of the beam.
- In the Concrete Frame Design postprocessor, when the shear design of columns is to be based on the maximum moment that the beams can deliver to the columns, the program compares

the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity of the beam.

- For any degree of freedom in the frame nonlinear hinge properties assigned to a concrete member that is specified as default, the program calculates the hinge force-deformation properties based on the larger of the calculated required reinforcement at the ends of the beam (assuming you have run the design through the Concrete Frame Design postprocessor) and the specified reinforcement overrides.

Wall/Slab/Deck Sections Command and Form

Use the **Define menu > Wall/Slab/Deck Sections** command or button to define wall, slab or deck section properties. The command or button brings up the Define Wall/Slab/Deck Sections form. The Sections area of this form lists the names of all the currently defined deck, slab, and wall section properties. The Click To area of the form has a drop-down box that allows you to Add New Deck, Add New Slab, or Add New Wall as well as **Modify/Show Section** and **Delete Section** buttons. Note that you can only delete deck, slab and wall sections if they are not currently assigned to any area objects in your model.

Wall/Slab Sections Form

When you select Add New Slab or Add New Wall in the drop-down box, or click the Modify/Show Section bottom and select a wall or slab section in the Define Wall/Slab/Deck Sections form, the Wall/Slab Section form appears. Following is a discussion of each of the areas in that form.

- **Section name:** Specify the name of a wall or slab section. You can modify this name if desired.

- **Material:** Here you can choose the material property for the slab or wall from a list of all defined material properties.

Shortcut:

*Use the **Assign menu > Shell/Area > Wall/Slab/Deck Section** command to simultaneously define wall, slab and deck sections and assign them to selected area objects.*

- **Thickness:** Two thicknesses are specified: membrane and bending. Typically these thicknesses are the same but they can be different. For instance, they may be different if you are trying to model full shell behavior for a corrugated metal deck.

  The membrane thickness is used for calculating:

  - ✓ The membrane stiffness for full shell and pure membrane sections.
  - ✓ The element volume for element self-mass and self-weight calculations.

  The bending thickness is used for calculating the plate-bending and transverse-shearing stiffnesses for full shell and pure plate sections.

- **Type:** A wall or slab section can have Shell, Membrane or Plate-type behavior. Membrane-type behavior means that only in-plane membrane stiffness is provided for the section. Plate-type behavior means that only out-of-plane plate bending stiffness is provided for the section. Shell-type behavior means that both in-plane membrane stiffness and out-of-plane plate bending stiffness are provided for the section.

  When a section has plate-type or shell-type behavior, you have the option of including or not including Thick Plate behavior. When thick plate behavior is included (i.e., the Thick Plate check box is checked), out-of-plane shearing deformations are considered in the analysis. When thick plate behavior is *not* included (i.e., the Thick Plate check box is not checked), these shearing deformations are not considered in the analysis.

  We recommend that in this program, you typically do not use the thick plate option, except perhaps when you are modeling thick footings or mat foundations.

- **Display Color:** Here you assign a color to the wall or slab section. If you use the **View menu > Set Building View Options** command to display the Set Building View Options

form, you can then choose an option to view the model based on the colors associated with the section properties. In that case, each object appears in a color associated with its assigned section property. See the section entitled "Set Building View Options Command" in Chapter 6 View Menu for more information. You can change the color associated with the material by clicking in the Display Color box.

Deck Sections Form

When you select Add New Deck from the drop-down box or the Modify/Show Section button and select at deck section in the Define Wall/Slab/Deck Sections form, the Deck Section form appears. Following is a discussion of each of the areas in this form.

Tip:

If you want to use the Composite Beam Design post-processor, you must define the slab using Deck properties, not slab properties, even if you are using a solid slab over the composite beams.

- **Section name:** Specifies the name of a deck section; you can modify this name if desired.

- **Type:** The three options for the deck type are filled deck, unfilled deck and solid slab. The type of deck section controls which features are active in the rest of the form. Following is a description of the three deck type options.

  - ✓ **Filled deck:** If you select the filled deck option, all items in the Geometry and Composite Deck Studs areas are active and the Slab Material item in the Material area is active.

  - ✓ **Unfilled deck:** If you select the unfilled deck option, the slab cover item in the Geometry area (Slab Depth) is set to zero and grayed out (inactive), the entire Composite Deck Studs area is grayed out and the Deck Material and Deck Shear Thickness items in the Material area are active.

  - ✓ **Solid slab:** If you select the solid slab option, the deck depth, rib width and rib spacing items in the Geometry area are set to zero and grayed out (inactive), the entire Composite Deck Studs area is active and the Slab Material item in the Material area is active.

Note:

The deck always spans in the same direction as the local 1-axis of the area object that it is assigned to. You can use the ***Assign menu > Shell/ Area > Local Axes*** *command to change the direction of the area object local 1-axis.*

- **Geometry:** You specify the geometry of the slab and deck in this area. The following items are specified:
  - ✓ **Slab Depth (tc):** Depth of the slab, not including the height of the metal deck.
  - ✓ **Deck Depth (hr):** Depth (height) of the metal deck.
  - ✓ **Rib Width (wr):** Average width of the metal deck ribs.
  - ✓ **Rib Spacing (Sr):** Distance from the center of one down flute of the metal deck to the center of an adjacent down flute.
- **Composite Deck Studs:** You specify the design information for the composite beam shear studs in this area. The following items are specified:
  - ✓ **Diameter:** Diameter of the shear studs.
  - ✓ **Height (hs):** Height of the shear studs after welding.
  - ✓ **Tensile Strength, Fu:** Fu value for the shear studs.
- **Material:** Specifies the material property used for determining the deck shear stiffness (membrane stiffness) in this area. If the deck is filled or there is a solid slab, specify a slab material property. If the deck is unfilled, specify a deck material property and a deck shear thickness. The following items are specified in this area:
  - ✓ **Slab Material:** Name of the concrete material property associated with the slab.
  - ✓ **Deck Material:** Name of the steel material property associated with the deck. The mass and weight per unit volume specified for the steel material property (using the **Define menu > Material Properties** command) are not used for the deck (unless the specified mass per unit volume is zero and the deck is unfilled). See the upcoming Metal Deck Unit Weight bullet item for more information.

Note:

Deck section properties have membrane behavior only. No plate bending behavior is modeled for deck sections.

Note:

When you assign deck section properties, the program assumes that the deck spans in the same direction as the local 1-axis of the area object to which the deck is assigned

- ✓ **Deck Shear Thick:** Thickness of the deck used for calculating shear (membrane) stiffness when the deck type is unfilled deck.

- **Metal Deck Unit Weight:** This is the unit weight of the deck in force/length^2 units. This unit weight is included when determining the total self-weight of the floor system.

  When determining the self-mass of the floor system, the metal deck unit weight is converted to a unit mass. This unit mass is added to the unit mass specified for the material property designated by you as the Deck Material or Slab Material (depending on the deck type) in the Material area of the form.

  A special case does exist for this mass. If the deck is a filled deck and the mass per unit volume of the designated Slab Material is zero, the program assumes the mass of the metal deck is also zero. Similarly, if the deck is an unfilled deck and the mass per unit volume of the designated Deck Material is zero, the program assumes the mass of the metal deck is also zero.

- **Display Color:** Use this box to assign or change the color assigned to the deck section. If you use the **View menu > Set Building View Options** command to display the Set Building View Options form, you can choose an option to view the model based on the colors associated with the section properties. In that case, each object appears in a color associated with its assigned section property. See the section entitled "Set Building View Options Command" in Chapter 6 View Menu for more information.

The program has three built-in default area object properties. They are DECK1, SLAB1 and WALL1. These are, as the names suggest, metal deck, slab and wall properties. You can add additional properties as desired. You can also delete properties, including the built-in ones if they are not currently assigned to objects. However, the program does not let you delete the last deck, slab or wall property. In other words, you must always have at least one deck property, one slab property and one wall property defined, even if they are never assigned to anything.

Link Properties Command

The different types of link properties available in the program are as follows:

- Linear
- Damper
- Gap
- Hook
- Plastic1
- Isolator1
- Isolator2

Note:

In a linear analysis, the program converts the specified effective damping for link elements into equivalent modal damping and adds it to the specified modal damping.

Typically link elements can have two different sets of properties assigned to them: linear properties and nonlinear dynamic properties that are used for nonlinear dynamic (time history) analysis. Linear link elements can only have linear properties assigned to them. Note that you must have the nonlinear version of ETABS to use the nonlinear link properties.

The linear property that you specify for each of the six degrees of freedom of a linear link element is an effective stiffness. This is simply a spring stiffness.

The linear properties that you specify for each of the six degrees of freedom of all other types of link elements are an effective stiffness and effective damping. Again the effective stiffness is simply a spring stiffness. The effective damping specifies dashpot-type damping; it is *not* a specification of percent critical damping. In the advanced Nllink properties, the fraction of Moment to each end of the link to be transferred to the major and minor axis can be specified.

In a linear analysis, the program converts the specified effective damping for a link element to modal damping. It then adds the

modal damping calculated for all link elements in the model that have effective damping specified to any modal damping already specified for the structure as a whole to get the final modal damping. The program reports this final modal damping in the printed analysis output for building modes. To get this output click, the **File menu > Print Tables > Analysis Output** command and check the Building Modal Info check box.

Frame Nonlinear Hinge Properties Command

Nonlinear hinge properties are assigned to line objects with frame section property assignments for use in nonlinear static (pushover) analysis. The nonlinear hinge properties are defined using the **Define menu > Frame Nonlinear Hinge Properties** command. Note that you must have the nonlinear version of ETABS to utilize the frame nonlinear hinge properties. Description of the frame nonlinear hinge properties is beyond the scope of this manual.

Groups Command

Use the **Define Menu > Groups** command to define a new group name, change a group name or delete a group name. Use the **Assign menu > Group Names** command, described in Chapter 10 Assign Menu, to designate the objects to be assigned to the group name. Note that group names can be also be defined during the assignment process.

On the Define Group form, which comes up when you click the **Define Menu > Groups** command, the ALL group listed in the Groups area is a default group to which all objects in the model are automatically assigned. The ALL group name cannot be changed nor can the ALL group be deleted.

Use the **Add New Group**, **Change Group Name** and **Delete Group** buttons to add a new name, change a group name or delete a group name. The **Change Group Color** button can be used to change the color of objects assigned to a group.

Note that when the program is set to display colors by group, if an object is part of more than one group, it will be displayed using the color of the earliest defined group to which it is assigned. This can sometimes make it difficult to tell which objects are assigned to a particular group. To determine which objects are assigned to a group, first view the entire model. Then click the **Select menu > Select by Groups** command and select the desired group and click the **View menu > Show Selection Only** command to see only the objects that are part of the desired group.

Section Cuts Command

Note:

You can get resultant forces reported at any location for section cuts that you define through all or a portion of your structure.

Section cuts allow you to get resultant forces acting at section cuts through your structure. You can define section cuts before or after you run an analysis. Typically, you should not define section cuts, and more importantly the groups used in the section cut definition, until you have completed all manual meshing of your model. If you define these groups before manual meshing, some of the point objects that should be in the group may not yet be created. It is safest to wait until after you have run the analysis to define the section cuts.

Use the **Define menu > Section Cuts** command to define section cuts in this program. However, before you use this command, you first should define the group that to be used to specify the extent of the section cut. Groups are defined by selecting the objects that are to be part of the group and using the **Assign menu > Group Names** command, which is described in Chapter 10 Assign Menu.

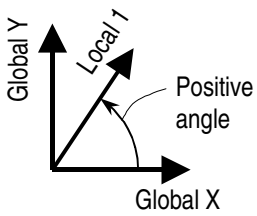
Section Cuts Form

Clicking the **Define menu > Section Cuts** command brings up the Section Cuts form. The Section Cuts area of this form lists the names of all of the currently defined section cuts. The Click To area of the form allows you to define new section cuts, modify existing section cut definitions and delete existing section cuts.

When you click the **Add Section Cut** button in the Section Cuts form or when you highlight an existing section cut name and click the **Modify/Show Section Cut** button, the Section Cut Data form appears. This form is broken into four areas that are described below.

- **Section cut name:** Specifies the name of a section cut; use this edit box to modify the name if desired.
- **Group:** Specifies the name of the group that defines the section cut; use this edit box to modify the group if desired.
- **Summation about this location:** Defines the location (point) about which section cut forces are summed. There are two options for specifying this point.
  - ✓ **Default:** By default, the section cut forces are reported at a location (point) that has coordinates equal to the average of the coordinates of all of the point objects included in the group that defines the section cut.
  - ✓ **User-Defined:** You can specify any arbitrary point that section cut forces are to be summed about by entering the global X, Y and Z coordinates of that point.

  Note that there does not need to be a Point object defined at the point that you are summing the section cut forces about.
- **Local 1-Axis Orientation:** By default the positive local 1, 2 and 3 axes of the section cut correspond to the global X, Y and Z axis respectively. You can rotate the local 1 and 2 axes about the 3-axis (Z-axis).



The direction of the positive local 1-axes is specified by an angle measured in degrees from the positive global X-axis. A positive angle appears counterclockwise as you look down on it from above. An angle of 0 degrees means the positive local 1-axis is in the same direction as the positive global X-axis. An angle of 90 degrees means the positive local 1-axis is in the same direction as the positive global Y-axis.

The angle described in the paragraph above is entered in the Local 1-Axis Orientation area of the Section Cut Data form. Any value between -360 degrees and +360 degrees, inclusive can be input.

7 Response Spectrum Functions Command

A response spectrum function is simply a list of period versus spectral acceleration values. In this program, the acceleration values in the function are assumed to be normalized; that is, the functions themselves are not assumed to have units. Instead, the units are associated with a scale factor that multiplies the function and is specified when you define the response spectrum case. See the subsection entitled "Input Response Spectra" under the section titled "Response Spectrum Cases" later in this chapter for more information.

Note:

In ETABS the acceleration values in a response spectrum function are assumed to be normalized, that is, the functions themselves are not assumed to have units.

Click the **Define menu > Response Spectrum Functions** command or this button to define response spectrum functions. When you execute this command, the Define Response Spectrum Functions form appears. The Response Spectra area of this form lists the names of all the currently defined response spectrum functions. The Click To area of the form allows you to add a new spectrum from a text file, add a new user-defined response spectrum function, add a new response spectrum function based on one of several ETABS built-in code response spectra, modify an existing response spectrum function definitions, and delete existing response spectrum function definitions.

Add Response Spectrum from File Form

You can add a response spectrum definition to the program from a text file. The text file should have period and spectral acceleration values. One set of values (period and spectral acceleration) should be provided on each line. Any line that has a $ symbol in the first character space is treated as a comment line and ignored. You can specify any number of header lines at the beginning of the file that the program should ignore. Those header lines do not

need $ symbols at the beginning of them. The program quits reading the file when it reaches the end of the file or when it reaches a blank line. Note that the program considers a line with the first character space blank, the second character space a $ symbol and anything beyond the $ symbol as a blank line.

Tip:

There are many code-specific response spectrum templates built into this program.

Click the **Add Spectrum from File** button in the Click To area of the Define Response Spectrum Functions form to add a new response spectrum function definition from an existing text file. This brings up the Response Spectrum Function Definition form. The following areas exist in that form:

- **Function name:** Specifies the name of the response spectrum function; use this edit box to modify the name if desired.

- **Function file:** Click on the **Browse** button in this area to bring the Pick Function Data File form where you indicate the name of the text file that includes your response spectrum data.

  Typically the program does not import the file into its database. It simply maintains a link to the file location. Thus, if you move the response spectrum file, or if you move your .edb file to another location, the program may suddenly be unable to locate the response spectrum file. If you click the **Convert to User-Defined** button, the program imports the response spectrum into its database file and the data will always be available to your model. Do not click the **Convert to User-Defined** button until you have specified the file name and indicated the number of header lines to skip.

  Note that when reading the function file, the program skips the number of lines at the top of the file indicated in the Header Lines to Skip item.

- **Value are:** The function can be Frequency vs. Value or Period vs. Value. No values appear in this area until you actually display the graph of the function.

- **Function graph:** This area displays a graph of the function. First specify the text file name and the number of header lines

to skip in the Function File area of the form. Then click the **Display Graph** button in the Function Graph area of the form to display the graph of the function. This also fills in the values in the Values are area of the graph.

You can run your mouse pointer over the function graph and a dot appears along the line representing the response spectrum. The coordinates of the dot are reported in the box just below the graph.

The program reads the response spectrum function file in the following way:

- First, it skips the specified number of header lines.
- Next, it checks to see if a line has a $ symbol as the first character. If it does, the program skips to the next line.
- If there is not a $ symbol as the first character on the line, the program reads the information on the line.
- If the line is blank or if the end of the file is reached, the program stops reading and closes the file.

User-Defined Response Spectrum Functions

Note:

Response spectra in this program are always defined as period versus spectral acceleration.

Click the drop down box just below the **Add Spectrum from File** button in the Click To area of the Define Response Spectrum Functions form and click on Add User Spectrum to add a new user-defined response spectrum. This brings up the Response Spectrum Function Definition form. The following areas exist in this form:

- **Function name:** Specifies the name of the response spectrum function; use this edit box to modify the name if desired.
- **Define Function:** Input the period and spectral acceleration values for the function in this area. Type the first set of period and spectral acceleration values into the edit boxes at the top of this area. Then click the **Add** button. Type in the next set of

period and spectral acceleration values and again click the **Add** button. Continue this process until all sets of values have been entered.

If you want to modify an existing set of values, first highlight the appropriate values in the list box. Note that when you highlight them, they appear in the edit boxes at the top of the area. Modify the values in the edit boxes and then click the **Modify** button.

If you want to delete an existing set of values, first highlight the appropriate values in the list box. Note that when you highlight them, they appear in the edit boxes at the top of the area. Then click the **Delete** button.

- **Function graph:** This area displays a graph of the function. It updates automatically as additional points are defined for the function. If your computer has any problem with the automatic update, click the **Display Graph** button located just below the graph.

  You can run your mouse pointer over the function graph and a dot appears along the line representing the response spectrum. The coordinates of the dot are reported in the box just below the graph.

Code-Specific Response Spectrum Functions

The program allows you to easily define code-specific response spectrum functions for a variety of building codes.

Click the drop-down box just below the **Add Spectrum from File** button in the Click To area of the Define Response Spectrum Functions form and click on one of the code-specific items. For example, click on Add UBC97 Spectrum to add a new response spectrum based on the 1997 UBC.

Clicking on one of the code-specific items brings up a code-specific Response Spectrum Function Definition form. The following areas exist in this form:

- **Function name:** Specifies the name of the response spectrum function; use this edit box to modify the name if desired.

- **Parameters:** You specify the parameters that define the code-specific response spectrum in this area. These parameters vary from code to code. The parameters specified for each of the codes included in this program are described in subsequent subsections.

- **Define Function:** This area displays the period and spectral acceleration values for the function. You can only view the values in this area. You cannot edit these values unless you convert the function to a user-defined function. The values shown here update every time you redefine the spectrum parameters.

  Note that you can click the **Convert to User-Defined** button at any time to convert the function to a user-defined function. Then you are able to edit values in the Define Function area.

- **Function graph:** This area displays a graph of the function. It updates automatically as you redefine the spectrum parameters. If your computer has any problem with the automatic update, click the **Display Graph** button located just below the graph.

  You can run your mouse pointer over the function graph and a dot appears along the line representing the response spectrum. The coordinates of the dot are reported in the box just below the graph.

1994 UBC Parameters for a Response Spectrum Function

The 1994 UBC response spectrum function is based on Figure 16-3 in Chapter 16 of the 1994 UBC. The digitization of these response spectra are based on Section C106.2.1 in the 1996 SEAOC Recommended Lateral Force Requirements and Commentary (more commonly called the SEAOC Blue Book).

The parameters you enter are a seismic zone factor, Z and a soil type. Any positive, nonzero value can be specified for the seismic zone factor; see Table 16-I in the 1994 UBC for typical values. The soil type can be input as 1, 2 or 3; see Table 16-J in the 1994 UBC for typical values.

1997 UBC Parameters for a Response Spectrum Function

The 1997 UBC response spectrum function is constructed as shown in Figure 16-3 in Chapter 16 of the 1997 UBC. The parameters you enter are seismic coefficients C_a and C_v. Any positive, nonzero value can be specified for the seismic coefficients. See Tables 16-Q and 16-R in the 1997 UBC for typical values of these coefficients.

1996 BOCA Parameters for a Response Spectrum Function

The following parameters are input for the 1996 BOCA response spectrum function. Any positive, nonzero value can be input for these parameters.

A_a = Seismic coefficient representing the effective peak acceleration as determined in 1996 BOCA Section 1610.1.3.

A_v = Seismic coefficient representing the effective peak velocity-related acceleration as determined in 1996 BOCA Section 1610.1.3.

R = The response modification factor determined from 1996 BOCA Table 1610.3.3.

S = The coefficient for the soil profile characteristics of the site as determined by 1996 BOCA Table 1610.3.1.

The 1996 BOCA response spectrum function is based on 1996 BOCA Section 1610.5.5. The response spectrum is constructed by plotting the modal seismic design coefficient, C_{sm}, versus the modal period of vibration, T_m. For a given period, T_m, the value of C_{sm} is determined using Equation 7-3.

$$C_{sm} = \frac{1.2A_vS}{RT_m^{2/3}} \leq \frac{2.5A_a}{R} \qquad \text{Eqn. 7-3}$$

1995 NBCC Parameters for a Response Spectrum Function

The following parameters are input for the 1995 NBCC (Canadian) response spectrum function.

v = Zonal velocity ratio.

Z_a = Acceleration-related seismic zone.

Z_a = Velocity-related seismic zone.

Values for these parameters can be found in Appendix C of the 1995 NBCC. Any positive, nonzero value can be input for the zonal velocity ratio, v. Any positive integer, or zero, can be input for the acceleration and velocity-related seismic zones.

The 1995 NBCC response spectrum function is based on item 44(a) in Commentary J of the 1995 NBCC.

IBC2000 Parameters for a Response Spectrum Function

The following parameters are input for the IBC2000 response spectrum function. Any positive, nonzero value can be input for these parameters.

S_{DS} = The 5% damped design spectral response acceleration at short periods as specified in IBC2000 Section 1613.2.1.3.

S_{D1} = The 5% damped design spectral response acceleration at a one second period as specified in IBC2000 Section 1613.2.1.3.

The IBC2000 response spectrum function is based on the procedure described in IBC2000 Section 1613.2.1.4.

1997 NEHRP Parameters for a Response Spectrum Function

The following parameters are input for the 1997 NEHRP response spectrum function. Any positive, nonzero value can be input for these parameters.

S_{DS} = The design earthquake spectral response acceleration at short periods as specified in 1997 NEHRP Equation 4.1.2.5-1.

S_{D1} = The design earthquake spectral response acceleration at a one second period as specified in 1997 NEHRP Equation 4.1.2.5-2.

The 1997 NEHRP response spectrum function is based on the procedure described in 1997 NEHRP Section 4.1.2.6.

1998 Eurocode 8 Parameters for a Response Spectrum Function

The 1998 Eurocode 8 response spectrum function is constructed as described in 1998 Eurocode ENV 1998-1-1:1994 Section 4.2.2. The parameters you enter are the design ground acceleration, a_g, the subsoil class and the damping correction factor, η. Any positive, nonzero value can be specified for the design ground acceleration. The damping correction factor must satisfy $\eta \geq 0.7$. The subsoil class can be input as A, B or C.

The ordinates of the response spectrum are calculated using Equations 4.1 through 4.4 in 1998 Eurocode ENV 1998-1-1:1994 Section 4.2.2. The values of β_o, T_B, T_C, T_D, k_1, k_2 and S are taken from Table 4.1 in 1998 Eurocode ENV 1998-1-1:1994 Section 4.2.2. Note that the value of these items depends on the specified subsoil class.

1992 NZS 4203 Parameters for a Response Spectrum Function

For the 1992 NZS4203 (New Zealand) response spectrum function you input a scaling factor and a site subsoil category. Any

positive, nonzero value can be specified for the scaling factor. The site subsoil category can be input as A, B or C.

The 1992 NZS4203 (New Zealand) response spectrum function is constructed as specified in 1992 NZS4203 Section 4.6.

The ordinates of the response spectrum are calculated using 1992 NZS4203 Equations 4.6.3 and 4.6.4. If you are using Equation 4.6.3 then you input the scaling factor as $S_p * R * Z * L_s$. If you are using Equation 4.6.4 then you input the scaling factor as $S_m * S_p * R * Z * L_u$.

ETABS calculates the $C_h(T, 1)$ term in Equations 4.6.3 and 4.6.4 based on the input site subsoil category and the values for $\mu=1.0$ in Figures 4.6.1a, b and c and in Tables 4.6.1a, b and c. In Table 4.6.1a the coefficient values for periods of 0, 0.09 and 0.20 seconds are taken as 0.40, 0.68 and 0.68, respectively. In Table 4.6.1b the coefficient values for periods of 0, 0.13 and 0.20 seconds are taken as 0.42, 0.80 and 0.80, respectively. In Table 4.6.1c the coefficient values for periods of 0 and 0.10 seconds are taken as 0.42 and 0.72, respectively.

Modifying and Deleting Response Spectrum Functions

In the Define Response Spectrum Functions form, highlight an existing response spectrum name and then click on the **Modify/Show Spectrum** button to modify the spectrum. The same form that appeared when you defined the function appears and you can make any changes or modifications that you desire.

To delete an existing response spectrum function, highlight its name in the Define Response Spectrum Functions form and click the **Delete Spectrum** button.

Time History Functions Command

A time history function may be a list of time and function values or just a list of function values that are assumed to occur at equally spaced intervals. The function values in a time history

function may be ground acceleration values or they may be multipliers for specified (force or displacement) load cases.

Tip:

Define time history functions using one of several built-in time history function templates.

Click the **Define menu > Time History Functions** command or this button to define time history functions. When you execute this command or click the button, the Define Time History Functions form appears. The Functions area of this form lists the names of all the currently defined time history functions. The Click To area of the form allows you to Add Function from (text) File, Add User Function (a user-defined time history function), add a new time history function based on one of several built-in function templates (Sine, Cosine, Ramp, Sawtooth, Triangle), Modify/Show Function to modify an existing time history function definition and Delete Function to delete existing time history function definitions.

Add Functions from File Button

You can add a time history definition to the program from a text file. Any line that has a $ symbol in the first character space is treated as a comment line and ignored. You can specify any number of header lines at the beginning of the file that the program should ignore. Those header lines do not need $ symbols at the beginning of them. The program quits reading the file when it reaches the end of the file or when it reaches a blank line. Note that the program considers a line with the first character space blank, the second character space a $ symbol and anything beyond the $ symbol as a blank line.

Click the **Add Function from File** button in the Click To area of the Define Time History Functions form to add a new time history function definition from an existing text file. This brings up the Time History Function Definition form. The following areas exist in this form:

- **Function name:** Specify the name of the time history function; use this edit box to modify the name if desired.

- **Function file:** Click on the **Browse** button in this area to bring up the Pick Function Data File form where you select the name of the text file that includes your time history function data.

  Typically, the program does not import the file into its database. It simply maintains a link to the file location. Thus, if you move the time history function file, or if you move your .edb file to another location, the program may be unable to locate the function file. If you click the **Convert to User-Defined** button, the program imports the time history into its database file and the data will always be available to your model. Do not click the **Convert to User-Defined** button until you have specified all information in the Function file, Values are and Format type areas.

  Note that when reading the function file, the program skips the number of lines at the top of the file indicated in the Header Lines to Skip item. It also skips the number of characters specified in the Prefix Characters per Line item at the beginning of each line.

  The Number of Points per Line item tells the program how many function values or sets of time and function values are specified on each line.

- **Values are:** Specifies whether the text file contains time and function values or function values that are spaced at equal time intervals. If the file contains function values that are spaced at equal time intervals, also specify the time interval.

- **Format type:** The format type can be specified as either free format or fixed format. In free format, items on the lines can be separated by spaces or tabs. If you specify a fixed format type, you also specify the number of characters per item. Each item on a line is assigned the same number of character spaces. The program begins counting the spaces after it skips the number of prefix characters specified in the Function File area.

- **Function graph:** This area displays a graph of the function. First specify all of the data in the Function file, Values are and

Format type areas. Then click the **Display Graph** button in the Function Graph area of the form to display the graph of the function.

You can run your mouse pointer over the function graph and a dot appears along the line representing the time history function. The coordinates of the dot are reported in the box just below the graph.

The program reads the function file in the following way:

- First, the program skips the specified number of header lines.
- Next, the program checks to see if a line has a $ symbol as the first character. If it does, the program skips to the next line.
- If there is not a $ symbol as the first character on the line, the program reads the information on the line, skipping the specified number of characters at the beginning of the line.
- If the line is blank or if the end of the file is reached, the program stops reading and closes the file.

Add User Function

Click the drop-down box just below the **Add Function from File** button in the Click To area of the Define Time History Functions form and click on Add User Function to add a new user-defined history function. This brings up the Time History Function Definition form. The following areas exist in this form:

- **Function name:** Specifies the name of the time history function; use this edit box to modify the name if desired.
- **Define Function:** Input the time and associated function value for the function in this area. Type the first set of time and function values in the edit boxes at the top of this area. Then click the **Add** button. Type in the next set of time and function values and again click the **Add** button. Continue this process until all sets of values have been entered.

If you want to modify a set of values after you have entered them, first highlight the appropriate values in the list box. Note that when you highlight them, they appear in the edit boxes at the top of the area. Modify the values in the edit boxes and then click the **Modify** button.

If you want to delete a set of values after you have entered them, first highlight the appropriate values in the list box. Note that when you highlight them, they appear in the edit boxes at the top of the area. Then click the **Delete** button.

- **Function graph:** This area displays a graph of the function. It updates automatically as additional points are defined for the function. If your computer has any problem with the automatic update, click the **Display Graph** button located just below the graph.

You can run your mouse pointer over the function graph and a dot will appear along the line representing the time history function. The coordinates of the dot are reported in the box just below the graph.

Add (Template) Function - Sine, Cosine, Ramp, Sawtooth and Triangular

The program allows you to easily define sine, cosine, ramp, sawtooth and triangular time history functions using built-in program time history function templates.

Click the drop down box just below the **Add Function from File** button in the Click To area of the Define Time History Functions form and click on one of the sine, cosine, ramp, sawtooth and triangular items. For example, click on Add Sine Function to add a new time history function based on a sine function.

Clicking on one of these time history function items brings up a the Time History (template name) Function Definition form. The

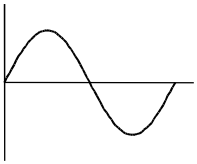
forms for all templates are the same except for the name in the title of the form. Those areas include the following:

- **Function name:** Specifies the name of the time history function; use this edit box to modify the name if desired.
- **Parameters:** Specify the parameters that define the template time history function in this area. Those parameters vary for different templates. The parameters specified for each of the templates are described in subsequent subsections.
- **Define Function:** This area displays the time and function values for the time history function. You can only view the values in this area. You cannot edit the values unless you convert the function to a user-defined function. The values shown here update every time you redefine the template parameters.

  Note that you can click the **Convert to User-Defined** button at any time to convert the function to a user-defined function. Converting the template to a user-defined function would allow you to edit values in the Define Function area.
- **Function graph:** This area displays a graph of the function. It updates automatically as you redefine the template parameters. If your computer has any problem with the automatic update, click the **Display Graph** button located just below the graph.

  You can run your mouse pointer over the function graph and a dot appears along the line representing the time history function. The coordinates of the dot are reported in the box just below the graph.

Sine Time History Function Template Parameters



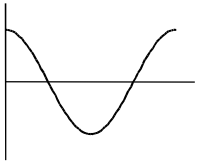
The sine time history function is a periodic function. A sine function cycle starts at a function value of 0, proceeds to its positive maximum value (positive value of amplitude), continues to a value of 0, progresses to its negative minimum value (negative value of amplitude), and returns to a value of 0 again. The

following parameters are specified in the sine time history function template.

- **Period:** The time in seconds that it takes for the sine function to complete one cycle.
- **Number of Steps per Cycle:** The number of steps (i.e., function value points) provided for each cycle of the function.
- **Number of Cycles:** The number of cycles in the function.
- **Amplitude:** The maximum function value of the sine function.

Cosine Time History Function Template Parameters

The cosine time history function is a periodic function. A cosine function cycle starts at its positive maximum value (positive value of amplitude), proceeds to a value of 0, continues to its negative minimum value (negative value of amplitude), and returns to a value of 0 again and finally returns to it positive maximum value again. The following parameters are specified for the cosine time history function template.



- **Period:** The time in seconds that it takes for the cosine function to complete one cycle..
- **Number of Steps per Cycle:** The number of steps (i.e., function value points) provided for each cycle of the function.
- **Number of Cycles:** The number of cycles in the function.
- **Amplitude:** The maximum function value of the cosine function.

Ramp Time History Function Template Parameters

A ramp function is defined by three (time, function value) points. Those three points, in order, are (0, 0), (Ramp time, Amplitude) and (Maximum time, Amplitude). The ramp time, amplitude and maximum time parameters are described as follows.

- **Ramp Time:** Time that it takes for the ramp function to initially reach its maximum value. It is usually set to one second.
- **Amplitude:** The maximum value of the ramp function. It is usually set to 1.
- **Maximum Time:** The time at the end of the ramp function. It is usually between 10 and 20 seconds.

Sawtooth Time History Function Template Parameters

The sawtooth time history function is a periodic function. A single cycle of a sawtooth function is defined by seven (time, function value) points. Those seven points, in order, are (0, 0), (Ramp time, Amplitude), (0.5 * Period - Ramp time, Amplitude), (0.5 * Period, 0), (0.5 * Period + Ramp time, -Amplitude), (Period - Ramp time, -Amplitude) and (Period, 0). Figure 7-4 illustrates a single cycle of a sawtooth time history function and the seven points.

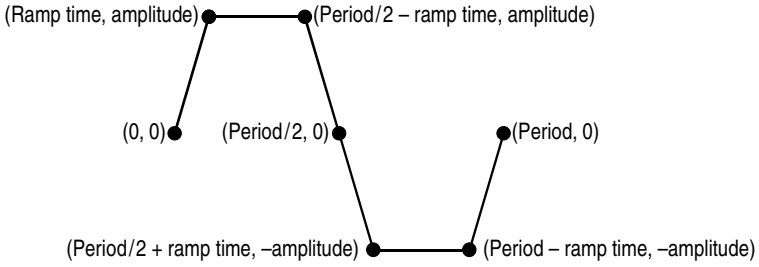


Figure 7-4: *Points that define a cycle of a sawtooth time history function*

The following parameters are specified in the sawtooth time history function template.

- **Period:** The time in seconds that it takes for the sawtooth function to complete one cycle.
- **Ramp Time:** The time that it takes for the sawtooth function to ramp up from a function value of 0 to its maximum amplitude.

- **Number of Cycles:** The number of cycles in the function.
- **Amplitude:** The maximum function value of the sawtooth function.

Triangular Time History Function Template Parameters

The triangular time history function is a periodic function. A single cycle of a triangular function is defined by five (time, function value) points. Those five points, in order, are (0, 0), , (0.25 * Period, Amplitude), (0.5 * Period, 0), (0.75 * Period, -Amplitude) and (Period, 0). The following parameters are specified in the Triangular time history function template.

- **Period:** The time in seconds that it takes for the triangular function to complete one cycle.
- **Number of Cycles:** The number of cycles in the function.
- **Amplitude:** The maximum function value of the triangular function.

Add User Periodic Function

Click the drop down box just below the **Add Function from File** button in the Click To area of the Define Time History Functions form and click on Add User Periodic Function to add a new user-defined periodic time history function. This brings up the Time History User Periodic Function Definition form. The following areas exist in this form:

- **Function name:** Specifies the name of the time history function; use this edit box to modify the name if desired.
- **Number of Cycles:** This is the number of cycles in the function.
- **Define Function:** Input the time and associated function value for the function in this area. Type the time and function values

in the edit boxes at the top of this area. Then click the **Add** button.

NOTE: For a user periodic function, specify the time and function values for one cycle of the function and specify the number of cycles. When the program uses the function, it continues for the specified number of cycles, even though you only specified values for the first cycle.

If you want to modify the set of values you have entered, first highlight the appropriate values in the list box. Note that when you highlight them they appear in the edit boxes at the top of the area. Modify the values in the edit boxes and then click the **Modify** button.

If you want to delete the set of values you have entered, first highlight the appropriate values in the list box. Note that when you highlight them they appear in the edit boxes at the top of the area. Then click the **Delete** button.

- **Function graph:** This area displays a graph of the function. It updates automatically as additional points are defined for the function. If your computer has any problem with the automatic update then simply click the **Display Graph** button located just below the graph.

  You can run your mouse pointer over the function graph and a dot appears along the line representing the time history function. The coordinates of the dot are reported in the box just below the graph.

If you convert a user periodic function to a user function, values are shown for all of the specified cycles.

Static Load Cases Command

In this program, you first define static load cases and then you assign various types of loads to the static load cases using commands available on the Assign menu, which is described in

Chapter 10. There is no limit on the number of static load cases that you can define.

As you will see in the ensuing description, you actually can assign three types of loads to static load cases as you define them. These three types of loads are self-weight, automatic static earthquake loads and automatic static wind loads.

Click the **Define menu > Static Load Cases** command or the button to define static load cases. This brings up the Define Static Load Case Names form. The form has two areas: Loads and Click to.

The Loads area has four separate items:

- **Load:** This is the name of the static load case.
- **Type:** This is the type of the static load case. The program uses these values when automatically creating design load combinations for the design postprocessors. The factors used in the design load combinations are different for the various types of loads. The choices for load types are:
  - ✓ **Dead:** Dead load.
  - ✓ **Super Dead:** Superimposed dead load. This is used in the Composite Beam design postprocessor.
  - ✓ **Live:** Live load.
  - ✓ **Reduce Live:** Reducible live load. A live load that is specified as reducible is reduced automatically by the program for use in the design postprocessors. The live load reduction parameters are specified using the **Options menu > Preferences > Live Load Reduction** command. See the subsection entitled "Preference > Live Load Reduction Command" in Chapter 14 Options Menu for more information.
  - ✓ **Quake:** Earthquake load.

- ✓ **Wind:** Wind load.
- ✓ **Snow:** Snow load.
- ✓ **Other:** Other load that does not fit into one of the above categories or that you do not want included in the design load combinations automatically created by the program.

- **Self-weight multiplier:** The self-weight of the structure is determined by multiplying the weight per unit volume of each object that has structural properties times the volume of the object. The weight per unit volume is specified in the material properties.

  You can specify that a portion of the self-weight be applied to any static load case. The self-weight multiplier controls what portion of the self-weight is included in a load case. A self-weight multiplier of 1 means to include the full self-weight of the structure in the load case. A self-weight multiplier of 0.5 means to include half of the self-weight of the structure in the load case.

  Normally you should specify a self-weight multiplier of 1 in one static load case only, usually your dead load load case. All other static load cases then have self-weight multipliers of zero. Note that if you include a self-weight multiplier of 1 in two different load cases, and then combine those two load cases in a load combination, the results for the load combination are based on an analysis where double the self-weight of the building has been applied as a load.

- **Auto Lateral Load:** The Auto Lateral Load item is inactive unless the load type specified is Quake or Wind. When you specify the load type as Quake or Wind, the Auto Lateral Load drop-down box becomes active and you can choose None or one of several different code-specified loads that are then automatically created for the specified load case.

  If you do not want to use the automatic lateral loads and instead plan to assign your own loads using the commands avail-

able on the Assign menu, choose None from this drop-down box. Otherwise select the automatic load that you want to create from the drop-down box. Initially default values are used for the automatic lateral load. If you want to review and/or modify those values, click the **Modify Lateral Load** button.

Define Static Load Case Names Form

Use the following procedure to add a new static load case in the Define Static Load Case Names form:

- Type the name of the load case in the Load edit box. The program does not allow you to use duplicate names.
- Select a load type from the Type drop-down box.
- Type a self-weight multiplier in the Self-Weight Multiplier edit box (see **Self-weight multiplier** bullet in the previous section for suggested values).
- If the load type specified is Quake or Wind, select an option from the Auto Lateral Load drop-down box.
- Click the **Add New Load** button.

NOTE: If your selected an automatic lateral load in the Auto Lateral Load drop-down box, click the **Show Lateral Loads** button and review or modify the parameters for the automatic lateral load in the resulting form. Then click the **OK** button to return to the Define Static Load Case Names form.

Modify an Existing Static Load Case

Use the following procedure to modify an existing static load case in the Define Static Load Case Names form:

- Highlight the the existing load case in the Loads area of the form. Note that the data associated with that load case appears in the edit and drop-down boxes at the top of the Loads area.
- Modify any of the data in the Loads area for the load case.

- Click the **Modify Load** button. If necessary, click the **Show Lateral Loads** button to modify the automatic lateral load parameters.

Delete an Existing Static Load Case

Use the following procedure to delete an existing static load case in the Define Static Load Case Names form. Note that when you delete a static load case here, all of the loads that have been assigned to the model as a part of that static load case are also deleted.

- Highlight the existing load case in the Loads area of the form. Note that the data associated with that load case appears in the edit and drop-down boxes at the top of the Loads area.
- Click the **Delete Load** button.

Response Spectrum Cases Command

Click the **Define menu > Response Spectrum Cases** command or the button to define a response spectrum case. This command brings up the Define Response Spectra form. *Note that you must have at least one response spectrum function defined for this command to be active.*

The Spectra area of the Define Response Spectrum form lists the names of all the currently defined response spectrum cases. The Click To area of the form allows you to define new response spectrum cases, modify existing response spectrum cases and delete existing response spectrum cases.

Response Spectrum Case Data Form

Clicking on the **Add New Spectrum** button or highlighting an existing spectrum and clicking the **Modify/Show Spectrum** button brings up the Response Spectrum Case Data form. The following subsections describe each of the areas in this form.

Spectrum Case Name

Here you can specify or modify the name of the response spectrum case.

Structural and Function Damping

It is important that you understand the structural and function damping item. This item specifies modal damping that is present for all modes in the response spectrum analysis. Also, the program assumes that the response spectrum functions specified for the response spectrum case are all specified for this particular damping ratio.

For example, if you specify 2% damping for this term, you are specifying that there is 2% modal damping in all modes for the response spectrum analysis **and you are also telling the program that the response spectrum functions specified for this response spectrum case are for 2% damping.**

If you have link elements defined in your model and damping is specified in the linear properties of the link element, the actual damping for a mode may be larger than that specified in the structural and function damping term because the program converts the damping for the links into modal damping and adds that modal damping to the specified modal damping to get the final total modal damping. See the previous section in this chapter entitled "Link Properties Command" for additional information on total modal damping.

In the cases where the final modal damping is different from the damping specified in the structural and function damping edit box (larger than because of added damping from link elements), the program modifies the input response spectrum to match this larger damping. The damping modification is based on the 50% median values for velocity in Table 2 of N. M. Newark and W. J. Hall (1981).

For all response spectra, the program reduces the entire spectrum based on the velocity formula (2.31 - 0.41 $\ln\beta$) in Table 2 of N.

M. Newark and W. J. Hall (1981). A maximum reduction of 50% is made.

For example, suppose that a response spectrum is specified as a 4% damped response spectrum and the actual final damping for a mode is 6.3% (because of added link elements). The program then modifies the specified 4% damped spectrum by the factor determined in Equation 7-7.

$$\frac{2.31 - 0.41 \ln 6.3}{2.31 - 0.41 \ln 4} = \frac{1.555}{1.742} = 0.89 \qquad \text{Eqn. 7-7}$$

Thus the spectral ordinate at the modal period in the 4% damped response spectrum is multiplied by a factor of 0.89 to obtain the spectral ordinate for 6.3% damping, which is the actual final damping associated with the mode.

Note that unlike time history analysis, for response spectrum analysis, you cannot override the modal damping specified for all modes on a mode-by-mode basis.

Modal Combination

In this area, you specify the method that the program uses to combine modal responses in the response spectrum analysis. Here you also define a damping value.

Note:

The program defaults to the CQC method of modal combination.

The following options are available for modal combinations:

- **CQC:** This is the Complete Quadratic Combination method described by E. L. Wilson, A. D. Kiureghian and E. Bayo (1981). This modal combination technique takes into account the statistical coupling between closely spaced modes caused by modal damping. Increasing the modal damping increases the coupling between closely spaced modes. If the modal damping is 0 for all modes, the CQC method degenerates to the SRSS method.

- **SRSS:** This is the Square Root of the Sum of the Squares method. This modal combination technique does not take into

account any coupling of modes as do the CQC and GMC methods.

- **ABS:** This is the Absolute method. This modal combination technique simply combines the modal results by taking the sum of their absolute values. This method is usually over-conservative.

- **GMC:** This is the General Modal Combination method that is also known as the Gupta method. This method is the same as the complete modal combination procedure described in Equation 3.31 in A. K. Gupta (1990). The GMC method takes into account the statistical coupling between closely spaced modes similar to the CQC method, and it also includes the correlation between modes with rigid-response content.

  The GMC method requires that you specify two frequencies, f_1 and f_2, that define the rigid-response content of the ground motion. These must satisfy $0 < f_1 < f_2$. The rigid-response parts of all modes are assumed to be perfectly correlated.

  The GMC method assumes no rigid response below frequency f_1, full rigid response above frequency f_2 and an interpolated amount of rigid response for frequencies between f_1 and f_2.

  Frequencies f_1 and f_2 are properties of the seismic input, not of the structure. Gupta defines f_1 as shown in Equation 7-4.

$$f_1 = \frac{S_{Amax}}{2\pi S_{Vmax}} \qquad \text{Eqn. 7-4}$$

  where S_{Amax} is the maximum spectral acceleration and S_{Vmax} is the maximum spectral velocity for the ground motion considered. The default value for f_1 is unity.

  Gupta defines f_2 as shown in Equation 7-5.

$$f_2 = \frac{1}{3} f_1 + \frac{2}{3} f_r \qquad \text{Eqn. 7-5}$$

where f_r is the rigid frequency of the seismic input; that is, the frequency above which the spectral acceleration is essentially constant and equal to the value at zero period (infinite frequency). Others have defined f_2 as shown in Equation 7-6.

$$f_2 = f_r \qquad \text{Eqn. 7-6}$$

The default value for f_r is zero, indicating infinite frequency. For the default value of f_2, the GMC method gives results similar to the CQC method.

Directional Combination

For each displacement, force or stress quantity in the structure, modal combination produces a single, positive result for each direction of acceleration. These directional values for a given response quantity are combined to produce a single positive result. The two available choices for directional combination are:

- **SRSS:** Combines the directional results by taking the square root of the sum of their squares. All other input items remaining unchanged, the results obtained using this method do not vary, regardless of the excitation angle that you specify. This is the recommended method for directional combination and is the default.

- **ABS:** This is the scaled absolute sum method. Here the directional results are combined by taking the maximum, over all directions, of the sum of the absolute values of the response in one direction plus a scale factor times the response in the other directions.

  For example, if the scale factor equals 0.3, the spectral response, R, for a given displacement, force or stress would be:

  $$R = \max\left(\overline{R}_1, \overline{R}_2, \overline{R}_3\right)$$

  where,

  $$\overline{R}_1 = R_1 + 0.3\left(R_2 + R_3\right)$$

$$\overline{R}_2 = R_2 + 0.3\left(R_1 + R_3\right)$$

$$\overline{R}_3 = R_3 + 0.3\left(R_1 + R_2\right)$$

and R_1, R_2 and R_3 are the modal combination values for each direction.

All other input items remaining unchanged, the results obtained using this method will vary depending on the excitation angle you choose. Results using a scale factor of 0.3 are comparable to the SRSS method (for equal input spectra in each direction) but may be as much as 8% unconservative or 4% overconservative, depending on the excitation angle chosen. Larger scale factors tend to produce more conservative results.

Input Response Spectra

Here you can specify any defined response spectrum function for each of the three local coordinate system directions of the response spectrum case as defined by the excitation angle. See the subsection below for discussion of the excitation angle. You can also specify a scale factor along with each function.

Note that this scale factor has units of Length/seconds^2 and that its value will change as you change the units in your model. Essentially the program assumes the response spectrum functions are unitless (normalized) and that the scale factor converts them into the appropriate units.

If you are scaling your response spectrum to match some static analysis results (e.g., base shear) you may want to include that in the scale factor specified for the response spectrum function in the input response spectra area. In this case you would input a scale factor equal to the product of the scale factor to covert the spectrum to the appropriate units and the scale factor to scale the response spectrum base shear to the appropriate level.

Excitation Angle

The response spectrum case positive local 3-axis is always in the same direction as the positive global Z-axis. The response spectrum case local 1 and 2 axes lie in the horizontal global XY plane.

The excitation angle is an angle measured from the positive global X-axis to the response spectrum case positive local 1-axis. A positive angle appears counterclockwise as you look down on the model.

Thus the direction of the response spectrum local 1-axis is determined by the excitation angle, the local 3-axis is in the same direction as the Z-axis, and the local 2-axis is determined from the local 1 and 3 axes using the right hand rule.

Time History Cases Command

Click the **Define menu > Time History Cases** command or the button to define a time history case. This command brings up the Define Time History Cases form. *Note that you must have at least one time history function defined for this command to be active.*

The History area of the Define Time History Cases form lists the names of all the currently defined time history cases. The Click To area of the form allows you to define new time history cases, modify existing time history cases and delete existing time history cases.

Time History Case Data Form

Clicking on the **Add New History** button or highlighting an existing time history and clicking the **Modify/Show History** button brings up the Time History Case Data form. The following subsections describe each of the areas in this form:

History Case Name

Specifies the name of the time history case. Use this edit box to modify the name if desired.

Options

This area of the form has the following options that you specify for the time history analysis:

- **Analysis type:** Specifies the type of time history analysis. The possible choices are linear, periodic and nonlinear.

  - ✓ **Linear:** In a linear time history analysis, all objects behave linearly. Only the linear properties assigned to link elements are considered in a linear time history analysis. Frame nonlinear (pushover) hinges assigned using the **Define menu > Frame Nonlinear Hinge Properties** command have no effect on a linear time history analysis.

  - ✓ **Periodic:** A periodic time history analysis is a linear analysis. For this analysis, specify a single cycle of the periodic function, and then the program assumes that the specified cycle continues indefinitely.

    The program shows time history results for a single cycle that are generated after the output has stabilized such that the conditions at the beginning of the cycle are equal to those at the end of the cycle.

    In a periodic time history analysis, all objects behave linearly. Only the linear properties assigned to link elements are considered in a periodic time history analysis. Frame nonlinear (pushover) hinges assigned using the **Define menu > Frame Nonlinear Hinge Properties** command have no effect on a periodic time history analysis.

  - ✓ **Nonlinear:** In a nonlinear time history analysis, the nonlinear dynamic properties assigned to link elements are considered. The mode shapes obtained for the analysis are based on linear properties only. Frame nonlinear (push-

over) hinges assigned using the **Define menu > Frame Nonlinear Hinge Properties** command have no effect on a nonlinear time history analysis.

- **Modal damping:** Click the **Modify/Show** button adjacent to the Modal Damping item to bring up the Modal Damping form where you can specify or modify the modal damping. In this form you can specify a damping that applies to all modes and then, if desired, overwrite the damping for any mode(s). The three bullet items that follow describe the three areas of the Modal Damping form.

  - ✓ **Damping for All Modes:** Enter the damping for all modes in this form. This is a percent critical damping. A damping that is 5% of critical damping is entered as 0.05.

Note:

In this program, enter 5% of critical damping as 0.05.

  - ✓ **Damping Override Options:** You can choose one of two options in this area. If you choose the Specify Modal Damping Overrides option, the Modal Damping Overrides area becomes active and you can specify damping overrides for any mode(s). Use the overrides when modal damping for some modes is different from the damping that is specified for all modes.

    If you choose the No Damping Overrides/Delete Overrides option, the Modal Damping Overrides area becomes inactive and any damping overrides that were specified are deleted.

  - ✓ **Modal Damping Overrides:** This area is only active if the Specify Modal Damping Overrides option has been selected in the Damping Override Options area. To override a modal damping value type, select the mode number in the Mode box and then type the damping value in the damping box. (A damping that is 5% of critical damping is entered as 0.05). Then click the **Add** button.

    To modify an existing modal damping value, highlight the existing damping value in the Modal Damping Overrides area of the form. Note that the data associated with that

load case appears in the edit and drop-down boxes at the top of this area. Modify the mode number or damping as desired. Then click the **Modify** button.

To delete an existing modal damping value, highlight the existing damping value in the Modal Damping Overrides area of the form. Note that the data associated with that load case appears in the edit and drop-down boxes at the top of this area. Then click the **Delete** button.

Important note: Do not forget to specify the number of modes to be used for your analysis. To specify the number of modes, click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options form. Make sure that the Dynamic Analysis check box is checked and click the **Set Dynamic Parameters** button to bring up the Dynamic Analysis parameters form where you can specify the number of modes that are used in the analysis.

You can specify a modal damping override for any mode number. If you specify a damping override for a mode that is larger than the number of modes specified for the analysis, this overly large damping override is simply ignored when the analysis is run.

Note:

If you start a time history from the final conditions of a previous history, both histories must be of the same type. That is, they must both be linear or they must both be nonlinear.

- **Number of Output Time Steps:** This is the number of equally spaced steps at which the output results are reported. Do not confuse this with the number of time steps in your input time history function. The number of output time steps can be different from the number of time steps in your input time history function. The number of output time steps multiplied by the output time step size is equal to the length of time over which output results are reported.

- **Output Time Step Size:** This is the time in seconds between each of the equally spaced output time steps. Do not confuse this with the time step size in your input time history function. The output time step size can be different from the input time step size in your input time history function. The number of

output time steps multiplied by the output time step size is equal to the length of time over which output results are reported.

- **Start from Previous History:** This option allows you to set the initial conditions for the time history analysis to the conditions that exist at the end of a previously run analysis (in the same analysis run). This option is not available for periodic time history analysis.

  Note that in many cases you can accomplish the same thing using the Arrival Time feature in the Load Assignment area. The advantage of the Start from Previous History option is that when you want to start several different time histories from the final conditions of another time history, such as a gravity load time history, you only have to run the other (gravity) time history once rather than multiple times.

  Often you will want to run gravity load as a time history and then start one or more lateral time histories from the final conditions of the gravity load time history using the Start from Previous History option. To run a gravity load time history, define the Load in the Load Assignments area as the load case that contains your gravity load and create an input function from the built-in Ramp time history function template. It is also helpful to set your modal damping high (say 0.99) for this gravity load time history.

Load Assignments

You can add as many different load assignments to a time history case as you desire. To define a load assignment, fill in the appropriate items in the Load, Function, Scale Factor, Arrival Time and Angle boxes and then click the **Add** button.

To modify an existing load assignment, highlight the existing load assignment in the Load Assignment area of the form. Note that the data associated with that load assignment appears in the

edit and drop-down boxes at the top of this area. Modify the load assignment data as desired. Then click the **Modify** button.

To delete an existing load assignment, highlight the existing load assignment in the Load Assignment area of the form. Note that the data associated with that load assignment appears in the edit and drop-down boxes at the top of this area. Then click the **Delete** button.

The following items are included in the Load Assignment area:

- **Load:** This can be defined as acc dir 1, acc dir 2, acc dir 3, or acc dir Z, or a static load case (Dead, Live). The three accelerations (acc dir 1, acc dir 2, and acc dir Z) are ground accelerations in the local axes directions of the time history. Positive acc dir 3 always corresponds to the positive global Z direction. See the description of the Angle item in this area for information about acc dir 1 and acc dir 2. When you specify one of the ground accelerations, your input function defines how the ground acceleration varies with time.

  The static load cases that you can specify in this area may be either force loads or displacement loads. In that case, your input function defines how this load or displacement varies with time.

  Note that you can perform multiple support excitation time history analysis in this program using displacement time histories. To do this, define a static load case with a unit displacement at one or more locations and also define a time history function that defines how that unit displacement varies with time. Repeat this as many times as required. Then, define a time history case with multiple load assignments where each load assignment consists of one of the unit displacement load cases and its associated time history function.

  When you create functions for time history displacement analysis, the time step for the input function should typically be smaller than that for a comparable acceleration time history. The reason for this is that when acceleration varies linearly

Note:

You can perform multiple support excitation time history analysis in this program using displacement time histories.

between two points, displacement varies as a cubic between those same two points. Thus, between those two points, you can define the acceleration just by defining the two points. However, you will have to define the two points and several more in between them to reasonably define the displacement.

- **Function:** This may be any defined time history function.

- **Scale Factor:** This item is a scale factor that is used as a multiplier on the input function values. The units for the scale factor depend on the type of load specified in the Load drop-down box. If the load is specified as a ground acceleration (that is, acc dir 1, acc dir 2, or acc dir Z), this scale factor has units of Length/seconds^2. If the load is a static load case, this scale factor is unitless.

  The scale factor can be any positive or negative number, or zero.

- **Arrival Time:** The arrival time is the time at which a particular load assignment starts. Suppose that you want to apply the same ground acceleration that lasts 30 seconds to your building in the global X and global Y directions. Further suppose that you want the ground acceleration in the global Y direction to start 10 seconds after the ground acceleration in the global X direction begins. In that case, you could specify an arrival time of 0 for the load assignment for the global X direction shaking and an arrival time of 10 for the load assignment for the global Y direction shaking.

  The arrival time can be zero or any positive or negative time. The time history analysis for a given time history case always starts at time zero. Thus, if you specify a negative arrival time for a load assignment, any portion of its associated input function that occurs before time zero is ignored. For example, suppose a particular load assignment has an arrival time of -5 seconds. In that case, the first 5 seconds of the input function associated with that load assignment is ignored by the program.

- **Angle:** The local 1 and 2 axes of the time history case coordinate system lie in the global XY plane. By default the local 1-axis is in the same direction as the positive global X-axis, the local 2-axis is in the same direction as the positive global Y-axis and the local 3-axis is in the same direction as the positive global Z-axis. You can rotate the local 1 and 2 axes of the time history coordinate system about the local 3 (global Z) axis. The Angle item specifies the angle in degrees measured from the positive global X-axis to the positive local 1-axis of the time history case coordinate system. Positive angles appear counterclockwise as you look down on the model.

  The Angle item is only considered if the Load item is either acc dir 1 or acc dir 2. Otherwise, the angle item has no meaning. Note that the angle is always measured to the local 1-axis, even when the Load item is specified as acc dir 2. Thus, if the Load item is specified as acc dir 2, and the angle is specified as 30 degrees, then acc dir 2 and local coordinate direction 2 are oriented at an angle of 120 degrees (measured counterclockwise) from the positive global X-axis.

Load Combinations Command

Click the **Define menu > Load Combinations** command or the button to define load combinations. This command brings up the Define Load Combinations form. The Combinations area of this form lists the names of all the currently defined load combinations. The Click To area of the form allows you to define new load combinations, modify existing load combinations and delete existing load combinations.

Clicking on the **Add New Combo** button or highlighting an existing load combination and clicking the **Modify/Show Combo** button brings up the Load Combination Data form. The following bullet items include brief discussions of each of the areas in this form.

- **Load combination name:** Specifies the name of the load combination; use this edit box to modify the name if desired.
- **Load combination type:** Specifies the type of load combination as ADD (Additive), ENVE (Envelope), ABS (Absolute) or SRSS. The most common type of load combination is ADD.
- **Define Combination:** The actual load combination is created by specifying one or more load cases, each with an associated scale factor. To add a load case to the load combination definition, select the load case name from the Case Name drop-down box, type in an appropriate scale factor in the Scale Factor edit box and click the **Add** button.

  To modify the scale factor for a load case already specified as a part of the load combination definition, highlight the load case name. Note that the load case name and associated scale factor appear in the drop-down box and edit box at the top of the Define Combination area. Type in the revised scale factor in the Scale Factor edit box and click the **Modify** button.

  To delete a load case from the load combination definition, highlight the load case name. Note that the load case name and associated scale factor appear in the drop-down box and edit box at the top of the Define Combination area. Click the **Delete** button.

Mass Source Command

Click the **Define menu > Mass Source** command or the button, which brings up the Define Mass Source form. The following are bullet items describe each of the areas on this form.

- **Mass Definition:** Specifies whether the program determines the building mass based on element/object self masses and any additional masses that you specify, or based on a load combination that you specify. By default the program determines the mass from element/object masses and additional specified masses.

- **Define Mass Multiplier for Loads:** This area is only active if you select the From Loads option in the Mass Definition area. When this area is active, you specify a load combination from which the program determines the building mass. The mass source load combination is created by specifying one or more load cases, each with an associated scale factor. To add a load case to the mass source load combination definition, select the load case name from the Load drop-down box, type in an appropriate scale factor in the Multiplier edit box and click the **Add** button.

  To modify the scale factor for a load case already specified as a part of the mass source load combination definition, highlight the load case name. Note that the load case name and associated scale factor appear in the drop-down box and edit box at the top of the Define Mass Multiplier for Loads area. Type in the revised scale factor in the Multiplier edit box and click the **Modify** button.

  To delete a load case from the mass source load combination definition, highlight the load case name. Note that the load case name and associated scale factor appear in the drop-down box and edit box at the top of the Define Combination area. Click the **Delete** button.

- **Include Lateral Mass Only:** If this check box is checked, only assigned translational mass in the global X and Y axes directions and assigned rotational mass moments of inertia about the global Z-axis will be considered in the analysis. All other assigned masses will be ignored for the analysis. Checking this box is useful if you do not want to consider vertical dynamics in your model. If you do want to consider vertical dynamics, this box must be unchecked.

- **Lump Lateral Mass at Story Levels:** If this check box is selected, all assigned translational mass in the global X and Y direction and assigned rotational mass about the Global Z axis will be lumped together for a given story level.

Chapter 8

Draw Menu

This chapter describes the drawing tools and controls that are available on the Draw menu.

Select Object Command

Tip:

Most of the tools available on the Draw menu are also available on the side toolbar.

The **Draw menu > Select Object** command is used to switch from a drawing mode where mouse clicks draw objects into a selection mode where mouse clicks select objects. Alternative methods of switching from a drawing mode to a selection mode include pressing the Esc key on your keyboard, clicking the button, and executing one of the Select menu commands.

Note that your model must be unlocked in order to draw objects in it. Typically your model is locked after you run an analysis. If you need to unlock your model, use the **Options menu > Lock Model** command or you can click the **Lock/Unlock Model** button, . Note that both the menu command and the toolbar button act as toggle switches to lock and unlock the model.

Typically when you enter a drawing mode, you remain in that drawing mode until you do something to exit it. For example if you are in a mode to draw point objects, you will draw a new point object every time you click the left mouse button until you do something to enter a different drawing mode (e.g., **Draw menu > Draw Line Objects > Draw Lines (plan, elev, 3D)**) or until you do something to enter the select mode (e.g., press the Esc key on your keyboard).

The Similar Stories Feature

Note:

The similar stories feature is only active in plan view. It works for drawing, assigning and selecting.

The Similar Stories feature is helpful when drawing objects in plan view in this program. Similar story assignments are made in the story level data (**Edit menu > Edit Story Data > Edit** command). The drop-down box located in the lower right-hand corner of the screen (just to the left of the drop-down box for the coordinate system) is the similar stories feature drop-down box. The selection made in this box controls what happens when an object is drawn in this program.

The similar stories drop-down box can be set to One Story, All Stories or Similar Stories. Each of these is described herein. Note that the similar stories feature is only active in plan view. It works in conjunction with drawing, assigning and selecting.

- **One Story:** This option means that the drawn object only occurs at the story level on which it is drawn.
- **All Stories:** This option means that the drawn object occurs at all story levels even though it is drawn at only one story level.

- **Similar Stories:** This option means that the drawn object occurs at all story levels designated as similar in the story level data. Suppose that Level XX is designated similar to Level YY. Then, when this option is active, an object drawn on Level XX also occurs in Level YY and an object drawn on Level YY also occurs in Level XX.

Reshape Object Command

You can activate the reshaper tool by either clicking on **the Reshaper** button, , or by clicking **Draw menu > Reshape Object**. When you activate the reshaper tool, you enter reshape mode. When active, you remain in reshape mode until you do one of the following:

- Click the **Pointer** button, .
- Press the Esc key on your keyboard.
- Choose one of the drawing options from the Draw menu or clicking one of the **Draw** buttons.
- Click on one of the select items in the Select menu.
- Run an analysis.

While in reshape mode, click on an area, line or point object and modify it in one of several ways. The ways you can modify each of these objects are described in the subsections in this chapter.

Tip:

When reshaping objects, you cannot move them in the Z direction such that they cross a story level.

Whether you are reshaping an area, line or point objects, be sure to read the subsection entitled "Moving/Reshaping Objects in the Z Direction."

Note that the drawing constraints described in the subsection entitled "Drawing Constraints" in this chapter are available when you use the reshaper tool.

Reshaping Area Objects

When you are in reshape mode and you click on an area object, a series of selection handles (squares that are the opposite color from the background color) appear at all corners of the area object. You can then do any of the following:

- Left click on the area object and while holding down the left mouse button, drag it to a new location. The area object retains its original shape; it is simply relocated. Note that when you move the area object in this way, the corner points of the area object are disconnected from any other objects they might have been connected to. Thus reshaping the area object in this manner only affects the area object, not any surrounding elements that it may be connected to.

- Left click on one of the corner points of the area object and while holding down the left mouse button, drag the corner point to a new location. The other corner points remain in their original locations; the area object takes on a different shape. Note that when you move the corner point of an area object in this way, the corner point is disconnected from any other objects it might have been connected to. Thus, reshaping the area object in this manner only affects the area object, not any surrounding elements that it may be connected to.

- Right click on one of the corner points of the area object. This brings up a form where you can modify the global X and/or Y and/or Z coordinates of the corner point. The other corner points remain in their original locations; the area object takes on a different shape. Note that when you move the corner point of an area object in this way, the corner point is disconnected from any other objects it might have been connected to. Thus reshaping the area object in this manner only affects the area object and not the surrounding elements that it may be connected to. For more information, see the subsection in this chapter entitled " Moving/Reshaping Objects in the Z Direction."

Note:

The drawing constraints described in the subsection entitled "Drawing Constraints" are available when you use the reshaper tool.

Reshaping Line Objects

When you are in reshape mode and you click on a line object, selection handles (squares that are the opposite color from the background color) appear at the ends of the line object. You can then do any of the following:

- Left click on the line object and while holding down the left mouse button drag, it to a new location. The line object retains its original length; it is simply relocated. Note that when you move the line object in this way, the end points of the line object are disconnected from any other objects they might have been connected to. Thus reshaping the line object in this manner only affects the line object and not the surrounding elements that it may be connected to.

- Left click on one of the end points of the line object and while holding down the left mouse button drag the end point to a new location. The other end point remains in its original location; the length of the line object changes. Note that when you move the end point of a line object in this way, the end point is disconnected from any other objects it might have been connected to. Thus reshaping the line object in this manner only affects the line object and not the surrounding elements that it may be connected to.

- Right click on one of the end points of the line object. This brings up a form where you can modify the global X and/or Y and/or Z coordinates of the end point. The other end point remains in its original location; the length of the line object changes. Note that when you move the end point of a line object in this way, the end point is disconnected from any other objects it might have been connected to. Thus reshaping the line object in this manner only affects the line object and not the surrounding elements that it may be connected to. For more information, see the subsection in this chapter entitled " Moving/Reshaping Objects in the Z Direction."

Reshaping Dimension Lines

Dimension lines are described in the subsection entitled "Dimension Lines" in this chapter. You can use the reshaper tool to move a dimension line to a new location. You cannot lengthen or shorten a dimension line, even by using the reshaper tool.

To relocate a dimension line, first click on the line so that the selection handles (squares that are the opposite color from the background color) appear at the ends of the dimension line. Then left click on the dimension line and, while holding down the left mouse button, drag it to a new location. The dimension line retains its original length; it is simply relocated. The leader lines are automatically adjusted as needed.

Reshaping Point Objects

When you are in reshape mode and you click on a point object, a selection handle (square that is the opposite color from the background color) appears on the point object. You can then do any of the following:

- Left click on the point object and, while holding down the left mouse button, drag it to a new location. When you move the point object in this way, all of the objects connected to the point move too; they are either reoriented or resized, or both. Unlike area and line objects, the point object does not gets disconnect from the objects it is attached to when it is reshaped/moved.

- Right click on the point object. This brings up a form where you can modify the global X and/or Y and/or Z coordinates of the point. When you move the point object in this way, all of the objects connected to the point move too; they are either reoriented or resized, or both. Unlike area and line objects, the point object does not disconnect from the objects it is attached to when it is reshaped/moved. For more information, see the subsection in this chapter entitled " Moving/Reshaping Objects in the Z Direction."

Moving/Reshaping Objects in the Z Direction

You can only move/reshape objects in the Z-direction within their own story level or to the story level below. You cannot specify a Z coordinate that requires an object to move across a story level.

For example, suppose you have a four-story building with 10-foot story heights at all levels. Thus, the first story level is at an elevation of 10 feet, the second story level is at 20 feet, the third story level is at 30 feet and the fourth story level is at 40 feet. Further, suppose that you are relocating an area object corner point that occurs at the mid-height of the third story level; that is, at an elevation of 25 feet.

You can specify a new Z coordinate for this corner point between 20 feet and 30 feet inclusive; that is, between the elevations of the second and third story levels inclusive. If you specify a Z coordinate less than the second story level elevation, the program moves the point to the second story level elevation. If you specify a Z coordinate greater than the third story level elevation, the program moves the point to the third story level elevation. If you specify a Z coordinate between the second and third story elevations, inclusive, the program moves the point to the specified elevation.

Draw Point Objects Command

You can only draw point objects in plan view. You cannot draw them in an elevation or three-dimensional view. To draw point objects, click the **Draw menu > Draw Point Objects** command or click the **Draw Point Objects** button . After you have activated the Draw Point Objects command or button, there are two ways you can draw the point objects. They are:

- Left click at any location in a plan view to draw a point object.
- Working in plan view, depress and hold down the left button on your mouse. While keeping the left button depressed, drag

your mouse to " band" a window around one or more grid line intersections. Then release the left mouse button. Point objects will automatically be placed at each grid line intersection *in the same coordinate/grid system* included in the "rubber band" window.

Draw Line Objects Command

To draw line objects, use the **Draw menu > Draw Line Objects** command. A submenu will appear with five line object drawing options. Alternatively, you can select one of the five line object drawing options directly by clicking the appropriate button. Those line object drawing options and their associated toolbar buttons are described in the bullet items that follow. Note that the items in parenthesis, such as (plan, elev, 3D), indicate the types of views for which the line object drawing option can be enabled.

Also note that if the Show Flowing Property Window toggle switch under the **Options** menu is enabled and you click a menu item or toolbar button to draw a line object, a Properties of Object box will pop up. The Properties of Object box provides various definition parameters and drawing controls for line objects, depending on the drawing tool selected. Review the parameters and controls shown in this box before drawing your line object to ensure that they are what you want. Change any entry in the box by clicking on it and making a new selection from the drop-down box or typing in new information into the edit box, as appropriate.

Tip:

To finish drawing a line object, either double left click on the end point or single left click and then press the Enter key (or the Esc key) on your keyboard.

- **Draw Lines (plan, elev, 3D),** **:** To draw a line object using this command, left click once at the beginning of the line and then drag the mouse to the end location of the line and left click again. Note that as you drag the mouse, a dashed line is visible, indicating the current extent of the line object.

  If you left click once on the end point of the line object, the program assumes that you want to draw another line object

starting from that point. Thus when you left click on a third point, a second line object is created spanning from the second point clicked to the third point clicked. This process continues indefinitely when you left click (single click) on the end point of the line object. You can always tell if the program is expecting you to draw the second point of a line object because you will see the dashed "rubber band" line indicating the current extent of the line as you drag the mouse. You might think of the single left click as finishing the line but not picking up your pencil from the paper.

If you double left click on the end point of the line, the program assumes that you do not want to draw another line starting at that point. You might think of the double left click as finishing the line and picking your pencil up from the paper. When you double left click to finish drawing a line object, the line drawing mode is still enabled. In other words, you can move your mouse pointer to a new location and start to draw a new line.

If you single click to finish drawing a line object and then decide that you did not want to draw another line object starting from that point, press the Enter key on the keyboard. This will terminate the drawing of the next line and you will notice that the dashed "rubber band" line disappears. Single left clicking to finish drawing a line object and then pressing the Enter key on the keyboard is equivalent to double left clicking to finish drawing a line object.

You can single click to finish drawing a line object and then press the Esc key on your keyboard. This terminates the drawing of the next line and takes you out of Draw mode and into Select mode. See the section entitled "Two Modes" in Chapter 1 for more information about the draw and select modes.

When using this command in an elevation view or 3D view, if you draw a line object that crosses story levels, the program immediately breaks the object up at the story levels. For exam-

ple, if you draw a line object that has its top at the 4th story level and it bottom at the 2nd story level, the program immediately breaks the object up into two objects, with the break point at the 3rd story level.

- **Create Lines in Region or at Clicks (plan, elev, 3D),** **:** This command works in two different ways. You can click on any grid line (in plan view only) and a line object is drawn on that grid line between the two adjacent intersecting grid lines from the same coordinate/grid system.

  Alternatively, in all views, you can depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to "rubber band" a window around one or more grid line segments. Then release the left mouse button. Line objects are automatically placed at each grid line segment included in the "rubber band" window. The term grid line segment in this paragraph means that portion of a grid line that is between the two adjacent intersecting grid lines *from the same coordinate/grid system*.

- **Create Columns in Region or at Clicks (plan),** **:** After you have activated the Create Columns in Region or at Clicks (plan) command, you can draw the columns in two ways:

  ✓ Left click at any location in a plan view to draw a column (vertical line object below).

  ✓ Working in plan view, depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to "rubber band" a window around one or more grid line intersections. Then release the left mouse button. Columns (vertical line objects below) are automatically placed at each grid line intersection of two grid lines *in the same coordinate/grid system* included in the "rubber band" window.

  The columns (vertical line objects) extend from the story level that you draw them on to the story level below, and if

Note:

The spacing and orientation of secondary beams are controlled in the floating Properties of Object window.

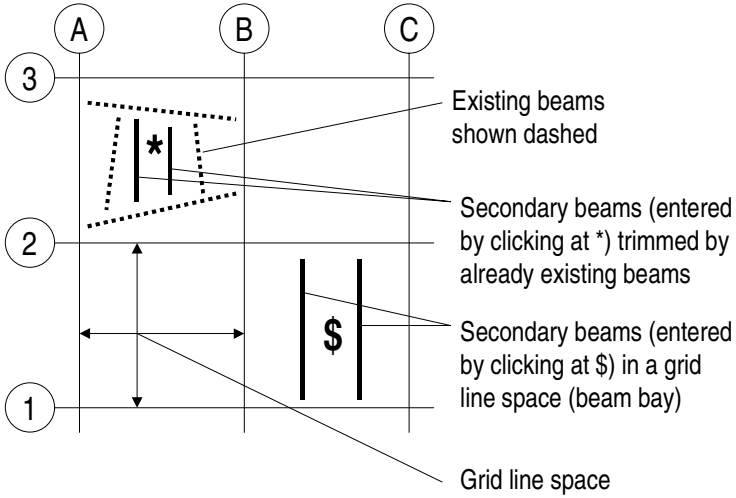


Figure 8-1: *Example of secondary beams*

you have the similar stories feature in the program status bar activated, also to other story levels.

- **Create Secondary Beams in Region or at Clicks (plan),** **:** This command allows you to draw typical infill (secondary) beams for an entire grid line space (beam bay) in a single click. The grid line space is defined by four adjacent intersecting grid lines. If there are existing beams already in the grid line space, the spacing and extent (length) of the secondary beams is based on the existing beams rather than the grid lines. Figure 8-1 shows an example of a grid line space and secondary beams. Note that secondary beams are not included on the grid lines.

  This command works in two different ways. You can click inside the space created by adjacent intersecting grid lines (from the same coordinate/grid system), and secondary beams will be drawn in that space.

  Alternatively, you can depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to "rubber band" a window around one or more grid line spaces. Then release the left mouse button. Secondary beams are automatically placed in each grid line space within

Note:

The type of brace created (X, V, chevron, eccentric, etc.) is controlled in the floating Properties of Object window.

the region that is fully included in the "rubber band" window. The term grid line space in this paragraph means the space created by adjacent intersecting grid lines (from the same coordinate/grid system).

- **Create Braces in Region or at Clicks (elev), :** This command works in an elevation view. It allows you to quickly draw brace elements in a space bounded by two adjacent grid lines (from the same coordinate/grid system) and two adjacent story levels.

  This command works in two different ways. You can click inside the space created by the intersection of two adjacent grid lines (from the same coordinate/grid system) and two adjacent story levels.

  Alternatively, you can depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to "rubber band" a window around one or more grid line/story level spaces. Then release the left mouse button. Braces are automatically placed in each grid line/story level space within the region that is fully included in the "rubber band" window. The term grid line/story level space in this paragraph means the space created by the intersection of two adjacent grid lines (from the same coordinate/grid system) and two adjacent story levels.

Draw Area Objects Command

To draw area objects, use the **Draw menu > Draw Area Objects** command. A submenu will appear with five area object drawing options. Alternatively, you can select one of the five area object drawing options directly by clicking the appropriate toolbar button. Those area drawing options and their associated toolbar buttons are described in the bullet items that follow. Note that the items in parenthesis, such as (plan, elev, 3D), indicate the types of views for which the area object drawing option is active.

will pop up as soon as you click the menu item or toolbar button to draw a line object.

Also note that if the Show Flowing Property Window toggle switch under the **Options** menu is enabled and you click a menu item or toolbar button to draw an area object, a Properties of Object box will pop up. The Properties of Object box provides various definition parameters and drawing controls for area objects, depending on the drawing tool selected. Review the parameters and controls shown in this box before drawing your area object to ensure that they are what you want. Change any entry in the box by clicking on it and making a new selection from the drop-down box or typing in new information into the edit box, as appropriate.

Tip:

To finish drawing an area object, double left click on the last point or single left click and then press the Enter key on your keyboard.

- **Draw Areas (plan, elev, 3D), :** To draw an area object using this command, left click once at the first corner point of the area, drag the mouse to the next corner point and left click, and so on to define each corner point of the area object. Note that as you drag the mouse, a dashed line is visible indicating the current extent of the area object.

  When you reach the last corner point of the area object, double left click or single left click and then press the Enter key on the keyboard to finish the object.

  An area object drawn using this command must have at least three corner points. Typically area objects are limited to no more than four corner points; however, there is no limit on the maximum number of corner points allowed for horizontal area objects (in the global XY plane).

  When using this command in a 3D view, the program does not allow the area object drawn to cross a story level. For example, you cannot draw a vertical area object in a 3D view that has its top at the 4th story level and its bottom at the 2nd story level. With the top at the 4th level, the bottom cannot be below the 3rd story level because this would cause the area object to cross a story level.

- **Draw Rectangular Areas (plan, elev),** **:** This command requires two left clicks to draw the rectangular area object. Left click once to define the position of one corner point of the area. Then drag the mouse and left click again to define the diagonally opposite corner point. Note that as you drag the mouse, a dashed line is visible indicating the current extent of the area object.

  When using this command in an elevation view, if you draw an area object that crosses story levels, the program immediately breaks the object up at the story levels. For example, if you draw an area object that has its top at the 4th story level and it bottom at the 2nd story level, the program immediately breaks the object up into two objects, with the break line at the 3rd story level.

- **Create Areas at Click (plan, elev),** **:** This command allows you to draw area objects in one or more grid line spaces at a single click. The grid line space is defined by four adjacent intersecting grid lines. Figure 8-1 shows an example of a grid line space. To draw the area object simply left click in the grid line space.

- **Draw Walls (plan),** **:** Once you have activated the Draw Walls (plan) command working in plan view, left click once at the beginning of the wall below and then drag the mouse to the end of the wall below and left click again. Note that as you drag the mouse, a dashed line is visible indicating the current extent of the area object (wall below).

  If you left click once on the end of the wall below, the program assumes that you want to draw another area object (wall below) starting from that point. Thus when you left click on a third point, a second area object (wall below) is created extending from the second point clicked to the third point clicked. This process continues indefinitely when you left click (single click) on the end of the area object. You can always tell if the program is expecting you to draw the end point of an area object (wall below) because you will see the dashed "rub-

ber band" line indicating the current extent of the wall as you drag the mouse. You might also think of the single left click as finishing the wall but not picking up your pencil from the paper.

If you double left click on the end of the wall below (area object), the program assumes you do not want to draw another wall starting at that point. You might think of the double left click as finishing the wall and picking your pencil up from the paper. When you double left click to finish drawing an area object (wall below), you still remain in same area object drawing mode. In other words, you can move your mouse pointer to a new location and start to draw a new wall.

If you single click to finish drawing a wall below and then decide that you did not want to draw another wall below starting from that point, press the Enter key on the keyboard. This will terminate the drawing of the next wall; notice that the dashed "rubber band" line disappears. Single left clicking to finish drawing an area object (wall below) and then pressing the Enter key on the keyboard is equivalent to double left clicking to finish drawing an area object.

Note that area objects representing walls are broken at story levels. They are also broken at turns in developed elevations; that is, at locations where the plane displayed by the developed elevation changes.

- **Create Walls in Region or at Click (plan), :** This command works in two different ways. You can click on any grid line (in plan view) and a wall below (area object) is drawn on that grid line between the two adjacent intersecting grid lines from the same coordinate/grid system.

  Alternatively, you can depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to "rubber band" a window around one or more grid line segments. Then release the left mouse button. Area objects (walls below) are automatically placed at each grid line

segment included in the "rubber band" window. The term grid line segment in this paragraph means that portion of a grid line that is between the two adjacent intersecting grid lines from the same coordinate/grid system.

Draw Developed Elevation Definition Command

Developed elevations are special user-defined elevations. These elevations can simultaneously show multiple faces of the building in a single "unfolded" elevation view. Use the following steps to define a developed elevation:

- Click the **Draw menu > Draw Developed Elevation Definition** command.
- The Elevation Views form appears. In this form, you can define names for developed elevations that you plan to draw, modify the names of developed elevations after they have been drawn, and delete names of existing developed elevations. When finished defining or modifying developed elevation names, highlight the name of the elevation that you first want to work with and then click the **OK** button.
- The program windows then all switch to special plan views used for creating and modifying developed elevation views. Note the following about these special plan views:
  - ✓ In the special plan views, you can only see and work with one developed elevation definition at a time. You can use the drop down box in the status bar (at the bottom of the program window) that ordinarily displays the similar stories option to switch to plan views of different developed elevations.
  - ✓ When you click the **OK** button in the Elevation Views form, you enter the special plan views to work on the developed elevation that was highlighted in the form. This developed elevation may or may not currently be defined. If it is not defined, you will see a blank plan view, other-

Note:

A line defining a developed elevation can not intersect (cross) itself and it also cannot close.

wise you will see a plan view with the existing developed elevation view shown in it.

When you initially enter the special plan view, you are in a mode where you can start drawing the developed elevation. You do not need to click on any buttons or menu commands to draw the line defining the developed elevation. Simply start left clicking to draw the line. The line defining the developed elevation may be multi-segmented. When you get to the last point defining the developed elevation, double left click or single left click and then press the Enter key on your keyboard to complete the definition.

Important Note: The line defining the developed elevation can not intersect (cross) itself and it also cannot close with itself.

After you have drawn a developed elevation or if the developed elevation you highlighted in the Elevation Views form already exists such that when the special plan view appears a developed elevation definition is showing, you can modify the developed elevation definition. To do this:

- Click on the **Draw menu > Reshape Object** command or the Reshaper button, , on the side toolbar.

- Click on the line defining the developed elevation. Selection handles squares that are the opposite color from the background color appear along the line at the points where you clicked to define the line.

  If you then left click on the line and hold down the mouse left button, you can drag the entire line defining the developed elevation to a new location. Note that the shape of the line does not change.

  Alternatively, you can left click on one of the selection handles, hold down the mouse left button, and drag the selection handle (point) to a new location, thus changing the shape of the line defining the developed elevation.

When you have finished reshaping a developed elevation definition, do one of the following:

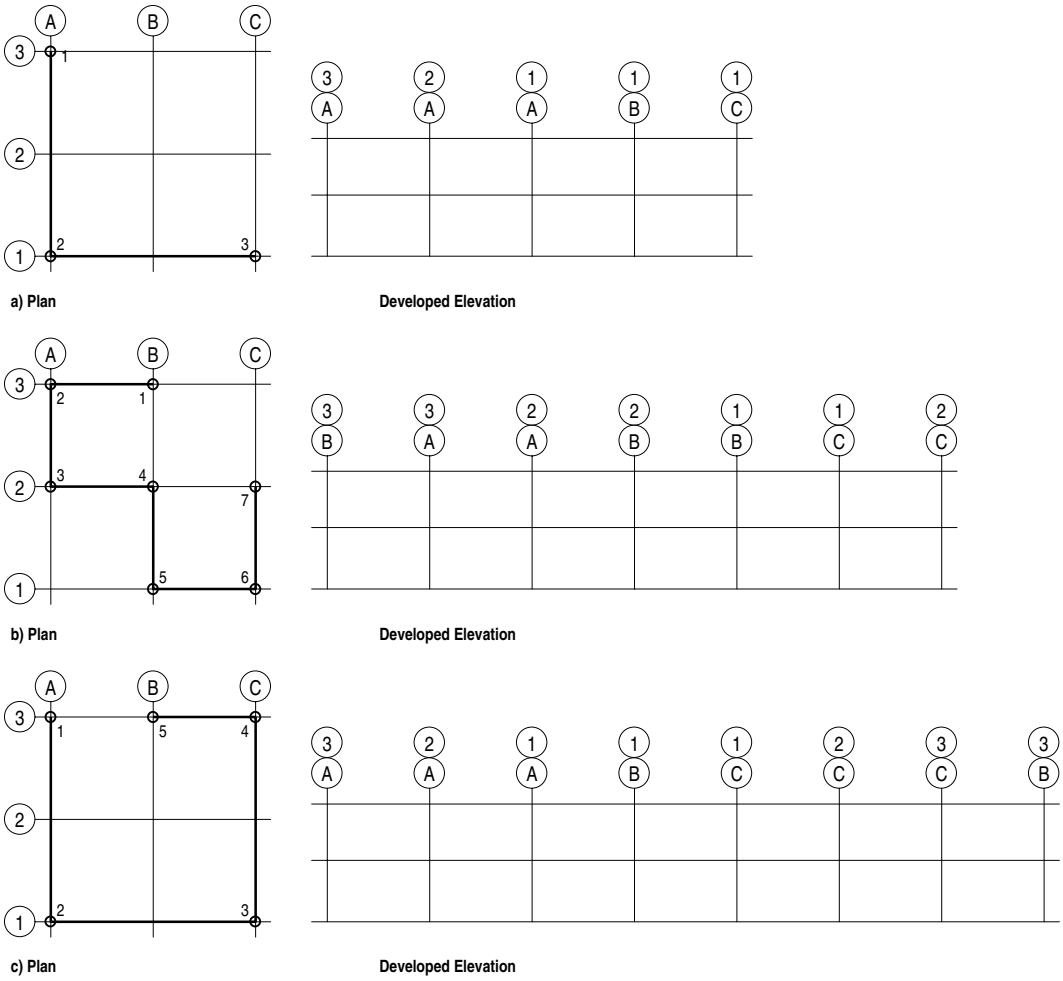
- Right click in the program window (not on any objects) to bring up a selection menu. Click **Restore Views** on this menu. You return to the views you were in before you began to define/modify the developed elevations. Alternatively you could select these views from the view menu or associated toolbar buttons, but it is much easier to use the Restore Views command.

- If instead you want to work with another developed elevation, select a developed elevation view name from the drop-down box in the status bar (at the bottom of the program window) that ordinarily displays the similar stories option. When you click on a developed elevation view in this drop-down box, you enter a mode where you are ready to draw a new elevation, not reshape it. If you want to reshape/modify this developed elevation, click on the **Draw menu > Reshape Object** command or the Reshaper button .

Note that when you have finished drawing a line that defines a developed elevation you can then immediately exit the developed elevation view (we recommend using the Restore Views command as described above to do this), reshape the line (as described above), or click in the drop-down menu on the status bar to work on another developed elevation (as described above).

Figure 8-2 shows three example developed elevations together with their associated plan views. The elevations associated with a particular plan are shown next to the plan. The heavy line in the plan views shows the extent of each developed elevation. The open circles on the heavy lines show the locations where left mouse clicks are required to define the developed elevation. Note that each of the left click locations is numbered in the order the clicks occur.

Note that in Figure 8-2c the final mouse click (point 5) cannot occur at grid intersection A-3 (the same location as point 1) be-



(Above)
Figure 8-2:
Example definitions of developed elevations

cause this would close the line and developed elevations are not allowed to be closed in this manner.

Also note that the three examples shown in Figure 8-2 all have the mouse clicks occurring on grid intersections. It is not necessary to click on grid intersections to define developed elevations. You can click anywhere in plan when defining a developed elevation and the elevation will be created at the point of your click.

Draw Dimension Line Command

You can use the **Draw menu > Draw Dimension Line** command to draw dimension lines at any location in a plan or elevation view. You cannot draw dimension lines in a three-dimensional view. After you have clicked the **Draw menu > Draw Dimension Line** command, use the following steps to draw the dimension line:

- Left click on the point that you want to dimension *from*.
- Left click on the point that you want to dimension *to*.
- The dimension line appears between the two points you clicked on but you are not quite finished creating it. Now drag the dimension line in a direction perpendicular to its original location to the final location for the line. When you have dragged the dimension line to its final location, left click the mouse again and the dimension line is complete.

Note that three left clicks are required to completely draw a dimension line. A completed dimension line has arrow heads at each end, dimension text displaying the length of the dimension line, and leaders to the points being dimensioned if you dragged the dimension line away from those points. You can specify the units that the dimensions are to be displayed in by using the **Options menu > Preferences > Output Decimals** command. See Chapter 14 Options Menu for more information.

Dimension lines are similar to other line objects in that they only appear on the story level you draw them on unless you are using the similar stories feature, which can be enabled using drop-down box on the program status bar. The similar stories feature works for dimension lines.

You can use the **View menu > Set Building View Options** command or the **Set Building View Options** button to toggle the visibility of the dimension lines on or off.

After you have drawn a dimension line, you can select it any time you are in the select mode by clicking directly on it. You cannot select a dimension line by windowing it. You might want to select a grid line to relocate it. There are two features available for relocating grid lines. They are the nudge feature and the reshaper tool.

The nudge feature is described in the section entitled "Nudge Feature" in Chapter 5 View Menu. This feature works as described there for dimension lines.

The reshaper tool is described earlier in this chapter. In addition, the section entitled "Reshaping Dimension Lines" provides information on using the tool to relocate dimension lines.

Draw Reference Point Command

Reference points are drawn in plan view. They are useful because they allow you to snap to the same plan location at any story level. Reference point function somewhat similar to vertical grid lines but they have no identification and no line associated with them. To draw a reference point, use the **Draw Menu > Reference Point** command.

Snap To Command

The snap features allow you to snap to various items when you are drawing or editing. There are six separate snap features available. You can toggle those six features on or off in any combination for simultaneous action using the **Draw menu > Snap to** command or by clicking one or more of the six snap feature buttons. The six snap features and their associated buttons are as follows:

- **Snap to Grid Intersections and Points,** **:** This feature snaps to points and grid line intersections of two grid lines in the same coordinate/grid system. This feature works in plan, elevation and three dimensional views.

- **Snap to Line Ends and Midpoints,** : This feature snaps to the midpoints and ends of line objects and to the midpoints and ends of edges of area objects. Note that the end of an edge of an area object is a corner point of the area object. This feature works in plan, elevation and three dimensional views.

- **Snap to Intersections,** : This feature snaps to the intersections of line objects with other line objects and with the edges of area objects. It does not snap to the intersection of the edge of one area object with the edge of another area object. This feature works in plan view only. It does not work in elevation or three-dimensional views.

- **Snap to Perpendicular Projections,** : This feature works as follows. First draw the first point for a line or area object. Then, if this snap feature is active, place the mouse pointer over another line object or edge of an area object and left click. A line object or edge of an area object will then be drawn from the first point perpendicular to the line object or edge of an area object that the mouse pointer was over when the second point was clicked. This feature works in plan view only. It does not work in elevation or three dimensional views.

- **Snap to Lines and Edges,** : This feature snaps to grid lines, line objects and edges of area objects. This feature works in plan, elevation and three dimensional views.

- **Snap to Fine Grid,** : This feature snaps to an invisible grid of points. The spacing of the points is controlled by the Plan Fine Grid Spacing Item, which is accessed using the **Options menu > Preferences > Dimensions/Tolerances** command. This feature works in plan view only. It does not work in elevation or three-dimensional views.

Use the following procedure when using the snap commands:

- If the appropriate snap tool is not already activated, select it or use the **Draw menu > Snap to** command and select it from the submenu.

- Move the mouse pointer in the graphics window.
- When a snap location is found close to the mouse pointer, a dot appears at the snap location as well as a text field describing the snap location.

  Note that the distance the pointer must be from a snap location before it snaps to that location is controlled by the Screen Snap to Tolerance item, which is accessed using the **Options menu > Preferences > Dimensions/Tolerances** command.
- When the desired snap location has been found, click the left mouse button to accept it.
- Modify the snap options if necessary and continue drawing or editing objects.

The snap options are evaluated in the order they are listed above. This order is repeated below for easier reference. If more than one snap option is active and the mouse pointer is located such that it is within the screen snap to tolerance of two different snap features, it will snap to the snap feature that is first in the list below. This is true even if the item associated with the other snap feature is closer to the mouse pointer as long as both items are still within the screen snap to tolerance.

- **Snap to Grid Intersections and Points,** .
- **Snap to Line Ends and Midpoints,** .
- **Snap to Intersections,** .
- **Snap to Perpendicular Projections,** .
- **Snap to Lines and Edges,** .
- **Snap to Fine Grid,** .

As an example, suppose Snap to Intersections and Snap to Fine Grid are both enabled. Assume that the mouse pointer is located

such that it is within the screen snap to tolerance of both an intersection of two line objects and one of the invisible grid points. The snap will be to the intersection of the two line objects because this snap feature occurs first in the above list.

When two items from the same snap feature are within the screen snap to tolerance of the mouse pointer, the snap occurs to the first drawn item, which may or may not be the closest item.

Select Menu

General

The Select menu provides basic options and tools for selecting objects in your model. This chapter describes those options and tools. Do not overlook the Note about Window Selections in Plan View at the end of the "Window" bullet item in the next section.

Note that the similar stories feature works for selections but only in plan view.

Basic Methods of Selecting Objects

The three basic methods of selecting objects in this program are the following:

- **Left click:** Left click on an object to select it. If there are multiple objects, one on top of the other, hold down the

Ctrl key on your keyboard as you left click on the objects. A form will appear that allows you to specify which object you want to select.

Note:

When selecting by window in a plan view (not a perspective plan view), only the visible objects that lie fully in the plane of the plan view are selected.

- **Window or "Windowing":** Draw a window around one or more objects to select them. To draw a window around an object, first position your mouse pointer beyond the limits of the object; for example, above and to the left of the object(s) you want to window. Then depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse to a position below and to the right of the object(s) you want to select. Release the left mouse button to complete the selection. Note the following about window selection:

  - ✓ As you drag your mouse, a "rubber band window" appears. The rubber band window is a dashed rectangle that changes shape as you drag the mouse. One corner of the rubber band window is at the point where you first depressed the left mouse button. The diagonally opposite corner of the rubber band window is at the current mouse pointer position. Any visible object that is completely inside the rubber band window when you release the left mouse button is selected.

  - ✓ As long as you are beyond the limits of the object(s) you want to select, you can start the window at any point; for example, above and to the right, below and to the left or below and to the right of the object(s) you want to select. In all cases, you would then drag your mouse diagonally across the object(s) you want to select.

Note:

An entire object must lie within the rubber band window for the object to be selected.

An *entire* object must lie within the rubber band window for the object to be selected.

Note about Window Selections in Plan View: When selecting by window in a plan view (not a perspective

plan view) only the visible objects that lie *fully* in the plane of the plan view are selected. In other words, only the visible point objects, horizontal area objects and horizontal line objects within the select window are selected.

Tip:

Use the intersecting line selection method in a perspective view to select all columns at a story level.

- **Intersecting Line:** Draw a line through one or more objects to select them. To use this selection method, click the **Select menu > Select Using Intersecting Line** command or the **Set Intersecting Line Select Mode** button, . Then position your mouse pointer to one side of the object(s) you want to select. Depress and hold down the left button on your mouse. While keeping the left button depressed, drag your mouse across the object(s) you want to select. Release the left mouse button to complete the selection. Note the following about the intersecting line selection method:

  ✓ As you drag your mouse a "rubber band line" appears. The rubber band line is a dashed line that changes length and orientation as you drag the mouse. It extends from the point where you first depressed the left mouse button to the current mouse pointer position. Any visible object that is intersected (crossed) by the rubber band line when you release the left mouse button is selected.

  ✓ After using this method to make a selection, the program defaults to the window selection mode. Thus, each time you want to use the Select Using Intersecting Line method, you must use the menu or click the button.

Menu Methods of Selecting Objects

Table 1 identifies the submenu commands and related actions accessed using the **Select menu** command.

Table 1 Submenus of the Select Menu Command

Command	Action
Select on XY plane	Click on a single point and all objects (point, line and area) that are in the same XY plane as the selected point are also selected. The object must lie entirely in the associated plane to be selected.
Select on XZ plane	Click on a single point and all objects (point, line and area) that are in the same global XZ plane as the selected point are also selected. The object must lie entirely in the associated plane to be selected.
Select on YZ plane	Click on a single point and all objects (point, line and area) that are in the same global YZ plane as the selected point are also selected. The object must lie entirely in the associated plane to be selected.
Select by groups	Select the name of any collection of objects that has been defined as a group from the Select Group box and that group will be selected.

Note:

These selection methods also work for deselection.

Table 1 Submenus of the Select Menu Command

Command	Action
Select by frame sections	Select a frame section property name from the Select Sections box and all line objects that have been assigned that frame section property will be selected.
Select by wall/slab/deck sections	Select a wall/slab/ deck section property name from the Section Selections box and all area objects that have been assigned that wall/slab/deck section property will be selected.
Select by link properties	Select a link property name in the Select Properties box and all line objects that have been assigned that link property will be selected.
Select by line object type	Select a line objects type from the Select Line Object Type box. The choices for the types of line objects are column, beam, brace, null or dimen lines (short for dimension lines).
Select by area object type	Select an area objects type from the Select Area Object Type box. The choices for the types of area objects are floor, wall, ramp or null. Note that openings are a subset of null area objects.
Selecting by PIER ID	Select from the in the Select Pier IDs box the name of any Shell/Area that has been assigned a Pier Label.

Table 1 Submenus of the Select Menu Command

Command	Action
Selecting By Spandrel ID	Select from the Select Spandrel IDs box any Shell/Area that has been assigned a Spandrel Label.
Select by story level	Select a story level from the Story Level box, and all objects (point, line and area) associated with that story level will be selected.
Select all	Selects all objects in the model regardless of whether they are visible or not. Be careful using this command. It does not just select what is showing in a particular window, but rather it literally selects all objects in your model. You can also use the **Select All** button to execute this command.
Select invert	Changes the selection such that the currently selected objects are no longer selected and all objects that are not currently selected are selected.

Deselect Command

You can deselect objects one at a time by left clicking on the selected objects. Alternatively, use the **Select menu > Deselect** command and its submenus for quicker and more specific deselection actions. This command gives you access to submenu items similar to those described in the previous section, except that executing the **Select menu > Deselect** command and an associated submenu item deselects rather than selects an object(s).

For example, assume that you want to select all of the objects in your model except for those in a particular XZ plane. Do this quickly and easily by first using the **Select menu > Select All** command and then using the **Select menu > Deselect > XZ Plane** command.

Get Previous Selection Command

As the name suggestions, the **Select menu > Get Previous Selection** command, gets the previously selected object(s). For example, assume you select some line objects by clicking on them and you then assign them some frame section properties. Use the **Get Previous Selection** command or the

button to select the line objects again and assign something else to them, such as member end releases.

Clear Selection Command

As the name suggests, the **Select menu > Clear Selection** command and its associated button

clear all currently selected objects. It is an all or nothing command. You cannot selectively clear a portion of a selection using this command.

Assign Menu

General

The Assign menu provides basic options and tools for assigning section properties, loads, and more to area, line and point objects in your model. This chapter describes the Assign options and tools. Note that before you make an assignment to an object using the Assign menu, you must first **select** the object. Commands for selecting objects are described in Chapter 9 Select Menu.

Assign Joint/Point Commands

Use the **Assign menu > Joint/Point** command to make assignments to point objects. The following subsections describe the assignments that you can make to point objects.

Joint Point > Rigid Diaphragm Command

Use the **Assign menu > Joint/Point > Rigid Diaphragm** command to designate that a selected point is a rigid diaphragm. This command assigns a diaphragm constraint to all of the selected points. **The selected points should typically all lie in a plane that is parallel to the global X-Y plane.** No points other than those actually selected are included in the diaphragm constraint.

Tip:

*You can also assign rigid diaphragms to area objects using the **Assign menu > Shell/Area > Rigid Diaphragm** command.*

When you select one or more point objects and click the **Assign menu > Joint/Point > Rigid Diaphragm** command or the button, the Assign Diaphragms form appears. The Diaphragms area of the Assign Diaphragms form lists the names of all the currently defined rigid diaphragms. The Click To area of the form allows you to define new rigid diaphragms, change an existing diaphragm name and delete an existing diaphragm.

Note:

Rigid diaphragms can only be horizontal. Thus when assigning a rigid diaphragm constraint to point objects, all of the selected points should lie in a plane that is parallel to the global X-Y plane.

To assign a new diaphragm, select the point object(s) to which you want to assign the diaphragm designation and click the **Assign menu > Joint/Point > Rigid Diaphragm** command or the button. In the Assign Diaphragm form, type a new diaphragm name in the edit box of the Diaphragms area, for example D2. Click the **Add New Diaphragm** button, and then click the **OK** button to complete the assignment.

Now assume you want additional point objects to be rigid diaphragms and for those rigid diaphragms to have a D2 designation, the same as the point objects in the previous paragraph. To do this, select the additional point objects and click the **Assign menu > Joint/Point > Rigid Diaphragm** command or the button. Highlight the name D2 in the Diaphragms area of the Assign Diaphragm form; note that D2 appears in the edit box. Click the **OK** button. Note that this gives the selected points the D2 designation, increasing the number of point objects that are designated as rigid diaphragms and identified with the D2 assignment.

If you decide that some point objects previously identified as rigid diaphragms should not have that assignment, make the

Tip:

To delete specific point objects from a diaphragm definition, select those point objects, enter the Assign Diaphragm form, highlight None in the Diaphragms area and click the ***OK*** *button.*

change as follows. Select the point objects to be changed. Click the **Assign menu > Joint/Point > Rigid Diaphragm** command or the button. In the Diaphragms area of the Assign Diaphragm form, highlight None and click the **OK** button.

In the program a rigid diaphragm translates within its own plane (global X-Y plane) and rotates about an axis perpendicular to its own plane (global Z-axis) as a rigid body. Including point objects in a rigid diaphragm definition has no affect on the out-of-plane behavior of the point objects.

If you check the Disconnect from All Diaphragms check box, the point will be disconnected from all diaphragms, regardless of whether it is included within an area object given a rigid diaphragm assignment.

Note that you can also apply a rigid diaphragm to an area object. In most instances, it is better to assign the rigid diaphragm to an area object.

Joint/Point > Panel Zone Command

A panel zone assignment to a point object allows differential rotation and in some cases differential translation at beam-column, beam-brace and column-brace connections. You specify a panel zone assignment by selecting the point object and clicking the **Assign menu > Joint/Point > Panel Zone** command. This pops up the Assign Panel Zone form.

Making a panel zone assignment to a point object you indicates the properties of the panel zone, the connectivity at the panel zone, the local axes orientation for the panel zone and an assignment option for the panel zone. Each of those items is described in the subsections that follow. The headings used for the subsections correspond to the areas in the Assign Panel Zone form which comes up when you use the Assign men > Joint/Point > Panel Zone command or the button.

You cannot assign multiple panel zones to the same point object.

Properties

When you specify panel zone properties, you are actually specifying the stiffness of the springs used to model the panel zone. See the subsequent subsection entitled "Connectivity" for more information. The following options are available for specifying panel zone properties:

- **Elastic Properties from Column:** In this case, only rotational properties for bending about the major axis (local 3-axis) and minor axis (local 2-axis) are taken from the column. Those rotational properties are assigned to the panel zone spring that connects the two program-created internal joints at the panel zone. For all other degrees of freedom, the internal joints at the panel zone are assumed to be rigidly connected.

  When you select this properties option, the only active option for Connectivity is beam-column and the only active option for the local 2-axis is From Column.

  If you specify this option and there is no column connected to the point object with the panel zone assignment, the program ignores this panel zone assignment.

Tip:

Assigning link element properties to a panel zone is a little more complicated, but it provides the most versatility. If you want the panel zone to behave nonlinearly in a nonlinear static or dynamic analysis, you must specify the panel zone properties as a link property.

- **Elastic Properties from Column and Doubler Plate:** When using this option, you specify a doubler plate thickness. The program then changes the web thickness (local 2-axis direction) of the column to be equal to the original web thickness plus the specified doubler plate thickness and calculates the properties of this modified section. The rotational properties for bending about the major axis (local 3-axis) and minor axis (local 2-axis) are taken from the modified column section. Those rotational properties are assigned to the panel zone spring that connects the two program-created internal joints at the panel zone. For all other degrees of freedom, the internal joints at the panel zone are assumed to be rigidly connected.

  When you select this properties option, the only active option for Connectivity is beam-column and the only active option for the local 2-axis is From Column.

If you specify this option and there is no column connected to the point object with the panel zone assignment, the program ignores the panel zone assignment.

- **Specified Spring Properties:** When using this option, you specify rotational spring stiffnesses for major axis bending (about the local 3-axis of the column and panel zone) and minor axis bending (about the local 2-axis of the column and panel zone). Those two rotational spring properties are assigned to the panel zone spring that connects the two program-created internal joints at the panel zone. For all other degrees of freedom, the internal joints at the panel zone are assumed to be rigidly connected.

  When you select this properties option, the only active option for Connectivity is beam-column and the only active option for the local 2-axis is From Column.

- **Specified Link Property:** When using this option, you specify a link element property for the panel zone. The link element properties are assigned to the spring that connects the two program-created internal joints at the panel zone. In this case, this spring may have properties for all six degrees of freedom if nonzero link properties are defined for all six degrees of freedom. If the link element property has zero properties for a particular degree of freedom, the program assumes that the degree of freedom is rigidly connected. Therefore, given this assumption, if for some reason you want one of the degrees of freedom of the panel zone to have essentially zero stiffness, specify a small stiffness for that degree of freedom in the link properties.

  If you have nonlinear static properties defined for the link property, those properties are considered when you run a static nonlinear (pushover) analysis. Similarly, If you have nonlinear dynamic properties defined for the link property, those properties are considered when you run a nonlinear time history analysis. Thus, when you indicate that the panel zone proper-

Note:

A panel zone assignment allows relative movement, typically rotation, between beam and column, beam and brace, or brace and column members.

ties are based on a specified link property, you can model nonlinear behavior in the panel zone.

When you select this properties option, all three options are available and active in the Connectivity area and both options are available and active in the Local Axis area.

Connectivity

The choices for panel zone connectivity are beam-column, beam-brace and brace-column. Unless you choose the Specified Link Property option in the Properties area, the only active panel zone connectivity option is beam-column.

If you specify a type of panel zone connectivity for a point object and the element type specified does not connect to the point object, the program ignores the panel zone assignment. For example, if you specify brace-column connectivity at a point object and there are no braces at the point object, the panel zone assignment is ignored.

Following are descriptions of the three types of panel zone connectivity:

- **Beam - Column:** For beam-column connectivity, two separate joints are internally created by the program to model the panel zone. All beam members are connected to one of the joints and all column members are connected to the other joint. The two joints are connected by a spring that has the properties specified for the panel zone.

  Consider Figure 10-1a, which shows a beam-column joint. Figure 10-1b shows the effect of assigning a panel zone with beam-column connectivity to this joint. Joints 1 and 2 are created internally by the program. They both actually occur at the same location as the point object that is at the beam-column intersection. They are only shown in different locations in the figure for graphical explanation purposes. The column members are connected to joint 1. The beam members are connected to joint 2.

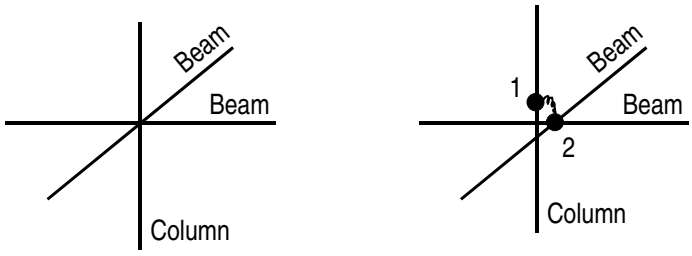


Figure 10-1: *Panel zone connectivity*

a) Beam-Column Connection **b) Panel Zone Representation**

Joints 1 and 2 are connected by zero-length springs that have properties based on the panel zone assignment. Note that the relative movement in the panel zone is between the column elements and the beam elements. There is no relative movement between individual column elements or individual beam elements.

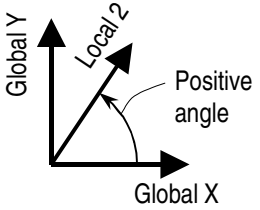
- **Beam - Brace:** For beam-brace connectivity, two separate joints are internally created by the program to model the panel zone. All beam members are connected to one of the joints and all brace members are connected to the other joint. The two joints are connected by a spring that has the properties specified for the panel zone. See the description of Beam - Column connectivity for additional information.

- **Brace - Column:** For brace-column connectivity, two separate joints are internally created by the program to model the panel zone. All brace members are connected to one of the joints and all column members are connected to the other joint. The two joints are connected by a spring that has the properties specified for the panel zone. See the description of Beam - Column connectivity for additional information.

Local Axis

The positive local 1-axis is in the same direction as the positive global Z-axis (upward), always.

On the form you specify the positive direction of the local 2-axis as one of the following:



- ✓ **From Column:** The positive local 2-axis of the panel zone is in the same direction as the positive local 2-axis of the column connected to the panel zone. If columns are connected to the panel zone from both above and below and they have different local axes orientations, the positive local 2-axis of the panel zone is in the same direction as the positive local 2-axis of the column below (and connected to) the panel zone.

  If you specify that the local 2 axis is based on a column and no column exists at the panel zone location, the positive local 2-axis is oriented in the same direction as the positive global X-axis.

- ✓ **Angle:** The direction of the positive local 2-axis of the panel zone is specified by an angle measured in degrees from the positive global X-axis. A positive angle appears counterclockwise as you look down on it from above. An angle of 0 degrees means the positive local 2-axis is in the same direction as the positive global X-axis. An angle of 90 degrees means the positive local 2-axis is in the same direction as the positive global Y-axis.

The direction of the positive local 3-axis is determined from the directions of the local 1 and 2 axes described above using the right hand rule.

Unless you choose the Specified Link Property option in the Properties area, the only active local axis option is From Column.

Options

Two assignment options are possible:

- **Replace Existing Panel Zones:** Replaces the currently specified panel zone (spring stiffness), if any, with the new panel zone assignment. If there is not an existing assignment, the new assignment is still made. This is the default option.

- **Delete Existing Panel Zones:** Deletes the panel zone assignment made to the selected point object(s).

  Note that the default option is Replace and that the program defaults to this every time the form is opened.

Joint/Point > Restraints (Support) Command

Use the **Assign menu > Joint/Point > Restraints (Supports)** command or the button to bring up the Assign Restraints form where you can assign restraints (supports) to selected point objects. Note that restraints are always specified in the global coordinate system.

The six possible degrees of freedom available for a point object are listed in the Restraints in Global Directions area of the form. Place a check in the check box associated with any degree of freedom that you want to be restrained. Any degree of freedom whose associated box is not checked is assumed to be unrestrained assuming, of course, that the degree of freedom has been designated as active for the model. See the Section entitled "Set Analysis Options Command" in Chapter 11 Analyze Menu for additional information.

Tip:

The fast restraint buttons provide a quick and easy way of assigning typical restraint conditions. For unusual conditions, the fast restraint buttons may not be appropriate.

The Fast Supports area of the Assign Restraints form provides four buttons that quickly set the restraint conditions for you by checking and unchecking various check boxes in the Restraints in Global Directions area. The four fast restraint buttons are:

- **:** This is the fast *fixed base* restraint button. All six degrees of freedom are restrained (boxes checked) when you click on this button.

- **:** This is the fast *pinned* base button. All three translation degrees of freedom are restrained (boxes checked) and all three rotation degrees of freedom are unrestrained (boxes not checked) when you click on this button.

- **:** This is the fast *roller support* button. Only the Z translation is restrained (box checked) when you click on this button. All other degrees of freedom are unrestrained (boxes *not* checked).

- **:** This is the fast *no support* button. All degrees of freedom are unrestrained (boxes *not* checked) when you click on this button.

Joint/Point > Point Springs Command

Use the **Assign Menu > Joint/Point > Point Springs** command or the button to open the Assign Springs form and assign point springs that are oriented in the global axes directions to point objects. Both translational and rotational springs can be assigned to a point object.

The following two areas appear in the Assign Springs form:

Tip:

Make sure that the point objects with spring assignments are connected, either directly or indirectly, to structural elements such as frames, shell and links.

- **Spring Stiffness in Global Directions:** Specify the spring stiffness for one to six of the degrees of freedom for the selected point objects. Note that point spring stiffnesses are always specified in the global coordinate system. There is no coupling of the six springs specified in this form.

- **Options:** The following three assignment options are available:

  - ✓ **Add to Existing Springs:** Adds the specified spring stiffness to the point object. If one or more point spring assignments have already been made, this option increases the existing spring stiffness, assuming that you are specifying a positive spring stiffness.

  - ✓ **Replace Existing Springs:** Replaces the currently specified spring stiffness, if any, with the new spring stiffness assignment. If there is not an existing assignment, the new assignment is still made. This is the default option.

- ✓ **Delete Existing Springs:** Deletes any and all point spring assignments made to the selected point object(s).

Important Note: It is possible to assign negative spring stiffness to a point object as long as the total stiffness at the point still remains positive (or is zero). If you decide to assign some negative spring stiffness to a point object, do it with great care because it can terminate your analysis. Negative spring stiffness at a point during the analysis causes your structure to be unstable and thus the program terminates the analysis and provides an error message that there is an instability. The program does not check for negative spring stiffness before running the analysis.

Coupled Springs

There is no coupling of the spring stiffnesses that are specified in the Spring Stiffness in Global Directions area of the form. That is, for the spring stiffnesses specified in the Spring Stiffness in Global Directions area, the deformation in one degree of freedom does not affect the deformation in another degree of freedom. It is also possible to specify point springs that have coupled behavior. The spring forces that act on a point object are related to the displacements of that point object by a 6x6 symmetric matrix of spring stiffness coefficients. Specify this 6x6 matrix by clicking the **Assign Menu > Joint/Point > Point Springs** command and then clicking the **Advanced** button on the Assign Springs form to bring up the Coupled 6x6 Spring form. In that form, define the 6x6 matrix for the coupled springs.

Figure 10-2 illustrates the 6x6 symmetric matrix of spring stiffness coefficients. When coupling is present, all 21 terms in the upper triangle of the matrix are specified. The other 15 terms are known by symmetry. For springs that do not couple the degrees of freedom, only the 6 diagonal terms are needed because the off-diagonal terms are all zero. You are specifying the diagonal terms for the spring stiffnesses when you use the **Assign Menu > Joint/Point > Point Springs** command and assign the stiffnesses in the Spring Stiffness in Global Directions area of the form rather than clicking on the **Advanced** button.

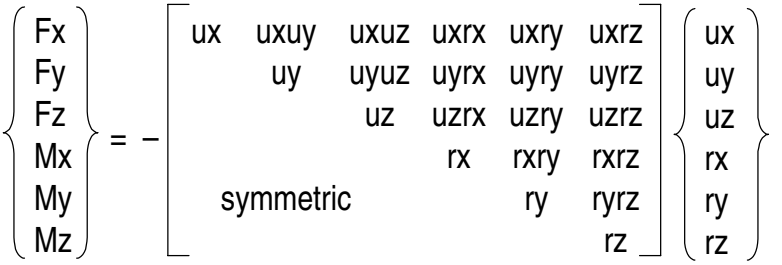


Figure 10-2: *6x6 symmetric matrix of spring stiffness coefficients*

10

Joint/Point > Link Properties Command

When link element properties are assigned to a point object, that link element is grounded. That is, one end is connected to the point object and the other end is connected to the ground. The element has zero length and no additional point is required to connect it to the ground. The local axes for the grounded, zero-length link element are as follows:

- The positive local 1-axis is up, in the same direction as the positive global Z-axis.
- The positive local 2-axis is in the same direction as the positive global X-axis.
- The positive local 3-axis is in the same direction as the positive global Y-axis.

You cannot modify the local axes directions for grounded, zero-length link elements.

Tip:

Use the ***Assign menu > Joint/Point > Panel Zone*** *command to assign panel zones with link properties to point objects.*

Use the **Assign menu > Joint/Point > Link Properties** command to assign link properties to a point object. This command brings up the Assign Link Properties form. Simply highlight the name of a defined link property in the form and then click the **OK** button to assign a link property to the selected point object(s). Note that link properties are defined using the **Define menu > Link Properties** command; see Chapter 7 Define Menu for more information about defining link properties.

Note:

You can assign multiple link properties (elements) to the same point object.

If you want to remove a link property assignment from a point object, select the point object, click the **Assign menu > Joint/Point > Link Properties** command, highlight "None" in the Link Properties area of the Assign Link Properties form and click the **OK** button.

Note you cannot use the **Assign menu > Joint/Point > Link Properties** command to assign panel zones to point objects even if the properties of the panel zone are based on a specified link property. You must use the **Assign menu > Joint/Point > Panel Zone** command to assign panel zones to point objects. See the previous subsection in this chapter titled "Joint Point > Panel Zone Command" for more information.

Joint Point > Additional Point Mass Command

Use the **Assign menu > Joint/Point > Additional Point Mass** command to assign additional point mass to a point object. Note that the additional point mass is only considered by the program if you have specified that the mass source is to be based on element masses and additional masses, not from a specified load combination. See the section entitled "Mass Source Command" in Chapter 7 Define Menu for more information.

Clicking the **Assign menu > Joint/Point > Additional Point Mass** command or the button brings up the Assign Masses form. Following are descriptions of the three areas in this form.

- **Masses in Global Directions:** Specify the translational masses in the global coordinate system direction in this area. The masses are entered in Force-Second^2/Length units.

- **Mom. of Inertia in Global Directions:** Specify the rotational mass moments of inertia about the global axes in this area. The rotational mass moments of inertia are entered in Force-Length-Second^2 units. Figure 10-3 is provided to assist you in calculating rotational mass moments of inertia for various shaped areas.

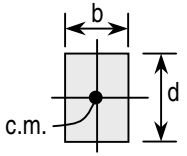
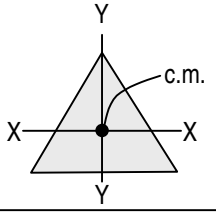
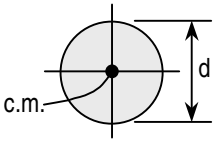
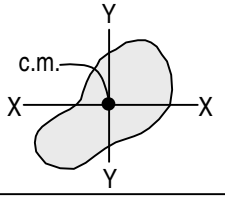
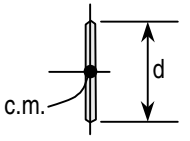
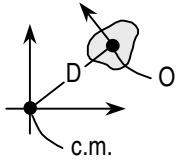
Shape in plan	Mass moment of inertia about vertical axis (normal to paper) through center of mass	Formula
	Rectangular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or W/g)	$MMI_{CM} = \frac{M(b^2 + d^2)}{12}$
	Triangular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or W/g)	Use general diaphragm formula
	Circular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or W/g)	$MMI_{CM} = \frac{Md^2}{8}$
	General diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or W/g) Area of diaphragm = A Moment of inertia of area about X-X = I_X Moment of inertia of area about Y-Y = I_Y	$MMI_{CM} = \frac{M(I_X + I_Y)}{A}$
	Line mass: Uniformly distributed mass per unit length Total mass of line = M (or W/g)	$MMI_{CM} = \frac{Md^2}{12}$
	Axis transformation for a mass: If mass is a point mass, $MMI_O = 0$	$MMI_{CM} = MMI_O + MD^2$

Figure 10-3:
Mass moment of inertia for various areas

- **Options:** The following three assignment options are available:

  - ✓ **Add to Existing Masses:** Adds the specified mass to the point object. If one or more mass assignments have already been made, this option increases the existing mass, assuming that you are specifying a positive mass.

  - ✓ **Replace Existing Masses:** Replaces the currently specified mass, if any, with the new mass assignment. If there is not an existing assignment, the new assignment is still made. This is the default option.

  - ✓ **Delete existing masses:** Deletes any and all mass assignments made to the selected point object(s).

Note that if you select the Include only Lateral Mass option when defining the mass source (you do this on the Define Mass form, which is accessed using the **Define menu > Mass Source command**), only the Direction X mass, Direction Y mass and the Rotation about Z moment of inertia are considered in the analysis.

Important Note: It is possible to assign negative mass to a point object as long as the total mass tributary to the point object still remains positive (or is zero). If you decide to assign some negative mass to a point object, do it with great care because it can terminate your analysis. If the program detects negative mass at a point during the analysis, it will terminate the analysis and provide an error message about negative mass. The program does not check for negative mass before running the analysis.

Frame/Line Commands

Use the **Assign menu > Frame/Line** command to make assignments to line objects. The following subsections describe the assignments that you can make to line objects.

Frame/Line > Frame Section Command

Shortcut:

You can use the ***Assign menu > Frame/Line > Frame Section*** *command to simultaneously define frame sections and assign them to selected line objects.*

Use the **Assign menu > Frame/Line > Frame Section** command or the button to open the Assign Frame Properties form and assign frame section properties to line objects. To use this command, select some line objects, then click the menu command to open the Assign Frame Properties form, highlight a frame section in the Properties area of the form and click the **OK** button to make the assignment.

The frame section property that you highlight in the Properties area can be a previously defined property or you can define it while you are in the Assign Frame Properties form. For defining frame section properties, the Assign Frame Properties form has all of the functionality that the Define Frame Properties form has. See the section titled "Frame Sections Command" in Chapter 7 Define Menu for more information.

Frame/Line > Frame Releases/Partial Fixity Command

Note:

Variable end releases that can supply from 0% to 100% fixity can be specified as spring stiffnesses to model different fixity conditions at the ends of frame elements.

You can release any of the three translational and three rotational degrees of freedom at either end of a line object. However, those releases are meaningful only if a frame section property is assigned to the line object. It is possible to specify partial fixity at the ends of the line object. This is performed by specifying a spring stiffness when you assign the member end release.

The releases are always specified in the line object (frame section) local coordinate system. End releases are always assumed to occur at the support faces; that is, at the inside end of the end offsets.

Use the **Assign menu > Frame/Line > Frame Releases/Partial Fixity** command or button to open the Assign Frame Releases form and assign frame releases to line objects. To use this command select some line objects, then click the menu command to open the form and do one of the following:

- Specify the desired end releases by checking the appropriate check boxes. Alternatively you can specify partial fixity by

entering a spring stiffness value for the frame partial fixity springs. Then click the **OK** button.

- If you want to remove all of the currently specified end releases, including partial fixity, from a member, check the No Releases check box at the bottom of the form and click the **OK** button.

Unstable End Releases

Any combination of end releases may be specified for a frame element, provided that the element remains stable. This ensures that all load applied to the element is transferred to the rest of the structure. The following sets of releases are unstable, either alone or in combination, and are not permitted. The program checks for these conditions when you click the **OK** button in the Assign Frame Releases form, and provides a message if unstable releases have been specified:

- Releasing Axial (U1) at both ends.
- Releasing Shear Force 2, Major (U2) at both ends.
- Releasing Shear Force 3, Minor (U3) at both ends.
- Releasing Torsion (R1) at both ends.
- Releasing Moment 22, Minor (R2) at both ends *and* Shear Force 3, Minor (U3) at either end.
- Releasing R3 (moment 33, major) at both ends *and* U2 (shear force 3, major) at either end.

Frame/Line > End (Length) Offsets Command

In this program, frame section properties are assigned to line objects. However, actual structural members have finite cross sectional dimensions. When two members, such as a beam and column, are connected at a point, there is some overlap of the cross sections. In many structures, the dimensions of the mem-

bers are large and the length of the overlap can be a significant fraction of the total length of the frame element. The program provides the capability of defining end offsets along the length of frame members to account for those finite dimensions of structural elements. See the subsequent subsection entitled "End Offsets Along the Length of Frame Elements" for more information.

Use the **Assign menu > Frame/Line > End (Length) Offsets** command or the button to open the Frame End Length Offsets form where you can define the end offsets along the length of frame elements.

Any end offset assigned to a line object is ignored unless the line object also has a frame section assigned to it.

End (Length) Offsets

Note:

The rigid zone factor for end offsets along the length of a frame element only applies to bending and shear deformations. It does not apply to axial and torsional deformations.

End offsets along the length of frame members are defined in the End Offset Along Length area of the Frame End Length Offsets form. Use the **Assign menu > Frame/Line > End (Length) Offsets** command or the button to open this form.

In the End Offset Along Length area you have the choice of having the program determine the end offset lengths automatically or specifying them yourself. You also can specify the rigid-zone factor. Descriptions of these items follow.

Automatically Calculated End Offset Lengths

The program automatically calculates offset lengths for beam- and column-type frame elements when the Automatic from Connectivity option is selected on the Frame End Length Offsets form. It assumes the offset length for all brace-type frame elements to be zero. (You can define your own non-zero offset lengths for brace elements if necessary.) Also, the dimensions of brace elements that frame into the ends of column and beam elements are not considered when calculating the end offset dimension for a column or a beam.

Note:

The program reports output forces at the inside face of end length offsets.

When the program automatically calculates the end offsets along the length of a beam, it bases the end offset length at an end of the beam on the maximum section dimensions of all columns that connect to that end of the beam. Similarly, when the program automatically calculates the end offsets along the length of a column, it bases the end offset length at an end of the column on the maximum section dimensions of all beams that connect to that end of the column.

Note the following about the program's automatically calculated end offsets along the length of frame members:

- When more than one beam frames into a column, the program bases the end offset in the column on the deeper beam.
- End offsets in beams are controlled by the size of the column below. The column above is not considered.

Rigid-Zone Factor

The rigid-zone factor specifies the fraction of each end offset assumed to be rigid for bending and shear deformations. When a fraction of the end offset is specified as rigid, the outside portion of the end offset is assumed to be rigid; that is, the portion at the end of the frame member. By default, the program assumes the rigid end factor to be zero; that is, the end offsets are fully flexible and they have the same frame section properties as those assigned to the rest of the member.

The rigid zones of the end offsets never affect axial and torsional deformations. The full element length is always assumed to be flexible for those deformations.

Output forces for the end of a frame member are provided at the inside face of the end offset along the length of the member. No output forces are produced within the end offset.

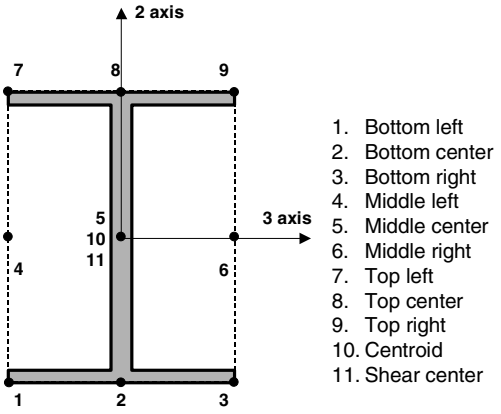
Frame/Line > Insertion Point Command

Note:

When you specify Insertion Point joint offsets, the local axes of the member are always based on the final position of the member after the end joint offsets have been applied.

When a line object is used to model a frame section, the line object is assumed to be located at the centroid of the column section and the top center of beam sections. Thus, when line objects (frame section) intersect in a model, it means that the centroids of the columns and the top center points of the beams intersect. In a real structure, this in not always the case. For example, it is not unusual for one or more floor beams in a building to frame eccentrically into a column. The program provides the capability to define Cardinal Points at different locations on the member and to offset the Cardinal Points, where required.

Figure 10-4 defines the Cardinal Point Options available and Figure 10-5 shows the default Cardinal Point location assumed by the program.



Note: For doubly symmetric members such as this one, cardinal points 5, 10, and 11 are the same.

Figure 10-4: *Cardinal Point options*

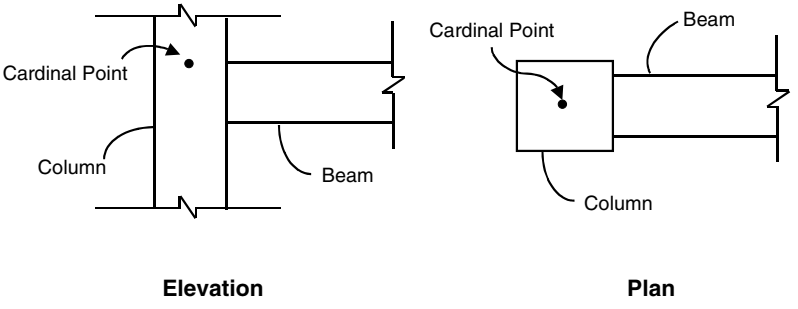


Figure 10-5: *Program default Cardinal Points*

Frame joint offsets from Cardinal Points are defined on the Frame Insertion Point form. Use the **Assign menu > Frame/Line > Insertion Point** command to open this form. In the Frame Joint Offsets from Cardinal Point area, specify the global X, Y and Z joint offsets at each end point of the frame element.

Note:

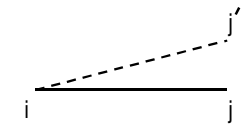
The locations of loads assigned to the line object are based on the final length and location of the member after the joint offsets have been applied.

This feature is useful for modeling beams and columns when the beams do not frame into the center of the column. Frame member joint offsets are always fully rigid.

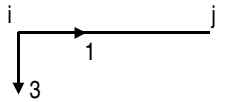
Figure 10-6 shows an elevation and plan view of a common framing arrangement where the exterior beams are offset from the column center lines to be flush with the exterior of the building. Also shown in this figure are the Cardinal Points for each member and the offset dimensions.

Figure 10-7 shows the Frame Insertion Point forms filled in for each member shown in Figure 10-6.

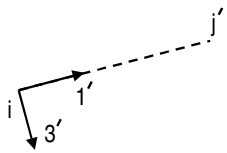
Important Note: When you specify member joint offsets, the local axes of the member are always based on the final position of the member after the joint offsets have been applied. Similarly, the location of loads assigned to the line object are based on the final length and location of the member after the joint offsets have been applied.



a) Beam with Joint Offset



b) Original Position of Beam



c) Final Position of Beam

Consider the example sketch shown to the left. Sketch a) shows a plan view of a beam that has the j-end joint offset. The end joint is offset such that the beam extends from i to j' rather than from i to j.

Sketch b) shows the local axes for the beam when it is in its *original* position *without* the joint offset. Sketch c) shows the local axes for the beam when it is in its *final* position *with* the joint offset. In sketches b) and c), the local 2-axis points upward and thus does not show in the plan view sketches. The program bases the local axes of the beam on those shown in sketch c).

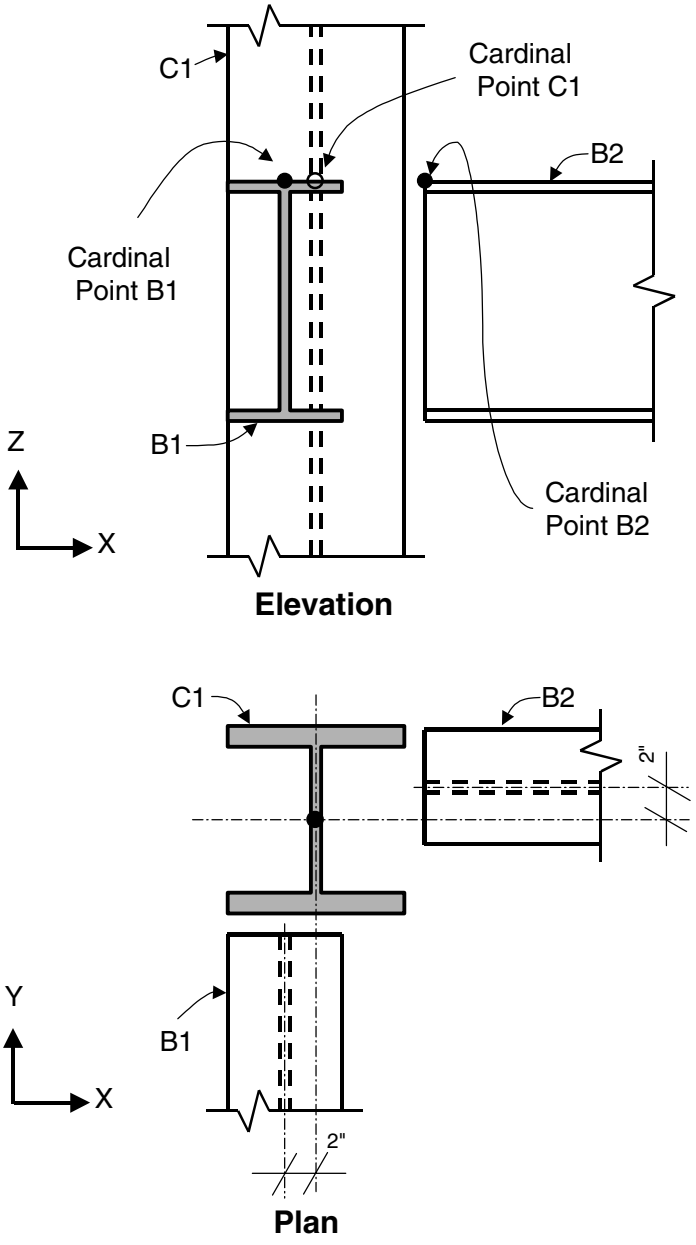
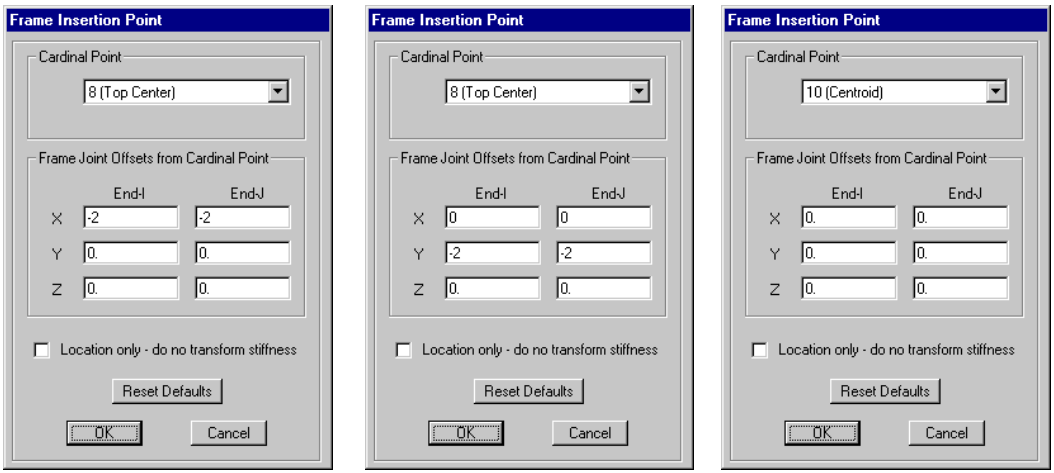


Figure 10-6: *Elevation and plan views of a common framing arrangement*



B1 B2 C1

Figure 10-7:
Frame Insertion Point form associated with Figure 10-6

10

Note:

When frame output stations are assigned to a line object, a text value is displayed on the line object. If the text value is reported in parenthesis, it is the minimum number of output stations. If it is not, it is the maximum spacing between output stations.

Frame/Line > Frame Output Stations Command

Frame output stations are designated locations along a frame element. They are used as locations to report output force and perform design and as plotting points for graphic display of force diagrams. When force diagrams are plotted, exact forces are plotted at each output station and then those points are connected by straight lines.

Important note: Output stations occur at user-specified locations *and* at point load locations along a beam.

Use the **Assign menu > Frame/Line > Frame Output Stations** command or the button to designate the output stations for a frame element. Two options are available for defining output stations for a beam on the Assign Output Station Spacing form:

- **Max Station Spacing:** Specifies the maximum spacing between stations along the beam. With this option, the program will first provide an output station at each point load location. Then it will provide an equally spaced number of stations between each adjacent pair of point loads where the spacing does not exceed the specified maximum spacing.

  Note that the output station spacing between one set of point loads may be different than that between another set.

- **Min Number Stations:** Specifies the minimum number of output stations along the beam. With this option, the program will first equally space the specified number of stations within the clear length of the beam. Then a station is added for each point load that does not fall at one of the previously defined output station locations.

  The minimum allowed number of equally spaced stations is three. This provides a station at each end of the beam and one at the center of the clear length. If there are end offsets specified for the beam, the stations at the end of the beam occur at the face of the end offset, not at the center of the support.

Note:

*Use the **View menu > Set Building View Options** command to toggle the display of frame element output stations on and off.*

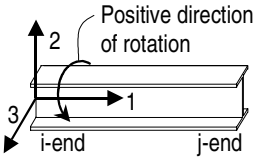
Note that the program determines the location of output stations in a different order, depending on whether you specify a minimum number of stations or a maximum spacing of stations.

By default for beams, output stations are provided at a maximum spacing of 2 feet for English units and 0.5 meter for metric units, and at all point load locations. By default, a minimum of three output stations are specified for columns and braces (the two ends and the middle).

Frame/Line > Local Axes Command

By default, the local 1-axis of a line object extends from the i-end of the element to the j-end. The default orientation of the local 2- and 3-axes depends on the frame-type (column, beam or brace) and in some instances, the orientation of the frame element itself.

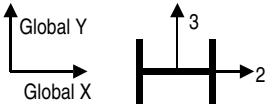
You can redefine the orientation of the local 2- and 3-axes of a line object by rotating them about the local 1-axis. To do this, select the line object and use the **Assign menu > Frame/Line > Local Axes** command or the button to bring up the Axis Orientation form. There are four options in this form:



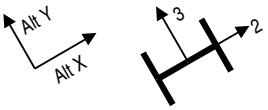
- **Angle:** Rotates the local 2-axis by the specified angle (in degrees) from its default position. When the 1-axis is pointing towards you, a positive rotation is counterclockwise; that is, the right hand rule applies.

- **Rotate by Angle:** Rotates the local 2-axis by the specified angle (in degrees) *from its current location* (not necessarily its default position). When the 1-axis is pointing towards you, a positive rotation is counterclockwise; that is, the right hand rule applies.

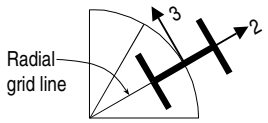
- **Column major direction is X (or Radial):** This option has no effect unless the selected element is a column. It sets the column major direction (the local 2-axis) as follows:



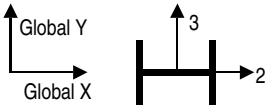
✓ If the column is at the intersection of two global coordinate system grid lines, the major direction (local 2-axis) is the same as the positive global X-axis.



✓ If the column is at the intersection of two grid lines from an additional rectangular coordinate system, the major direction (local 2-axis) is the same as the positive X-axis of that additional coordinate system.



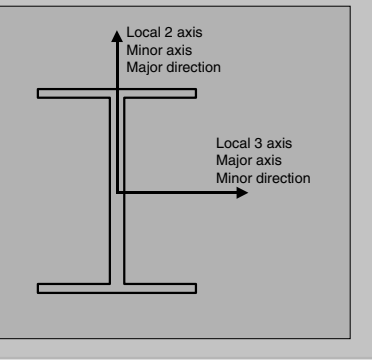
✓ If the column is at the intersection of two grid lines from an additional cylindrical coordinate system, the major direction (local 2-axis) is in the outward radial direction of that additional coordinate system.



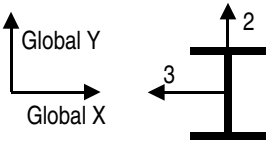
✓ If the column is not at the intersection of two grid lines from the same coordinate system, the major direction (local 2-axis) is the same as the positive global X-axis.

Column Major Direction

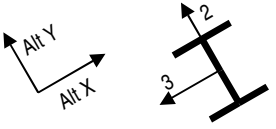
The column major direction is the same as the local 2-axis direction (which is also the same as the minor axis). Loads acting in the major direction cause M3 bending and V2 shear. In a wide flange member this corresponds to bending resisted by the flanges and shear resisted by the web.



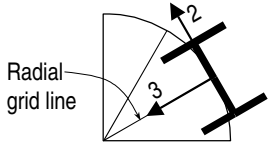
- **Column major direction is Y (or Tangential):** This option has no effect unless the selected element is a column. It sets the column major direction (the local 2-axis) as follows:



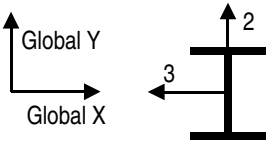
✓ If the column is at the intersection of two global coordinate system grid lines, the major direction (local 2-axis) is the same as the positive global Y-axis.



✓ If the column is at the intersection of two grid lines from an additional rectangular coordinate system, the major direction (local 2-axis) is the same as the positive Y-axis of that additional coordinate system.



✓ If the column is at the intersection of two grid lines from an additional cylindrical coordinate system, the major direction (local 2-axis) is in the tangential direction of that additional coordinate system pointing counterclockwise.



✓ If the column is not at the intersection of two grid lines from the same coordinate system, the major direction (local 2-axis) is the same as the positive global Y-axis.

Frame/Line > Frame Property Modifiers Command

Use the **Assign menu > Frame/Line > Frame Property Modifiers** command to bring up the Analysis Property Modification

Factors form. Here you can specify modification factors for the following frame analysis section properties in your model.

Note:

The frame property modifiers only affect the analysis properties. They do not affect the design properties.

- Cross-section (axial) area
- Shear Area in 2 direction
- Shear Area in 3 direction
- Torsional Constant
- Moment of Inertia about the 2-axis
- Moment of Inertia about the 3-axis

The modification factors are multiplied by the section properties specified for a frame element (see the **Define menu > Frame Sections** command) to obtain the final analysis section properties used for the frame element. Note that these modification factors only affect the analysis properties. They do not affect the design properties.

Frame/Line > Frame Line Type Command

Use the **Assign menu > Frame/Line > Frame Line Type** command to assign a frame line type to the selected line object. There are four line type options available. The selected line can be assigned as Beam, Column, Brace or Default. If the default option is selected, the line type will be that as defined using the **Define menu > Frame Sections** command (see the section entitled "Frame Sections Command" in Chapter 7 Define Menu).

Note:

You can assign multiple link properties (elements) to the same line object.

Frame/Line > Link Properties Command

Use the **Assign menu > Frame/Line > Link Properties** command to assign link properties to a line object. Select a line object(s) and use this command to bring up the Assign Link Properties form. On that form, highlight the name of a defined link property and click the **OK** button to assign a link property to the selected line object(s). Link properties are described in the sec-

tion entitled "Link Properties Command" in Chapter 7 Define Menu.

If you want to remove a link property assignment from a line object, select the line object, click the **Assign menu > Frame/Line > Link Properties** command, highlight "None" in the Link Properties area of the Assign Link Properties form, and click the **OK** button.

Frame/Line > Frame NonLinear Hinges Command

Use the **Assign menu > Frame/Line > Frame Nonlinear Hinges** command to bring up the Assign Frame Hinges (Push-over) form where you can assign nonlinear frame hinges (push-over) to line objects with frame section properties. Note that these hinge assignments are only used for static nonlinear analysis. They are not considered in a nonlinear time history analysis. Also note that nonlinear frame hinges are defined using the **Define menu > Frame Nonlinear Hinge Properties** command (see Chapter 7 Define Menu).

If you have selected a single element before implementing this command, the form shows you the currently assigned hinges, if any. If you have selected multiple elements before implementing this command, note the following:

Note:

You can assign multiple frame nonlinear hinges to the same line object. If desired, you can assign multiple hinges at the same location. This however may make it difficult for you to interpret some of the results.

- If all elements have the same hinge assignments, the form shows you those assignments.
- If the elements do not all have the same hinge assignments, the form is unfilled when it comes up.

A hinge assignment consists of a hinge property and a location for that hinge along the frame element. The location is specified as a relative distance along the *clear length* of the element, measured from the i-end. The relative distance is equal to the distance from the inside face of the end offset at the i-end of the element to the hinge location divided by the clear length of the frame element. Relative distances of 0, 0.5 and 1 specify hinges at the inside face of the end offset at the i-end, center of the clear

length and the inside face of the end offset at the j-end of a frame element, respectively.

To add a hinge assignment for the selected element(s), choose one of the defined hinge properties in the Hinge Property drop-down box, type in a distance in the relative distance box and click the **Add** button.

To modify an existing hinge assignment for the selected element(s), highlight the assignment in the Frame Hinge Data area. Note that the information for the highlighted assignment appears in the drop-down box and edit box at the top of the form. Modify the hinge property and relative distance as desired, and when finished, click the **Modify** button.

To delete an existing hinge assignment for the selected element(s), highlight the assignment in the Frame Hinge Data area. Note that the information for the highlighted assignment appears in the drop-down box and edit box at the top of the form. Click the **Delete** button.

When you have finished specifying the hinge property assignments, click the **OK** button to exit the form.

Frame/Line > Pier Label Command

A wall pier can consist of a combination of both area objects (shell elements) and line objects (frame elements). If you want to get output forces reported for wall piers, or if you want to design wall piers, you must first define them. You define a wall pier by selecting all of the line and/or area objects that make up the pier and assigning them the same pier label.

If a wall pier is made up of both line and area objects, assign the pier label to the line and area objects separately. For example, assume that a wall pier that is to be labeled P23 is made up of both line and area objects. You would first select the line objects and use the **Assign menu > Frame/Line > Pier Label** command to assign pier label P23 to the line objects. Then select the area objects and use the **Assign menu > Shell/Area > Pier Label**

command to assign pier label P23 to the area objects. See the Shear Wall Technical Notes for more information on wall pier labeling.

Typically, to assign a new pier label to a line object, you select the line object and click the **Assign menu > Frame/Line > Pier Label** command to enter the Pier Names form. On the form, highlight an existing pier name and click the **OK** button, or type a new pier name in the edit box in the Wall Piers area, click the **Add New Name** button and then click the **OK** button. When you highlight an existing pier name and click the **OK** button, the selected objects are added to the current objects that define the pier. The selected objects do *not* replace the current objects.

If you want to delete objects from a pier definition, select the objects to be deleted, enter the Pier Names form, highlight None and click the **OK** button.

You can click the **Assign menu > Frame/Line > Pier Label** command and enter the Pier Names form without first making a selection if you wish (regardless of whether the model is locked or unlocked). This is useful if you want to change a pier name or delete a pier definition. In that case, enter the Pier Names form without first making a selection, make the desired name changes or deletions and then click the **OK** button. Because you entered the form without first making a selection, the program knows not to make any assignment to the highlighted pier when you click the **OK** button. In the special case where you enter the form without a selection, whatever pier is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the form.

Frame/Line > Spandrel Label Command

A wall spandrel can be made up from a combination of both area objects (shell elements) and line objects (frame elements). If you want to get output forces reported for wall spandrels, or if you want to design wall spandrels, you must first define them. You define a wall spandrel by selecting all of the line and/or area ob-

jects that make up the spandrel and assigning them the same spandrel label.

If a wall spandrel is made up of both line and area objects, assign the spandrel label to the line and area objects separately. For example, assume that a wall spandrel that is to be labeled S23 is made up of both line and area objects. First select the line objects and use the **Assign menu > Frame/Line > Spandrel Label** command to assign spandrel label S23 to the line objects. Then select the area objects and use the **Assign menu > Shell/Area > Spandrel Label** command to assign spandrel label S23 to the area objects. See the Shear Wall Technical Notes for more information on wall spandrel labeling.

Typically, to assign a new spandrel label to a line object, select the line object and click the **Assign menu > Frame/Line > Spandrel Label** command to enter the Spandrel Names form. On the form, highlight an existing spandrel name and click the **OK** button, or type a new spandrel name in the edit box in the Wall Spandrels area, click the **Add New Name** button and then click the **OK** button. When you highlight an existing spandrel name and click the **OK** button, the selected objects are added to the current objects that define the spandrel. The selected objects do *not* replace the current objects.

If you want to delete objects from a spandrel definition, select the objects, enter the Spandrel Names form, highlight None and click the **OK** button.

You can click the **Assign menu > Frame/Line > Spandrel Label** command and enter the Spandrel Names form without first making a selection if you wish (regardless of whether the model is locked or unlocked). This is useful if you want to change a spandrel name or delete a spandrel definition. In that case, enter the Spandrel Names form without first making a selection, make the desired name changes or deletions and then click the **OK** button. Because you entered the form without first making a selection, the program knows not to make any assignment to the highlighted spandrel when you click the **OK** button. In the spe-

cial case where you enter the form without a selection, whatever spandrel is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the form.

Frame/Line > Line Springs Command

Tip:

The program distributes the springs associated with the line object to all of the nodes associated with the internal-to-the-program (analysis model) representation of the line object. If you are modeling a beam on elastic foundation with a line spring, you may want to mesh the line object yourself to ensure that a sufficient number of springs are used in the analysis model.

Use the **Assign menu > Frame/Line > Line Springs** command or the button to bring up the Assign Spring form where you can assign line springs to line objects. Line springs can be assigned in any of the local axes directions of the line object. Line springs are linear; that is, they support both tension and compression. You cannot define tension-only or compression-only line springs.

The program distributes the springs associated with the line object to all of the nodes associated with the internal-to-the-program (analysis model) representation of the line object. Note that internally, the program may mesh (break up) a line object into several elements with associated points between each element.

If you are modeling a beam on elastic foundation with a line spring, you may want to mesh the line object yourself to ensure that internally in the program a sufficient number of springs are used in the analysis model. The program will automatically determine the required stiffness for each spring. This saves you a considerable amount of time when the points where the springs actually occur are not uniformly spaced.

There are two areas in the Assign Spring form. They are:

- **Line spring:** Specifies the direction of the springs as one of the three local axes of the line object and specifies a stiffness for the spring. The units for the stiffness are Force/Length^2.
- **Options:** Three line spring assignment options are possible:
  - ✓ **Add to Existing Springs:** Adds the specified spring stiffness to the line object. If one or more spring stiffness assignments have already been made, this option increases

the existing spring stiffness, assuming that you specify a positive stiffness.

- ✓ **Replace Existing Springs:** Replaces the currently specified spring stiffness, if any, with the new spring stiffness. If there is not an existing assignment, the new assignment is still made. This is the default option.

- ✓ **Delete Existing Springs:** Deletes any and all spring stiffness assignments made to the selected line object(s). When this option is selected, the items in the Line Spring area of the form are ignored when you click the **OK** button.

  Note that the default option is Replace and that the program defaults to this every time the form is opened.

Important Note: It is possible to assign negative spring stiffness to a line object as long as the total stiffness at any point still remains positive (or zero). If you decide to assign some negative spring stiffness to a line object, do it with great care because it can terminate your analysis. If negative spring stiffness occurs at any point in your model during the analysis, the program terminates the analysis and provides an error message that there is an instability. The program does not check for negative spring stiffness before running the analysis.

Frame/Line > Additional Line Mass Command

Use the **Assign menu > Frame/Line > Additional Line Mass** command or the button to assign additional line mass to a line object. Note that the additional line mass is only considered by the program if you have specified that the mass source is to be based on element masses and additional masses, not from a specified load combination. See the section entitled "Mass Source Command" in Chapter 7 Define Menu for more information.

The additional line mass is only applied in the three translational degrees of freedom. If you have specified that only lateral mass

Tip:

Additional line mass is only considered by the program if the mass is specified to be determined from material property masses. Additional line mass is ignored if the mass is determined from a load combination.

is to be considered (see the section entitled "Mass Source Command" in Chapter 7 Define Menu), the additional line mass is only active in the global X and Y directions.

Clicking the **Assign menu > Frame/Line > Additional Line Mass** command or the button brings up the Assign Mass form. Following are descriptions of the two areas in that form.

- **Line Mass/Length:** Specifies the translational mass per unit length in this area. The masses are entered in Force-Second^2/Length^2 units.
- **Options:** Three line mass assignment options are possible:
  - ✓ **Add to Existing Masses:** Adds the specified line mass to the line object. If one or more line mass assignments have already been made, this option increases the existing line mass, assuming that you are specifying a positive mass.
  - ✓ **Replace Existing Masses:** Replaces the currently specified line mass, if any, with the new line mass. If there is not an existing assignment, the new assignment is still made. This is the default option.
  - ✓ **Delete Existing Masses:** Deletes any and all line mass assignments made to the selected line object(s).

  Note that the default option is Replace and that the program defaults to this every time the form is opened.

Frame/Line > Automatic Frame Subdivide Command

The program automatically subdivides frame elements as necessary in the analysis. In some cases, you may not want the program to automatically subdivide a frame element.

For example, when you have intersecting X-braces, the program would, by default, connect those braces at their intersection and divide each brace element into two pieces at the intersection

point. You may want to model it such that there is no connection between the braces where they cross.

In such a case, you can select the braces and use the **Assign menu > Frame/Line > Automatic Frame Subdivide** command to tell the program not to automatically subdivide them. You can also use this command again if you later decide that you want the program to automatically subdivide the braces. When you execute this command, you have three options:

- **Subdivide:** This tags the frame element to be automatically subdivide, as required, by the program. By default, all line objects have this tag when they are drawn.
- **Don't Subdivide:** This tags the frame element to *not* be automatically subdivide by the program.
- **Cancel:** This gives you a way to get out of the command without assigning a "subdivide it" or a "don't subdivide it" tag to the frame element.

Frame Line > Use Lines for Floor Meshing

The program automatically meshes area objects that are assigned deck properties or slab properties with membrane behavior only into the analysis model as necessary. There are several options available for auto meshing the area objects. One of those options is to mesh the area objects that are intersected by meshing lines or line objects with auto meshing option set to yes. In some cases, you may not want the program to automatically mesh an area object into the analysis model.

To tag a line object so that it is not used for auto meshing of area objects, select the line object and click the **Assign menu > Frame/Line > Use Line for Floor Meshing > No** command. You can also use this command again if you later decide that you want the program to use this line object to automatically mesh the area objects. When you execute this command, you have three options:

- **Yes:** This tags the line object to be used for automatic meshing of area objects intersected by the line object, as required, by the program into the analysis model. By default, all line objects have this tag when they are created.
- **Don't mesh it:** This tags the line object so that it is *not* used for the automatic meshing of area objects intersected by the line object in program analysis model.
- **Cancel:** This gives you a way to get out of the command without assigning one of the above options.

Shell/Area Commands

Use the **Assign menu > Shell/Area** command to make assignments to area objects. The following subsections describe the assignments that you can make to area objects.

Shell/Area > Wall/Slab/Deck Section Command

Shortcut:

*You can use the **Assign menu > Shell/Area > Wall/Slab/Deck Section** command to simultaneously **define** wall, slab and deck sections and **assign** them to selected area objects.*

Use the **Assign menu > Shell/Area > Wall/Slab/Deck Section** command or the button to open the Assign Wall/Slab/Deck Sections form and assign section properties to area objects. To use this command, select some area objects, then click the menu command or the button to open the form, highlight a wall, slab or deck section name in the Sections area of the form and click the **OK** button to make the assignment.

The wall, slab or deck section property that you highlight in the Sections area can either be a previously defined property (see Chapter 7 Define Menu) or you can define it while you are in this form. For defining wall, slab and deck section properties, the Wall/Slab/Deck Sections form has the same functionality as that of the Define Wall/Slab/Deck Sections form. See the section entitled "Wall/Slab/Deck Section Command" in Chapter 7 Define Menu for more information.

Important note concerning decks: When you assign deck section properties, the program assumes that the deck spans in the same direction as the local 1-axis of the area object to which the deck is assigned.

Shell/Area > Opening Command

Use the **Assign menu > Shell/Area > Opening** command or the button to bring up the Assign Openings form and designate a selected area object as an opening. In the Assign openings form, indicate that the area object is one of the following:

Tip:

You can assign ***unloaded*** *openings to area objects as you draw them by selecting the Openings option in the floating Properties of Object form (see Chapter 8 Draw Menue for more information).*

- **Not an Opening:** Use this to remove the designation of "opening" from an area object.
- **UnLoaded Opening:** Any loads applied to (or on) an unloaded-type opening are ignored by the program.
- **Loaded Opening:** The program considers all loads that are assigned to loaded-type openings.

The main purpose of designating area objects as openings is related to meshing. Both the automatic meshing of floors performed by the program and some of the manual meshing that you can do are based on openings.

A second purpose of designating area objects as openings is to allow area loads that are not directly supported by the structure to still be considered in an analysis. For example, if you are modeling a stair opening in a floor, you may want to consider the opening area as supporting some uniform dead and live load.

Shell/Area > Rigid Diaphragm Command

Use the **Assign menu > Shell/Area > Rigid Diaphragm** command or the button to designate a rigid diaphragm. *Rigid diaphragms can only be horizontal*. Thus, rigid diaphragm assignments are not applicable to wall-type and ramp-type area

objects. They are only applicable to floor type area objects and to null-type area objects that happen to be in a horizontal plane.

In this program, a rigid diaphragm translates within its own plane (global X-Y plane) and rotates about an axis perpendicular to its own plane (global Z-axis) as a rigid body. Designating an area object as a rigid diaphragm has no affect on the out-of-plane behavior of the area object. For example, if you specify a concrete floor slab to have plate-bending properties (i.e., out-of-plane bending capability), applying a rigid diaphragm constraint has no affect on the out-of-plane bending of the floor. It only effects in plane behavior of the floor.

Tip:

You can also assign rigid diaphragms to point objects using the ***Assign menu > Joint/Area > Rigid Point*** *command.*

Internal to this program, assigning a rigid diaphragm to an area object provides a diaphragm constraint to all of the corner points of the area object and to any additional point objects that are enclosed within the boundaries of the area object. This includes any points (joints) that are created as a result of the program's automatically meshing the area object.

Note:

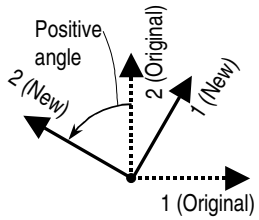
You can model the "actual" in-plane stiffness of a diaphragm by assigning slab properties to the floor and not specifying it as a rigid diaphragm. This is sometimes (and perhaps somewhat inappropriately) called a "flexible" diaphragm analysis.

When you select one or more area objects and click the **Assign menu > Shell/Area > Rigid Diaphragm** command or the button, the Assign Diaphragms form appears. Refer to the subsection entitled "Joint Point > Rigid Diaphragm Command" earlier in this chapter for a full description of this form.

Note that you can also apply a rigid diaphragm constraint directly to point objects. In most instances, it is better to assign the rigid diaphragm to an area object.

Shell/Area > Local Axes Command

By default, the local 3-axis of an area object is perpendicular to the plane of the area object. The local 1- and 2-axes lie in the plane of the object. The orientation of the 1- and 2-axes and the positive direction of the 3-axis depend on the type (orientation) of the area object.



You can rotate the area object local 1- and 2-axes about the local 3-axis. To do this, select an area object and use the **Assign menu > Shell/Area > Local Axes** command or the button to bring up the Assign Local Axis form. In this form, specify in degrees the angle from the default location of the local 2-axis (not necessarily the current location) to the new location of the local 2-axis. The angle is positive if it is counterclockwise when viewed from the positive local 3-axis side of the object.

Important Note: Do not confuse the local axes of area objects with those of pier and spandrel elements. They are different. You cannot rotate the local axes of pier and spandrel elements.

10

Shell/Area > Shell Stiffness Modifiers Command

Use the **Assign menu > Shell/Area > Shell Stiffness Modifiers** command to bring up the Analysis Stiffness Modification Factors form. Here you can specify Stiffness Modifiers for the following shell analysis section stiffnesses in your model.

Note:

The shell stiffness modifiers only affect the analysis properties. They do not affect any design properties.

- Membrane f11 Modifier
- Membrane f22 Modifier
- Membrane f12 Modifier
- Bending m11 Modifier
- Bending m22 Modifier
- Bending m12 Modifier

The stiffnesses for each of the items calculated based on the section properties specified for a shell element (see the **Define menu > Wall/Slab/Deck Sections** command) are multiplied by the specified modifiers to obtain the final stiffness used for the shell element in the analysis. Note that these modification factors only affect the analysis properties. They do not affect any design properties.

The f11, f22 and f12 modifiers are essentially equivalent to modification factors on the thickness of the shell element. The m11, m22 and m12 modifiers are essentially equivalent to modification factors on the thickness^3 of the shell element.

Shell/Area > Pier Label Command

A wall pier can consist of a combination of both area objects (shell elements) and line objects (frame elements). If you want to get output forces reported for wall piers, or if you want to design wall piers, you must first define them. Define a wall pier by selecting all of the line and/or area objects that make up the pier and assigning them the same pier label.

If a wall pier is made up of both line and area objects, assign the pier label to the line and area objects separately. For example, assume that a wall pier that is to be labeled P23 is made up of both line and area objects. First select the line objects and use the **Assign menu > Frame/Line > Pier Label** command to assign pier label P23 to the line objects. Then select the area objects and use the **Assign menu > Shell/Area > Pier Label** command or button to assign pier label P23 to the area objects. See the Shear Wall Technical Notes for more information on wall pier labeling.

Typically to assign a new pier label to an area object, select the area object and click the **Assign menu > Shell/Area > Pier Label** command or the button to enter the Pier Names form. On the form, highlight an existing pier name and click the **OK** button or type a new pier name in the edit box in the Wall Piers area, click the **Add New Name** button and then click the **OK** button. When you highlight an existing pier name and click the **OK** button, the selected objects are added to the current objects that define the pier. The selected objects do *not* replace the current objects.

If you want to delete objects from a pier definition, select the objects, enter the Pier Names form, highlight None, and click the **OK** button.

Shell/Area > Spandrel Label Command

A wall spandrel can consist of a combination of both area objects (shell elements) and line objects (frame elements). If you want to get output forces reported for wall spandrels, or if you want to design wall spandrels, you must first define them. Define a wall spandrel by selecting all of the line and/or area objects that make up the spandrel and assigning them the same spandrel label.

If a wall spandrel is made up of both line and area objects, assign the spandrel label to the line and area objects separately. For example, assume that a wall spandrel that is to be labeled S23 is made up of both line and area objects. First select the line objects and use the **Assign menu > Frame/Line > Spandrel Label** command to assign spandrel label S23 to the line objects. Then select the area objects and use the **Assign menu > Shell/Area > Spandrel Label** command or the button to assign spandrel label S23 to the area objects. See the Shear Wall Technical Notes for more information on wall spandrel labeling.

To assign a new spandrel label to an area object, select the area object and click the **Assign menu > Shell/Area > Spandrel Label** command or the button to enter the Spandrel Names form. In this form, highlight an existing spandrel name and click the **OK** button or type a new spandrel name in the edit box in the Wall Spandrels area, click the **Add New Name** button and then click the **OK** button. When you highlight an existing spandrel name and click the **OK** button, the selected objects are added to the current objects that define the spandrel. The selected objects do *not* replace the current objects.

If you want to delete objects from a spandrel definition, select the objects, enter the Spandrel Names form, highlight None, and click the **OK** button.

Shell/Area > Area Springs Command

Use the **Assign menu > Shell/Area > Area Springs** command of the button to bring up the Assign Spring form where you

can assign area springs to area objects. Area springs can be assigned in any of the local axes directions of the area object. Area springs are linear; that is, they support both tension and compression. You can not define tension-only or compression-only area springs.

Tip:

The program distributes the springs associated with the area object to all of the nodes associated with the internal-to-ETABS (analysis model) representation of the area object. If you are modeling a slab or mat on an elastic foundation with area springs you may want to mesh the area object yourself to assure that internally in ETABS a sufficient number of springs are used in the analysis model.

The program distributes the springs associated with the area object to all of the nodes associated with the internal-to-the-program (analysis model) representation of the area object. Note that in some cases, internally the program may mesh (break up) an area object into several elements with associated points between each element.

If you are modeling a slab or mat on an elastic foundation with area springs, you probably will want to mesh the area object yourself to ensure that internal to the program a sufficient number of springs are used in the analysis model. The program will automatically determine the required stiffness for each spring. This saves you a considerable amount of time when the points where the springs actually occur are not uniformly spaced.

There are two areas in the Assign Spring form. They are:

- **Area Spring:** Specifies the direction of the springs as one of the three local axes of the area object, and specifies a stiffness for the spring. The units for the stiffness are Force/Length^3.
- **Options:** Three area spring assignment options are possible:
  - ✓ **Add to Existing Springs:** Adds the specified spring stiffness to the area object. If one or more spring stiffness assignments have already been made, this option increases the existing spring stiffness, assuming that you are specifying a positive stiffness.
  - ✓ **Replace Existing Springs:** Replaces the currently specified spring stiffness, if any, with the new spring stiffness. If there is not an existing assignment, the new assignment is still made. This is the default option.

- ✓ **Delete Existing Springs:** Deletes any and all spring stiffness assignments made to the selected area object(s). When this option is selected, the Value item in the Area Spring area of the form is ignored when you click the **OK** button.

Note that the default option is Replace and that the program defaults to this every time the form is opened.

Important Note: It is possible to assign negative spring stiffness to an area object as long as the total stiffness at any point still remains positive (or zero). If you decide to assign some negative spring stiffness to an area object, do it with great care because it can terminate your analysis. If negative spring stiffness occurs at any point in your model during the analysis, the program terminates the analysis and provides an error message that there is an instability. The program does not check for negative spring stiffness before running the analysis.

Shell/Area > Additional Area Mass Command

Tip:

Additional area mass is only considered by the program if the mass is specified to be determined from material property masses. Additional area mass is ignored if the mass is determined from a load combination.

Use the **Assign menu > Shell/Area > Additional Area Mass** command or the button to assign additional area mass to an area object. Note that the additional area mass is only considered by the program if you have specified that the mass source is to be based on element masses and additional masses, not from a specified load combination. See the section entitled "Mass Source Command" in Chapter 7 for more information.

The additional area mass is only applied in the three translational degrees of freedom. If you have specified that only lateral mass is to be considered (see the section entitled "Mass Source Command" in Chapter 7), the additional area mass is only active in the global X and Y directions.

Clicking the **Assign menu > Shell/Area > Additional Area Mass** command or the button brings up the Assign Mass form. Following are descriptions of the two areas in this form.

- **Mass/Area:** Specifies the translational mass per unit area in this region. The masses are entered in Force-Second^2/Length^3 units.

- **Options:** Three area mass assignment options are possible:

  - ✓ **Add to Existing Masses:** Adds the specified area mass to the area object. If one or more area mass assignments have already been made, this option increases the existing area mass, assuming that you are specifying a positive mass.

  - ✓ **Replace Existing Masses:** Replaces the currently specified area mass, if any, with the new area mass. If there is not an existing assignment, the new assignment is still made. This is the default option.

  - ✓ **Delete Existing Masses:** Deletes any and all area mass assignments made to the selected area object(s) when you click the **OK** button.

  Note that the default option is Replace and that the program defaults to this every time the form is opened.

Load Commands

Use the **Assign menu** commands to make load assignments to point, line, and area objects. The following subsections describe the assignments that you can make to those objects.

Joint/Point Loads > Force Command

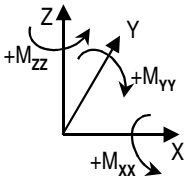
Use the **Assign menu > Joint/Point Loads > Force** command or the button to bring up the Point Forces form and assign point loads to selected point objects. Note that the point loads are specified in global coordinate system directions.

Assigning a force or moment to a point object is only meaningful if the point object is in one of the following locations:

- At the ends of structural line objects (beam, column, brace, link).
- At the corner points of structural area objects (floor, ramp, wall).
- Anywhere in the plane of a structural area object (floor, ramp, and wall). Note that in some cases, ramps may be slightly warped (four corners not coplanar) and thus it is difficult to impossible to tell if a point object actually lies in the plane of a ramp. Thus you should take great care in applying loads to point objects that are in the plane of ramps. We do not in general recommend that you apply point loads to ramps.
- Anywhere along the length of the line object with frame section properties (beam, column, and brace), unless the line object is tagged to not be automatically meshed. Note that the point object must lie exactly on the line object. We do not recommend that you attempt to apply point loads to frame elements in this manner. Instead, use the **Assign menu > Frame/Line Loads > Point** command to apply the point loads.

The following bullet items discuss three areas on the Point Forces form:

- **Load Case Name:** Select the name of a defined static load case that the specified loads are to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names.



- **Loads:** Input the point loads in the global coordinate system directions in this area. Positive directions of moments (shown in the sketch to the left) are based on the right hand rule.
- **Options:** The following three assignment options are available:
  - ✓ **Add to Existing Loads:** Adds the specified point loads to the point object. If one or more point load assignments

have already been made, this option increases the total point load on the point object, assuming that you are specifying a positive load.

- ✓ **Replace Existing Load:** Replaces the currently specified point load, if any, with the new point load assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the loads in the load case that is specified above are replaced.

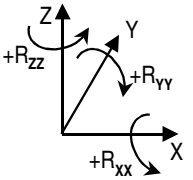
- ✓ **Delete existing loads:** Deletes any and all point load assignments made to the selected point object(s). Note that only the loads in the load case that is specified above are deleted.

Joint/Point Loads > Ground Displacement Command

Use the **Assign menu > Joint/Point Loads > Ground Displacement** command to bring up the Ground Displacements form and assign ground displacement loads to selected point objects. Note that the ground displacements are specified in global coordinate system directions. Please read the Important Information About Ground Displacement Assignments provided in the shadow box carefully.

The following bullet items describe three areas on the Ground Displacements form:

- **Load Case Name:** Select the name of a defined static load case that the specified displacements are to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names.



- **Displacements:** Input the displacements in the global coordinate system directions in this area. Positive directions of rotations (shown in the sketch to the left) are based on the right-hand rule.

- **Options:** The following three assignment options are available:

Important Information about Ground Displacement Assignments

Point object ground displacements are only meaningful when they are applied to point objects that are connected to the ground in the direction that the displacement is applied. Point objects are connected to the ground through one of the following:

- Restraints
- Springs
- Grounded link elements assigned to a single point object

When ground displacements are assigned to a point object that is restrained, the displacement takes place at the point object.

When ground displacements are assigned to point objects that have springs or grounded link elements assigned to them, the displacement takes place at the grounded end of the spring or link, not at the point object. **This is a subtle but very important distinction.**

If you apply a ground displacement to a point object that is not connected to the ground through a restraint, spring or grounded link element, that displacement is ignored by the program when the analysis is run.

- ✓ **Add to existing loads:** Adds the specified displacements to the point object. If one or more displacement assignments have already been made then this option increases the total displacement assigned to the point object assuming, of course, you are specifying a positive displacement.

- ✓ **Replace existing load:** Replaces the currently specified displacement, if any, with the new displacement assignment. If there is not an existing assignment then the new assignment is still made. This is the default option. Note that only the loads in the load case that is specified above are replaced.

- ✓ **Delete existing loads:** Deletes any and all displacement assignments made to the selected point object(s). When this option is selected the items in the Loads area of the form are ignored when you click the **OK** button. Note that

only the loads in the load case that is specified above are deleted.

Note that the default option is Replace and that the program defaults to this every time the form is opened.

Joint/Point Loads > Temperature Command

Use the **Assign menu > Joint/Point Loads > Temperature** command or the button to bring up the Point Temperatures form and assign a point temperature change to selected point objects. This temperature change in itself is not a temperature load.

Note:

The purpose of applying temperature changes to point objects is to allow you to specify that temperature changes in area and/or line objects (shell and/or frame elements) are to be determined from the temperature changes specified at the corner or end points of the elements.

Temperature loads actually act on area and line objects (shell and frame elements). One of the options available when you specify a temperature load on an area object (shell element) is that the value of the temperature load (change) is determined from previously specified point temperature changes at the points at the corners of the element. Similarly, one of the options available when you specify a temperature load on a line object (frame element) is that the value of the temperature load (change) is determined from previously specified point temperature changes at the points at the ends of the element.

Thus, the purpose of applying temperature changes to point objects is to allow you to specify that temperature changes in area and/or line objects (shell and/or frame elements) are to be determined from the temperature changes specified at the corner or end points of the elements. When you apply a temperature change directly to a shell or frame element, that temperature change is uniform throughout the element. Applying the temperature change based on the points allows you to have temperature changes that vary linearly along the length of frame elements and vary linearly over the surface area of shell elements. A positive temperature change corresponds to an increase in the temperature of an object.

The following bullet items describe three areas in the Point Temperatures form:

- **Load Case Name:** Select the name of a defined static load case that the specified temperature changes are to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names.

- **Temperature:** Specify the temperature change in this area. If you are working in English units, the temperature is specified in degrees Fahrenheit, °F. If you are working in metric units, the temperature is specified in degrees centigrade, °C.

- **Options:** The following three assignment options are available:

  - ✓ **Add to Existing Values:** Adds the specified temperature changes to the point object. If one or more temperature change assignments have already been made, this option increases the total temperature change on the point object, assuming that you are specifying a positive temperature change.

  - ✓ **Replace Existing Values:** Replaces the currently specified temperature change, if any, with the new temperature change assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the loads in the load case that is specified on this form are replaced.

Delete existing loads: Deletes any and all temperature change assignments made to the selected point object(s).

Frame/Line Loads > Point Command

Use the **Assign menu > Frame/Line Loads > Point** command or the button to bring up the Frame Point Loads form and assign point loads to selected line objects. The following bullet items describe the four areas on the Frame Point Loads form:

Note:

The positive directions for point moments are determined using the right hand rule. Note that the positive direction for the moment in the gravity direction is determined by pointing your right thumb in the Gravity (negative global Z) direction and applying the right hand rule.

- **Load Case Name:** Select the name of a defined static load case that the specified point loads are to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names (see Chapter 7 Define Menu).

- **Load Type and Direction:** Specifies loads as Forces or Moments. Also specifies the direction of the load. The following directions are possible:

  ✓ Local-1

  ✓ Local-2

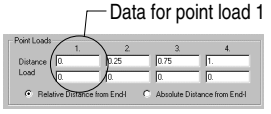
  ✓ Local-3

  ✓ Global-X

  ✓ Global-Y

  ✓ Gravity

  Note that the Gravity direction is downward in the negative global Z direction. Defining the direction as Gravity rather than Global-Z allows you to put in your gravity loads with positive signs (or more likely, no sign) rather than negative signs.



- **Point Loads:** Here you can specify up to four point loads acting on the line object (frame element) by indicating a location and a load for the point load. The data for the first point load is input in the first set of Distance and Load boxes (see sketch to the left), the data for the second point load is entered in the second set of Distance and Load boxes, and so on.

  The Distance to the point load is always measured from the i-end of the line object. You have the option to specify Relative Distance from End-I or Absolute Distance from End-I. The relative distance is equal to the distance from the left end of the line object to the point where the load intensity is specified divided by the length of the line object. The relative distance is never larger than 1.0. An absolute distance is the actual dis-

tance from the left end of the line object to the point where the load intensity is specified.

If you want to specify more than four point loads, simply specify the first four point loads and click the OK button to assign them, then reselect the line object and click the **Assign menu > Frame/Line Loads > Point** command or the button to again bring up the Frame Point Loads form and specify additional point loads.

- **Options:** The following three assignment options are available:

  - ✓ **Add to Existing Loads:** Adds the specified point loads to the line object. If one or more point load assignments have already been made at the same location on the line object, this option increases the total point load on the line object at that location, assuming that you are specifying a positive load.

  - ✓ **Replace Existing Loads:** Replaces the currently specified point load, if any, with the new point load assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the loads in the load case that is specified above are replaced.

  - ✓ **Delete Existing Loads:** Deletes any and all point load assignments made to the selected line object(s). When this option is selected, the items in the Point Loads areas of the form are ignored when you click the **OK** button. Note that only the loads in the load case that is specified in this form are deleted.

  Note that the default option is Replace and that the program defaults to this every time the form is opened.

Frame/Line Loads > Distributed Command

Use the **Assign menu > Frame/Line Loads > Distributed** command or the button to bring up the Frame Distributed

Note:

Distributed loads can be uniform or non-uniform (trapezoidal) and they can be full length or partial length.

Loads form and assign distributed loads to selected line objects. The distributed loads may be specified as uniform over the length of the line object or they may be specified as trapezoidal loads over any length of the line object. The following bullet items describe the four areas in the Frame Distributed Loads form:

- **Load Case Name:** Select the name of a defined static load case that the specified distributed load is to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names (see Chapter 7 Define Menu).

- **Load Type and Direction:** Specifies whether the loads are Forces (line loads) or Moments (line moments). Also specifies the direction of the load. The following directions are possible:

  - ✓ Local-1
  - ✓ Local-2
  - ✓ Local-3
  - ✓ Global-X
  - ✓ Global-Y
  - ✓ Gravity
  - ✓ Global-X Projection (only applicable to forces, not moments)
  - ✓ Global-Y Projection (only applicable to forces, not moments)
  - ✓ Gravity Projection (only applicable to forces, not moments)

Note:

The Gravity direction for loads is downward in the negative global Z direction

Note:

Only forces can be specified as projected loads, not moments.

Note that the Gravity direction is downward in the negative global Z direction. Defining the direction as Gravity rather than Global-Z allows you to put in your gravity loads with

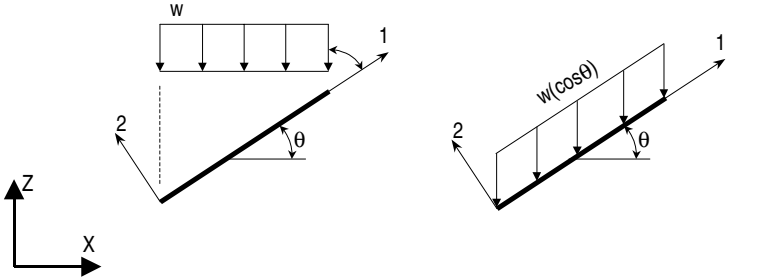
Note:

The positive directions for distributed moments are determined using the right hand rule. Note that the positive direction for the moment in the gravity direction is determined by pointing your right thumb in the Gravity (negative global Z) direction and applying the right hand rule.

positive signs (or more likely, no sign) rather than negative signs.

Also note that only forces can be specified as projected loads, not moments. Figure 10-8 shows an example of how the program considers projected loads on line objects. Figure 10-8a illustrates a projected uniform distributed load of intensity w. The direction of the load is the program gravity projection direction. Note that this is equivalent to a force of w(cosθ) acting along the entire length of the line object in the gravity direction, as shown in Figure 10-8b.

- **Trapezoidal Loads:** Specifies non-uniform distributed loads acting on a line object. The distributed loads can be specified over the full length of the line object or just over part of the length. Distributed load that you specify in this area, if any, is additive with that specified in the Uniform Load area.



a) Line load, w, applied to line object in Gravity projection direction

b) How projected line load is treated in ETABS

Figure 10-8: *Uniform load ,w, acting on a line object in the Gravity projection direction*

The loaded length for a trapezoidal load may be specified using Relative Distance from End-I or Absolute Distances from End-I. A relative distance is equal to the distance from the left end of the line object to the point where the load intensity is specified divided by the length of the line object. The relative distance is never larger than 1.0. An absolute distance is the actual distance from the left end of the line object to the point where the load intensity is specified.

Trapezoidal loads are defined by specifying up to four sets of distances and loads. The Distance and Load sets are specified at locations where the rate of change of the load intensity changes; that is, at the corners of the loading intensity diagram. Figure 10-9 shows some examples. All of the distances shown in Figure 10-9 are relative distances.

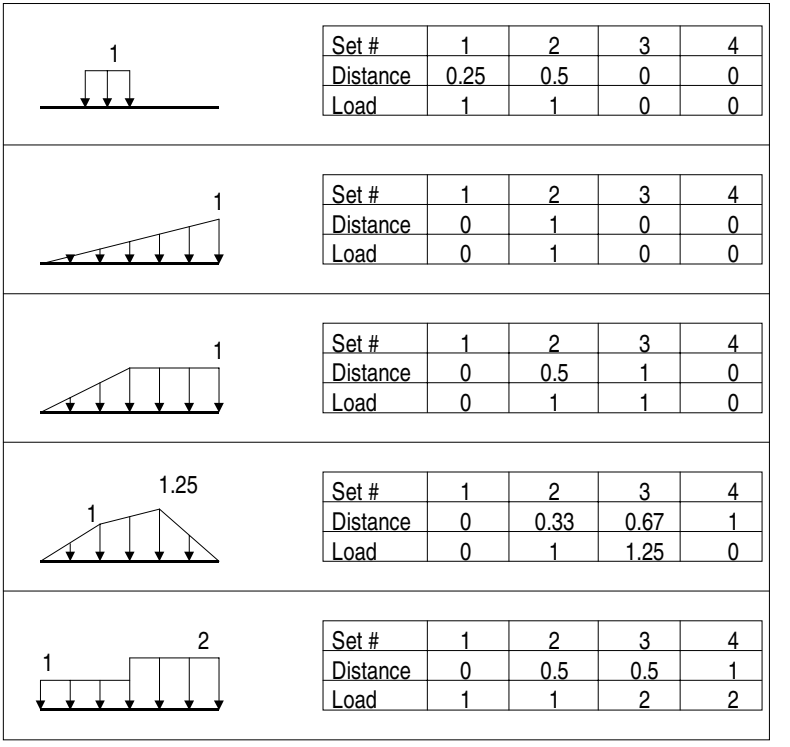


Figure 10-9: *Examples of trapezoidal loads*

10

Note:

Input partial length uniform loads as trapezoidal loads.

Specify the Distance and Load set closest to the i-end of the line object in box 1, the next set in box 2 and so on. For any sets of boxes that you do not use, set the distance to 0. The program ignores any boxes where the distance is smaller than the distance in the previous box.

- **Uniform Load:** Enter a uniform load value that applies over the entire length of the beam. Any load that is entered in this area is additive to any load specified in the Trapezoidal Loads area.

- **Options:** The following three assignment options are available:

  - ✓ **Add to Existing Loads:** Adds the specified point loads to the line object. If one or more point load assignments have already been made at the same location on the line object, this option increases the total point load on the point object at that location, assuming that you are specifying a positive load.

  - ✓ **Replace Existing Loads:** Replaces the currently specified point load, if any, with the new point load assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the loads in the load case that is specified above are replaced.

  - ✓ **Delete Existing Loads:** Deletes any and all point load assignments made to the selected line object(s). When this option is selected, the items in the Trapezoidal Loads and the Uniform Loads areas of the form are ignored when you click the **OK** button. Note that only the loads in the load case that is specified on this form are deleted.

Note:

In the Frame Distributed Loads form, trapezoidal and uniform load assignments are additive.

Note that the default option is Replace and that the program defaults to this every time the form is opened.

Frame/Line Loads > Temperature Command

Use the **Assign menu > Frame/Line Loads > Temperature** command or the button to bring up the Line Object Temperatures form and assign temperature loads to selected line objects. Note that temperature loads may be based on a uniform temperature change you specify for the object, or they may be based on previously specified point object temperature changes at the point objects at the ends of the line object, or they may be based on a combination of both. The following bullet items describe the four areas in the Line Object Temperatures form:

Note:

Temperature loads may be based on a uniform temperature change you specify for the object, previously specified point object temperature changes at the point objects at the ends of the line object, or a combination of both.

- **Load Case Name:** Select the name of a defined static load case that the specified line object temperature loading is to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names (see Chapter 7 Define Menu).

- **Object Temperature:** Specifies the uniform temperature change, if any, for the object. If you are basing the temperature load for the line object on the point temperatures at the end of the object only, enter 0 for the uniform temperature change. A positive temperature change corresponds to an increase in the temperature of an object.

- **End Point Temperature Option:** If you check the Include Effect of Point Temperatures check box in this area, the program considers the temperature change in the line object based on previously specified point object temperature changes at the point objects at the ends of the line object. The program assumes that the temperature change varies linearly along the length of the line object based on the specified changes at the end points.

  Checking this box has no affect on the uniform temperature change specified in the Object Temperature area. You can simultaneously specify a uniform temperature change and a temperature change based on specified end point temperatures if desired. Alternatively, and probably more commonly, you can specify one type of temperature change or the other. If you do not want to include the effect of point temperatures, leave the box unchecked.

  Note that the effect of the end point temperatures is not additive to itself. Either the end point temperatures are considered (check box is checked) or they are not (check box is unchecked). Thus, none of the three Object Temperature Options assignments (add, replace, or delete; described in the next bullet item) have an affect on this option.

Note:

When end point temperatures are specified to be included in the line object temperature load, the program assumes that the temperature change varies linearly along the length of the line object based on the specified changes at the end points.

- **Object Temperature Options:** It is very important to note that *these options only apply to the uniform temperature*

change in the Object Temperature area of the form. The following three assignment options are available:

- ✓ **Add to Existing Temperatures:** Adds the specified uniform temperature change to the line object. If one or more uniform temperature change assignments have already been made, this option increases the total uniform temperature change on the line object, assuming that you are specifying a positive uniform temperature change.

  This option has no affect on the end point temperature option. See the End Point Temperature Option bullet item for more information.

- ✓ **Replace Existing Temperature:** Replaces the currently specified uniform temperature change, if any, with the new uniform temperature change assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the temperature changes in the load case specified in this form are replaced.

  This option has no affect on the end point temperature option. See the End Point Temperature Option bullet item for more information.

- ✓ **Delete Existing Loads:** Deletes any and all uniform temperature change assignments made to the selected line object(s). Note that only the temperature changes in the load case that is specified on this form are deleted.

  This option has no affect on the end point temperature option. See the End Point Temperature Option bullet item for more information.

Note that the default option is Replace and that the program defaults to this every time the form is opened.

Shell/Area Loads > Uniform Command

Use the **Assign menu > Shell/Area Loads > Uniform** command to bring up the Uniform Surface Loads form and assign uniform loads to selected area objects. The following bullet items discuss the four areas in the Frame Distributed Loads form:

- **Load Case Name:** Select the name of a defined static load case that the specified uniform surface load is to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names (see Chapter 7 Define Menu).

- **Uniform Load:** Specifies the uniform load value and the direction of the load. The following directions are possible:

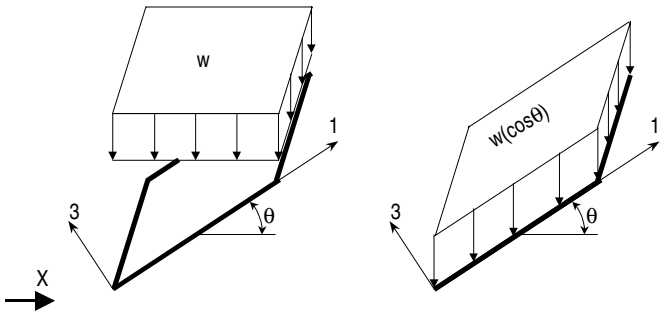
  - ✓ Local-1
  - ✓ Local-2
  - ✓ Local-3
  - ✓ Global-X
  - ✓ Global-Y
  - ✓ Gravity
  - ✓ Global-X projection
  - ✓ Global-Y projection
  - ✓ Gravity projection

Note:

The Gravity direction for loads is downward in the negative global Z direction

Note that the Gravity direction is downward in the negative global Z direction. Defining the direction as Gravity rather than Global-Z allows you to put in your gravity loads with positive signs (or more likely, no sign) rather than negative signs.

Figure 10-10 shows an example of how the program considers projected loads on area objects. Figure 10-10a illustrates a



a) Area load, w, applied to area object in Gravity projection direction

b) How projected area load is treated in ETABS

Figure 10-10: *Uniform surface load ,w, acting on an area object in the Gravity projection direction*

projected uniform surface load of intensity w. The direction of the load is the program gravity projection direction. Note that this is equivalent to a force of w(cosθ) acting over the entire surface of the area object in the gravity direction as shown in Figure 10-10b.

- **Options:** The following options are available:

  - ✓ **Add to Existing Loads:** Adds the specified uniform load to the area object. If one or more uniform load assignments have already been made to the area object, this option increases the total uniform load on the area object, assuming that you are specifying a positive load.

  - ✓ **Replace Existing Load:** Replaces the currently specified uniform load, if any, with the new uniform load assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the loads in the load case that is specified in this form are replaced.

  - ✓ **Delete Existing Loads:** Deletes any and all uniform load assignments made to the selected area object(s). When this option is selected, the items in the Uniform Load area of the form are ignored when you click the **OK** button. Note that only the loads in the load case that is specified in this form are deleted.

Note that the default option is Replace and that the program defaults to this every time the form is opened.

Shell/Area Loads > Temperature Command

Use the **Assign menu > Shell/Area Loads > Temperature** command to bring up the Area Object Temperatures form and assign temperature loads to selected area objects. Note that temperature loads may be based on a uniform temperature change you specify for the object, or they may be based on previously specified point object temperature changes at the point objects at the corners of the area object, or they may be based on a combination of both. The following bullet items discuss the four areas in the Area Object Temperatures form:

Note:

When end point temperatures are specified to be included in the area object temperature load, the program assumes that the temperature change varies linearly over the surface of the area object based on the specified changes at the corner points.

- **Load Case Name:** Select the name of a defined static load case that the specified area object temperature loading is to be assigned to. Note that you use the **Define menu > Static Load Cases** command to define load case names (see Chapter 7 Define Menu).

- **Object Temperature:** Specifies the uniform temperature change, if any, for the object. If you are basing the temperature load for the area object on the point temperatures at the corner points of the object only, enter 0 (zero) for the uniform temperature change. A positive temperature change corresponds to an increase in the temperature of an object.

- **Corner Point Temperature Option:** If you check the Include Effect of Point Temperatures check box in this area, the program considers the temperature change in the area object based on previously specified point object temperature changes at the point objects at the corners of the area object. The program assumes that the temperature change varies linearly over the surface of the area object based on the specified changes at the corner points.

  Checking this box has no affect on the uniform temperature change specified in the Object Temperature area. You can si-

multaneously specify a uniform temperature change and a temperature change based on specified corner point temperatures if desired. Alternatively, and probably more commonly, you can specify one type of temperature change or the other. If you do not want to include the effect of point temperatures, leave the box unchecked.

Note that the effect of the corner point temperatures is not additive to itself. The corner point temperatures are considered or they are not. You control this by either checking or unchecking the box. Thus the options in the Object Temperature Options area (add, replace, or delete) have no affect on the option to include the effect of the point temperatures.

Note:

The three Object Temperature Options have no affect on the corner point temperature option. The effect of the corner point temperatures is not additive to itself. Corner point temperatures are considered or they are not.

- **Object Temperature Options:** It is very important to note that *these options only apply to the uniform temperature change in the Object Temperature area of the form.* The following three assignment options are available:

  - ✓ **Add to Existing Temperatures:** Adds the specified uniform temperature change to the area object. If one or more uniform temperature change assignments have already been made, this option increases the total uniform temperature change on the area object, assuming that you are specifying a positive uniform temperature change.

    This option has no affect on the corner point temperature option. See the corner Point Temperature Option bullet item for more information.

  - ✓ **Replace Existing Temperature:** Replaces the currently specified uniform temperature change, if any, with the new uniform temperature change assignment. If there is not an existing assignment, the new assignment is still made. This is the default option. Note that only the temperature changes in the load case that is specified on this form are replaced.

This option has no affect on the corner point temperature option. See the Corner Point Temperature Option bullet item for more information.

- ✓ **Delete existing loads:** Deletes any and all uniform temperature change assignments made to the selected area object(s). When this option is selected, any value input in the Object Temperature area of the form for a uniform temperature change is ignored when you click the **OK** button. Note that only the temperature changes in the load case that is specified in this form are deleted.

  This option has no affect on the corner point temperature option. See the corner Point Temperature Option bullet item for more information.

Note that the default option is Replace and that the program defaults to this every time the form is opened.

Shell/Area Loads > Wind Pressure Coefficient

The **Assign menu > Shell/Area Loads > Wind Pressure Coefficient** command allows you to assign wind pressure coefficients to area objects. Those coefficients are used for automatic wind loads that have been specified to receive their loads from area objects. A static load case with an automatic wind load must be defined for this command to be enabled.

A positive coefficient, Cp, acts in the positive local 3-axis direction of the area object. When the Windward option is chosen, the wind pressure on the selected area objects is assumed to vary over the height of the building. When the Other option is chosen, the wind pressure on the selected area objects is assumed to be constant over the height of the building.

Group Names Command

To define a group, first select the objects that you want to be part of the group. Then click the **Assign menu > Group Names**

Note:

*If the name of the group appearing in the edit box in the Groups area of the Assign Groups form does not match any of the group names listed in that area, the **OK** button is not active until you click the **Add New Group** button to add that group name to the list of groups.*

command or the button to bring up the Assign Group form. Either highlight an existing group name in the form and click the **OK** button or create a new group name, click the **Add New Group** button and then click the **OK** button. The selected objects are assigned to whatever group name is highlighted when the **OK** button is clicked. Any object can be assigned to an unlimited number of groups.

Important note: If you highlight an existing group name, the selected objects ***replace*** *rather than add to* any objects that might have previously been defined for that group.

The Groups area of the Assign Group form lists the names of all the currently defined groups. The Click To area of the form allows you to define new group names, change an existing group name, change the display color for a group and delete an existing group.

To add a new group name, type in the name of the group in the edit box in the Groups area and then click the **Add New Group** button.

Tip:

Assignments made to existing groups replace what is in the group. They do not add to it. If you want to add to an existing group, first select the group, next select the objects you want to assign to the group, and then make the assignment.

To change a group name, highlight the group name in the Groups area. Note that the group name then appears in the edit box at the top of the Groups area. Edit the group name as desired and then click the **Change Group Name** button.

To change the display color associated with a group, highlight the group name in the Groups area and then click the **Change Group Color** button. A color box appears from which you can select any color for the group. Note that the display color associated with a group is used as the background color in the edit box in the Groups area of the Assign Groups form when that group name is highlighted in the form. See the subsection entitled "View by Colors of" in Chapter 6 View Menu for additional information.

To delete a group, highlight the group name in the Groups area. Note that the group name then appears in the edit box at the top of the Groups area. Click the **Delete Group** button to delete the

group. Note that the objects associated with the group are not deleted, the group definition is the only thing that is deleted.

You can click the **Assign menu > Group Names** command or the button and enter the Assign Group form without first making a selection if you wish (regardless of whether the model is locked or unlocked). This is useful if you want to change a group name, change a group color or delete a group. In those cases, enter the Assign Groups form without first making a selection, make the desired name changes, color changes or deletions and then click the **OK** button. Because you entered the form without a selection, the program knows not to make any group assignment to the highlighted group when you click the **OK** button. In this special case where you enter the form without a selection, whatever group name is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the form.

Clear Display of Assigns Command

When you make assignments to objects, those assignments are then displayed on the model. For example, if you assign a frame section property to a line object, the frame section assignments are displayed for all line objects in the model. Sometimes you may not want to display these assignments. You can use the **Assign menu > Clear Display of Assigns** command to remove the displays from the active window.

Note that you can also remove the display of assignments by clicking the **Show Undeformed Shape** button or by clicking the **Display menu > Show Undeformed Shape** command.

Copy Assigns Command

The **Assign menu > Copy Assigns** command works in conjunction with the **Assign menu > Paste Assigns** command. Use the **Assign menu > Copy Assigns** command to indicate which ob-

ject you want to copy from when you use the **Assign menu > Paste Assigns** command.

For example, assume that you want to copy assignments from Line Object 1 to Line Object 2. First select Line Object 1 and click the **Assign menu > Copy Assigns** command. This tells the program that you want to copy from Line Object 1. Then select Line Object 2 and use the **Assign menu > Paste Assigns > Frame/Line** command to complete the copy.

Paste Assigns Command

The **Assign menu > Paste Assigns** command works in conjunction with the **Assign menu > Copy Assigns** command to allow you to copy assignments from one object to another. You can copy from and paste assignments to point objects, line objects and area objects.

For example, assume that you want to copy assignments from Line Object 1 to Line Object 2. First select Line Object 1 and click the **Assign menu > Copy Assigns** command. This tells the program that you want to copy from Line Object 1. Then select Line Object 2 and use the **Assign menu > Paste Assigns > Frame/Line** command.

When you click the **Assign menu > Paste Assigns > Frame/Line** command, a form listing the possible assignments to line objects appears. Check the check boxes for the assignments you want to paste and click the OK button.

If the object you are copying from has a null assignment to it, pasting that assignment to another object will delete the assignment on the other object. For example, assume you paste a point force assignment from Point Object 1, which does not have a point force on it, to Point Object 2, which does have a point force on it. This will delete the point force assignment on Point Object 1. On the other hand, if you instead paste a point force assignment from Point Object 2 to Point Object 1, both points will have point force assignments.

Analyze Menu

The Analyze menu provides basic features for starting and controlling your building analysis. This chapter describes the commands available on the Analyze menu.

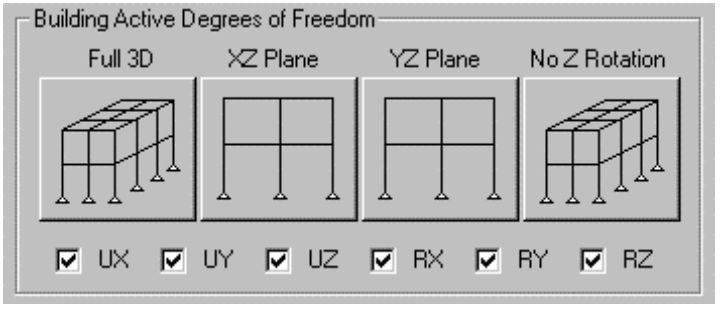
Set Analysis Options Command

Click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options form where you can set various parameters for your analysis. In this form, specify the parameters for building active degrees of freedom, dynamic analysis and P-Delta analysis. Each of these items is described in the subsections of this chapter.

Building Active Degrees of Freedom

The possible degrees of freedom for your building are UX, UY, UZ, RX, RY and RZ. In the Building Active Degrees of Freedom area of the Analysis Options form, specify which of those

degrees of freedom are to be active for your model. A check in the box associated with a degree of freedom means that degree of freedom will be active. You can check or uncheck the degree of freedom boxes as desired.



Four special buttons are provided in this area to allow you to quickly set the degrees of freedom for all of the typical cases that might arise. They are:

- **Full 3D:** This button sets all six degrees of freedom active. The vast majority of your building models should be run using this option.

Tip:

The degree of freedom buttons provide a fast and easy way to set the building active degrees of freedom for your analysis.

- **XZ Plane:** This button sets the UX, UZ and RY degrees of freedom active. It is intended for two-dimensional frames that are modeled in the global XZ plane.

- **YZ Plane:** This button sets the UY, UZ and RX degrees of freedom active. It is intended for two-dimensional frames that are modeled in the global YZ plane.

- **No Z Rotation:** This button sets all degrees of freedom active except for RZ. Often, to satisfy building code requirements, engineers run lateral force analyses of their structure with various positive and negative eccentricities of mass (lateral load) from the calculated center of mass of the building and with all six degrees of freedom active. In addition, analyses are run with the mass (lateral load) located at the calculated center of mass of the building and the Z-rotations locked. The design is then based on the worst case of all the analyses. This No Z

Rotation feature sets the degrees of freedom of your model appropriately to run an analysis with Z-rotations locked.

Set Dynamic Parameters Button

To set the dynamic analysis parameters, click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options form. Check the Dynamic Analysis check box, if it is not already checked, and click the **Set Dynamic Parameters** button. This opens the Dynamic Analysis Parameters form. The following bullet items describe the various areas in this form:

- **Number of Modes:** Specifies the number of Eigen or Ritz modes that you want the program to capture.

Tip:

*If you are running response spectrum or time history analysis, we **strongly recommend** that you use ritz-vectors. It is **especially important** that you use ritz-vectors when performing nonlinear time history analysis.*

- **Type of Analysis:** Choose either eigenvector or ritz-vector analysis in this area. If you are running response spectrum or time history analysis, we ***strongly recommend*** that you use ritz-vectors. It is ***especially important*** that you use ritz-vectors when performing nonlinear time history analysis.

- **EigenValue Parameters:** This area of the form is only active if you select "Eigenvector" in the Type of Analysis area. The following parameters are specified in this area:

  - ✓ **Frequency Shift (Center):** This is the center of the cyclic frequency range, f_0.

  - ✓ **Cutoff Frequency (Radius):** This is the radius of the cyclic frequency range, also known as the cutoff frequency, f_{max}.

  - ✓ **Relative Tolerance:** This is the relative convergence tolerance, ε.

  - ✓ **Include Residual-Mass Modes:** If you check this box, the program computes residual-mass (missing-mass modes). Those modes are used to approximate high-frequency behavior when the mass participation ratio for a given direction of acceleration load is less than 100%.

When this option is chosen, the number of eigenvector modes recovered is 3 less than the number specified, and up to 3 non-zero residual-mass modes are reported. Thus, when you check the Include Residual-Mass Modes check box, at least four modes need to be requested because the last 3 modes are automatically reserved for the residual-mass modes.

The default values for the Eigenvalue Parameters will be sufficient for most analyses.

- **Starting Ritz Vectors:** This area of the form is only active if you select "Ritz Vectors" in the Type of Analysis area. In this area, specify the starting ritz vectors.

  The possible ritz load vectors are the acceleration loads in the global X, Y and Z directions and all of your defined static load cases. You can use any of those loads as starting ritz-vectors.

  A load is used as a starting ritz-vector if it is in the Ritz Load Vectors list box in the Starting Ritz Vectors area of the Dynamic Analysis Parameters form. If the load is in the List of Loads list box, it is not used as a starting ritz-vector. You can use the **Add** and **Remove** buttons to shift loads into and out of the Ritz Load Vectors list box, respectively.

  The Include Nonlinear Link Vectors check box is visible if you have assigned link properties in your model. The check box is active if link properties have been assigned and the Ritz Vector analysis type has been specified.

  If you check the Include Nonlinear Link Vectors box, the program automatically provides a starting load vector for each nonlinear degree of freedom in each link element. When you use this option, be sure to specify a sufficient number of modes to allow the program to capture the modes associated with those special starting vectors. The program does not add additional modes to the number you requested when you check the Include Nonlinear Link Vectors check box.

Set P-Delta Parameters Button

To set the P-Delta analysis parameters, click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options form. Then check the Include P-Delta check box, if it is not already checked and click the **Set P-Delta Parameters** button. This opens the P-Delta Parameters form. The following bullet items describe the various areas in this form:

- **Method:** Initial P-Delta analysis in the program considers the P-Delta effect of a single loaded state upon the structure. There are two ways to specify this load:

  ✓ **Non-iterative Based on Mass:** The load is computed automatically from the mass at each level as a story-by-story load upon the structure. This approach is approximate, but does not require an iterative solution.

  This method essentially treats the building as a simplified stick model to consider the P-Delta effect. It is much faster than the iterative method. It does not capture local buckling as well as the iterative method. This method works best if you have a single rigid diaphragm at each floor level, although it also works for other cases.

  This method allows you to consider P-Delta in cases where you have not specified gravity loads in your model. If you have specified gravity loads in your model, in general, we recommend that you use the Iterative Based on Load Combination option.

  ✓ **Iterative Based on Load Combination:** The load is computed from a specified combination of static load cases. This is called the P-Delta load combination. For example, the load may be the sum of a dead load case plus a fraction of a live load case. This approach requires an iterative solution to determine the P-Delta effect upon the structure.

  This method considers the P-Delta effect on an element-by-element basis. It captures local buckling effects better

Tip:

We recommend that you use the Iterative Based on Load Cases method for P-Delta analysis unless there are no gravity loads specified in your model.

than the non-iterative method. We recommend that you use this iterative method in all cases, except those where no gravity load is specified in your model.

- **Iteration Controls:** This area is active if you select the Iterative Based on Load Cases option in the Method area of the form. When you specify a P-Delta load combination, the following parameters may also be specified to control the iterative solution:

  - ✓ **Maximum Number of Iterations:** Specifies the maximum number of additional analyses after the first analysis has been run. This is used to prevent excessive computational time, given that each iteration requires about as much computational effort as a linear static analysis. The default is one.

  - ✓ **Relative Displacement Convergence Tolerance:** This item measures convergence. The default value is 0.001. If the relative change in displacement from one iteration to the next is less than the tolerance, no further iterations are performed. The relative change in displacement is defined as the ratio of the maximum change in displacement to the largest displacement in either iteration. Note that rotations and translations are treated equally.

    If convergence has not been obtained after the maximum number of iterations has been performed, the results of the analysis may be meaningless, and they should be viewed with great skepticism. Failure to converge may be result from several causes:

    - ❑ Too few iterations were permitted. A reasonable number is usually 2 to 5, although more may be required, depending on the particular problem at hand.

    - ❑ A convergence tolerance that is too small is used. A reasonable value depends on the particular problem. Beware, however, that using a value that is too large may result in convergence to meaningless results.

Note:

P-Delta analysis based on specified load cases is an iterative analysis. It may take several iterations to achieve convergence.

- ❑ The structure is near buckling. The structure should be stiffened against buckling, or the magnitude of the P-Delta load combination should be reduced.

- **P-Delta Load Combination:** This area is active if you select the Iterative Based on Load Cases option in the Method area of the form. In this area, specify the single load combination to be used for the initial P-Delta analysis of the structure.

  As an example, assume that the building code requires the following load combinations to be considered for design:

  (1) 1.4 dead load

  (2) 1.2 dead load + 1.6 live load

  (3) 1.2 dead load + 0.5 live load + 1.3 wind load

  (4) 1.2 dead load + 0.5 live load - 1.3 wind load

  (5) 0.9 dead load + 1.3 wind load

  (6) 0.9 dead load - 1.3 wind load

  For this case, the P-Delta effect from overall sway of the structure can usually be accounted for, conservatively, by specifying the P-Delta load combination to be 1.2 times dead load plus 0.5 times live load. This will accurately account for this effect in load combinations 3 and 4 above, and will conservatively account for this effect in load combinations 5 and 6. This P-Delta effect is not generally important in load combinations 1 and 2 because there is no lateral load.

  It is also possible to accurately account for the P-Delta effect from the deformation of the members between their ends in the program analysis, ***but we do not recommend that you do this***. Instead, we recommend that you account for this effect using factors in your design. The program design postprocessors assume this is what you have done and includes those factors, where appropriate, in the design.

If you did want to account for the P-Delta effect from deformation of the members between their ends in the program analysis, first break up all of your columns into at least two objects between story levels. Then, run each of the six load cases described earlier separately with a different P-Delta load combination for each. Again, it is recommended that this effect be accounted for by using factors in your design, the same way it is performed in the program design postprocessors.

Save Access DB File Button

This program has the ability to save all input and output data to an Access Database file. Checking the Save Access DB File check box brings up the Save Access Database File As form. The program provides a filename and path, which you can accept or change. Click the Save button on the Save Access Database File As form to complete the save. Note that the name of the file, complete with path, appears in the edit box on the Analysis Options form. Also note that the File Name button is enabled. Click the File Name button to recall the Save Access Database File As form and store the file under a different name or path, if desired.

Run Analysis Command

Tip:

*The Run Minimized option has the advantage of providing a **Cancel** button while the analysis is running, which would allow you to easily abort the analysis at any time.*

Run an analysis of your building by clicking the **Analyze menu > Run Analysis** command or the **Run Analysis** button , or by pressing the F5 function key on you keyboard. When you execute this command, the Run Options form appears with the following three choices:

- **Run:** This option opens the Analysis Window on top of the main program window and runs the analysis. Information concerning the analysis scrolls by in the Analysis Window as the run progresses. See the subsequent subsection entitled "Analysis Window" for more information.

  This option runs your analysis in such a way that if you switch away from this program to another program while the analysis

is running, you may not be able to switch back until the analysis run is complete. Also, there is no way to cancel an analysis run after you have started it using this option.

In general you should only use this option for smaller models that run quickly.

- **Run Minimized:** This option prompts you for a file name to save your analysis results and closes the main program window, leaving only the Analysis Window open. Information concerning the analysis run scrolls by in the Analysis Window as the run progresses. See the subsequent subsection below entitled "Analysis Window" for more information.

  This option runs your analysis in such a way that if you switch away from this program to another program while the analysis is running, you *are* able to switch back to this program and observe the information in the Analysis Window. Also, this option provides you with a **Cancel** button that allows you to cancel (stop) the analysis at any time.

- **Cancel:** This cancels the Run Analysis command. The analysis is not run if you click this button.

Note:

Use the scroll bar in the Analysis Window to scroll through all of the information and check for any warnings or errors that might invalidate your analysis.

Analysis Window

As your program analysis runs, information about the analysis scrolls by in the Analysis Window. When the analysis is complete, and before you have clicked the **OK** button in the Analysis Window, use the scroll bar in the Analysis Window to scroll through all of the information and check for any warnings or errors that might invalidate your analysis. After you click the **OK** button in the Analysis Window, the warnings or error messages disappear. However, they are saved, in text form, in the .log file.

Calculate Diaphragm Centers of Rigidity

Clicking the **Analyze menu > Calculate Diaphragm Centers of Rigidity** command computes the location of diaphragm centers of rigidity automatically. This option is available only after an analysis has been run.

To view the results of the analysis, click the **Display menu > Set Output Table Mode command**, which brings up the Display Output Tables form. On this form, click the Building Output check box and the **OK** button. The Centroids of Cummulative Mass and Centers of Rigidity form will display the analysis results.

Display Menu

General

The Display menu provides options for displaying input and output information on the model and onscreen in a tabular form. This chapter describes those options.

Show Undeformed Shape Command

Clicking the **Display menu > Show Undeformed Shape** command or the **Show Undeformed Shape** button does the following:

- If you are creating your model, it clears the display of any assignments that are still showing on the model. It essentially functions the same as the **Assign menu > Clear Display of Assigns** command in this case.

- If you are currently looking at onscreen output results of any type that are plotted on the model, using this command clears the display of the results and returns the view to an undeformed shape view.

Note that this command only affects the active window.

Show Loads Command

Click the **Display menu > Show Loads** command to display loads that you have input for the model using the **Assign menu > Joint/Point Loads**, **Assign menu > Frame/Line Loads** and **Assign menu > Shell/Area Loads** commands. Clicking the **Display menu > Show Loads** command brings up a submenu where you can choose to display joint/point loads, frame/line loads or shell/area loads. Each of these options is described in a subsequent subsection. The loads that you specify are only displayed in the currently active window.

Note that alternatively you can right click on any object and then select the Loads tab to see what loads are assigned to that object.

Show Loads > Joint/Point Command

Click the **Display menu > Show Loads > Joint/Point** command of the button to bring up the Show Joint/Point Loads form. The following bullet items describe the various areas in this form.

Tip:

You can right click on any object and select the Loads tab as an alternative for viewing the loads on an object.

- **Load Case:** From the drop-down box choose the static load case for which you want to display the joint/point loads. Note that static load cases are defined using the **Define menu > Static Load Cases** command (see Chapter 7 Define Menu).
- **Load Type:** Choose the type of load that you want to display. The choices are forces, displacements or temperature values. You can display one of these types of loads at a time. Note that if the Load Type has not been assigned to the specified load

case (see Chapter 10 Assign Menu), the Load Type option is grayed out and inactive.

- **Show Loading Values:** When the Show Loading Values check box is *unchecked,* forces and displacements are indicated by arrows in the appropriate direction only. When the Show Loading Values check box is *checked,* forces and displacements are indicated by arrows in the appropriate direction together with loading values (text).

  Loading values are always shown when you choose the Temperature Values option in the Load Type area regardless of whether the Show Loading Values check box is checked or unchecked.

Show Loads > Frame/Line Command

Click the **Display menu > Show Loads > Frame/Line** command of the button to bring up the Show Frame/Line Loads form. The following bullet items describe the various areas in this form.

- **Load Case:** From the drop-down box choose the static load case for which you want to display frame/line loads. Note that static load cases are defined using the **Define menu > Static Load Cases** command (see Chapter 7 Define Menu).

Note:

The values displayed for line object temperature loads include the effect of point temperatures at the end points of the line object if you specified this when you assigned the temperature load.

- **Load type:** Choose the type of load that you want to display from this area. Note that you can only display one of these types of loads at a time. Also note that if the Load Type has not been assigned to the specified load case (see Chapter 10 Assign Menu), the Load Type option is grayed out and inactive. The choices are:

  ✓ **Span Loading (Forces):** This includes all of the point, uniform and trapezoidal force loads (not moment loads) applied to the line object.

- ✓ **Span Loading (Moments):** This includes all of the point, uniform and trapezoidal moment loads applied to the line object.

- ✓ **Temperature Values:** This includes all of the temperature loads applied to the line object.

Important Note: When temperature loads are displayed, two numbers are shown for each line object. Those two numbers correspond to the temperatures at the ends of the object. If upon assigning the temperature load you indicated that the effects of point temperature *were not* to be included, the two displayed temperatures for the line object are the same and are equal to the temperature you assigned to the object. This is true regardless of any point object temperatures that may be assigned to the point objects at the end of the line object.

If upon assigning the temperature load you indicated that the effects of point temperature *were* to be included, the two displayed temperatures for the line object are equal to the temperature you assigned to the object plus the temperature assigned to the point object at the considered end of the line object. In that case, the two displayed temperatures for a line object may be different.

- **Include Point Object Loads:** This check box toggles on and off whether point object loads are shown together with the line object loads. When this box is checked, both force and moment point object loads are displayed along with the selected type of line object loads.

- **Show Loading Values:** When the Show Loading Values check box is *unchecked*, forces and moments are indicated by arrows in the appropriate direction only. When the Show Loading Values check box is *checked,* forces and moments are indicated by arrows in the appropriate direction together with loading values (text).

  Loading values are always shown when you choose the Temperature Values option in the Load Type area, regardless of

whether the Show Loading Values check box is checked or unchecked.

Show Loads > Shell/Area Command

Click the **Display menu > Show Loads > Shell/Area** command of the button to bring up the Show Shell/Area Loads form. The following bullet items describe the various areas in this form.

- **Load Case:** From the drop-down box choose the static load case for which you want to display shell/area loads. Note that static load cases are defined using the **Define menu > Static Load Cases** command (see Chapter 7 Define Menu).

- **Load type:** Choose the type of load that you want to display from this area. Note that you can only display one of these types of loads at a time. Also note that if the Load Type has not been assigned to the specified load case (see Chapter 10 Assign Menu), the Load Type option is grayed out and inactive. The choices are:

  - ✓ **Uniform Load Values:** This is a uniform surface load on an area object. If you choose this option, also specify the loading direction for which you want the loads displayed by choosing a direction from the associated Direction drop-down box.

  - ✓ **Temperature values:** This includes all of the temperature loads applied to the area object.

    Important Note: When temperature loads are displayed, numbers are shown at each corner of the area object. Those numbers correspond to the temperatures at the corners of the object. If upon assigning the temperature load you indicated that the effects of point temperature *were not* to be included (see "Shell/Area Loads > Temperatures Command" in Chapter 10 Assign Menu), the displayed temperatures for the area object are the same and are equal

Note:

The values displayed for area object temperature loads include the effect of point temperatures at the corner points of the area object if you specified this when you assigned the temperature load.

to the temperature you assigned to the object. This is true regardless of any point object temperatures that may be assigned to the point objects at the corners of the area object.

If upon assigning the temperature load you indicated that the effects of point temperature *were* to be included, the displayed temperatures for the area object are equal to the temperature you assigned to the object plus the temperature assigned to the point object at the considered corner of the area object. In that case, the displayed temperatures for an area object may be different at each corner.

12

Set Input Table Mode Command

Use the **Display menu > Set Input Table Mode** command or button to display input data on screen. Use of this command provides you with the opportunity to complete an onscreen review of the input parameters you used in building your model.

Clicking the **Display menu > Set Input Table Mode** command or its associated button brings up the Data Tables Input form. This form has two tabs:

- **Definitions Tab** - The Definitions tab has three areas: Building Data, Properties, and Load Definitions. Each of those areas lists a series of data types that can be used in building a model using in this program. Include a data type in the onscreen display by checking its associated check box and clicking the **Ok** button. Note that if an option is grayed out or inactive, the data type is not used in the current model. The **Check/Uncheck All button** in this area simultaneously toggles all of the check boxes to be check or unchecked.

- **Assignments Tab:** The Assignments tab has three areas: Point Assignments, Line Assignments and Area Assignments. Each of those areas lists a series of data types that can be used in building a model using in this program. Include a data type in the onscreen display by checking its associated check box

and clicking the **Ok** button. Note that if an option is grayed out or inactive, the data type is not used in the current model. The **Check/Uncheck All button** in this area simultaneously toggles all of the check boxes to be check or unchecked.

Selection Only Check Box

When the Selection Only check box near the bottom of the Database Input Tables form is *checked*, the program will display only those data that have been selected by checking the check box in the six areas of the Definitions and Assignments tabs. This is the program default.

When the Selection Only check box is *unchecked*, the program will display all data types used in the model even though no check boxes have been checked on the Definitions and Assignments tabs.

Select Loads Button

Clicking the **Select Loads** button near the bottom of the Database Input Tables form brings up the Select Load Conditions form. Select the load condition(s) you want to display by highlighting it in the Select area of this form and clicking the **Ok** button. You can highlight one or more load cases in the list at the same time.

Note that the form also displays the number of loads selected relative to the number of loads available (e.g., 1 of 3 Loads Selected).

Check/Uncheck All Button

The **Check/Uncheck All** button near the bottom of the Database Input Tables form is similar to the **Check/Uncheck All** buttons found in each area of the Definitions and Assignments tabs. However, clicking this button selects/deselects *all data types* in *all areas* of *both the Definitions and the Assignments tabs*.

Show Deformed Shape Command

After an analysis has been run, clicking the **Display menu > Show Deformed Shape** command or the **Display Static Deformed Shape** button brings up the Deformed Shape form. Use this form to specify the load case for which you want to plot the deformed shape. The following bullet items describe the various areas in the Deformed Shape form.

- **Load:** From the drop-down box choose the load case for which you want to plot the deformed shape. Note that you can plot a deformed shape for any static load case, response spectrum case, time history case, static nonlinear case, or load combination. Following is a description of what is plotted for each of these items.

  - ✓ **Static load case (linear):** The deformed shape multiplied by a scale factor is plotted. When a deformed shape is displayed for a static load case, use the left and right arrow keys on the status bar, , to quickly display deformed shapes for other static load cases.

  - ✓ **Response spectrum case:** A deformed shape multiplied by a scale factor is plotted. In that case, the deformed shape has little meaning because the response spectrum analysis causes all results (deformations) to be positive, and the displacements at each point are the maximum displacement at that point that may not occur at the same time in an earthquake as the maximum displacement at another point.

    It can be useful to compare the deformed shape plot for a response spectrum with the undeformed shape to see which parts of the structure are experiencing the most displacement.

    When a deformed shape is displayed for a response spectrum case, use the left and right arrow keys on the status

bar, , to quickly display deformed shapes for other response spectrum cases.

- ✓ **Time history case:** When you choose a time history case, a box appears where you specify the time step in the time history analysis for which you want to display the deformed shape. Choose a time before clicking the **OK** button to plot the deformed shape.

  If you specify a time that is before the time history starts, the first step of the time history is displayed. If you specify a time that is after the time history finishes, the last step of the time history is displayed. If you specify a time during the time history that is not exactly the same as one of the output time step times, the nearest time step is displayed.

  A deformed shape multiplied by a scale factor is plotted for the chosen time of the time analysis. When a deformed shape is displayed, use the left arrow key on the status bar, , to display the previous time step in the analysis and the right arrow key on the status bar, , to display the next time step in the analysis.

- ✓ **Static nonlinear case:** When you choose a static nonlinear case, a box appears where you specify the step in the static nonlinear analysis for which you want to display the deformed shape. Choose a step number before clicking the **OK** button to plot the deformed shape. To get an idea of the force and deformation associated with any step in the pushover, click the **Display menu > Show Static Pushover Curve** command, click the File menu at the top of the resulting form and click the **Display Tables** command. This displays a table that, among other things, includes the force and deformation for each step of the nonlinear static analysis.

  A deformed shape multiplied by a scale factor is plotted for the chosen step of the static nonlinear analysis. When a deformed shape is displayed you can use the left arrow key

on the status bar, <<, to display the previous step in the analysis and the right arrow key on the status bar, >>, to display the next step in the analysis. In this manner you can easily step through deformed shape views of the entire pushover analysis if desired.

See the discussion of contours in the subsection entitled "Colors > Output Command" in Chapter 14 for additional information on deformed shapes for static nonlinear analyses.

✓ **Load combination:** A deformed shape multiplied by a scale factor is plotted. In this case the deformed shape may or may not have much meaning depending on what is in the load combination.

If the load combination is a single-valued load combination, the displayed results are meaningful. If the load combination is multi-valued, the displayed results have little meaning. In that case the value with the largest absolute value is displayed. For example, if the minimum value at point 1 is -3 and the maximum value is +2, the -3 value is displayed. If an adjacent point has a minimum value of -1 and a maximum value of +2, then +2 is displayed at that point. This process continues on for every point. It can lead to some unusual looking deformed shape plots.

When a deformed shape is displayed for a load combination, use the left and right arrow keys on the status bar, << >>, to quickly display deformed shapes for other load combinations.

▪ **Scaling:** Specifies the scaling that is used to scale the plotted deformations. If you specify a scaling factor of 100, all deformations are plotted to scale at 100 times their actual value. For example, a deformation of 1 inch is plotted to scale as if it is a 100-inch deformation.

By default the program automatically determines a scaling factor for the deformed shape plot. If you want the program to

automatically determine a scaling factor, leave the Auto option selected. Otherwise, select the Scale Factor option and specify your own scale factor.

The program calculates the scale factor as a multiple of the default font size, which itself is determined as a multiple of the average story height. The advantage to determining the scale factor as a multiple of the default font size is that the default font size is limited by a specified minimum and maximum size that is specified in the preferences. This helps keep the automatically determined scale factor within a reasonable range most, but not all, of the time. If the automatic scale factor seems to cause a display problem, specify your own factor.

- **Cubic Curve:** Specify if the cubic curve function should be used when plotting the deformed shape. Cubic curves affect how the line objects with frame section properties appear in the deformed shape plot.

Note:

The cubic curve feature only affects line objects with frame section properties.

The program only saves point/joint displacements from an analysis. Thus, when it prepares to plot a deformed shape, the only deformations it has available are the point/joint deformations. No deformations internal to the line objects are available.

When the deformed shape is plotted, the point/joints are put in their correct locations. If you specify that the cubic curves are not to be considered, the frame elements are simply drawn as straight lines connecting the appropriate points/joints.

If you specify that cubic curves are to be considered, the program does the following:

- ✓ Calculate an approximate deflection (translation and rotation) at the center of the beam.
- ✓ Draw a cubic curve from the left end of the beam to the center of the beam based on the actual translation and rotation at the left end and the approximate translation and rotation at the center.

- ✓ Draw a cubic curve from the center of the beam to the right end of the beam based on the approximate translation and rotation at the center and the actual translation and rotation at the left end.

  Note that drawing a single cubic curve from the left end to the right end of the beam does not give a very good representation of a loaded beam, but using two curves, as is performed in this program, gives a pretty good plot. Nevertheless, when plotting a deformed shape, keep in mind that the displacements at the joints are exact, whereas deformation shown for the frame members is approximate, even when the cubic curve option is activated.

When you are viewing a deformed shape, right click on any point object to bring up the Joint Displacements form that displays the displacements (translation and rotations) for that point object in the global coordinate system and in the current units. In this form, click on the **Lateral Drifts** button to display displacements at all story levels where a point object exists in the same plan location as the selected point object. Also, drifts are displayed for all story levels where a point object exists at the top and bottom of the story level in the same plan location as the selected point object. The drifts are calculated as the displacement at the top of the story level minus the displacement at the bottom of the story level divided by the story level height.

Note:

You can animate deformed shapes in this program.

Important Note: When an analysis is run the program automatically creates a point object at the center of mass of all rigid diaphragms. This point object is restrained against translation in the Z-direction and against rotation about the global X- and Y-axes to be compatible with the rigid diaphragm. You can right click on this point to see displacements at the center of mass of the diaphragm.

Note that the Z translation and X and Y rotations for those center of mass points are zero because the points are restrained. Also note that you can use the Point Objects item in the Object Visibility area of the Set Building View Options form to toggle those

center of mass joints on and off. Use the **Set Building View Options** button to access the form.

Finally, when a deformed shape is displayed in the active window, click on the **Start Animation** button on the status bar to animate the deformed shape. Sometimes this makes it easier to recognize the deformed shape. Click the **Stop Animation** button on the status bar to stop the animation. Note that the program includes sound with the animation. You can use the **Options menu > Sound** command to toggle this sound on and off.

Show Mode Shape Command

Click the **Display menu > Show Mode Shape** command or the **Show Mode Shape** button to bring up the Mode Shape form. In this form, specify the mode for which you want to plot the deformed shape. The following bullet items describe the various areas in the Mode Shape form.

- **Mode number:** Specifies the mode number for which you want to plot the deformed shape. Use the scroll buttons to scroll to the desired mode shape or simply type in the mode number you would like to display. If you type in a mode number larger than the number of modes used in the analysis, the program defaults to the highest mode number in the analysis.

- **Scaling:** Specify the scaling that is used to scale the plotted mode shapes. By default the program automatically determines the scaling factor for the mode shape plot. If you want the program to automatically determine a scaling factor, leave the Auto option selected. Otherwise, select the Scale Factor option and specify your own scale factor. When you select your own scaling factor, a factor of 1 gives the same plot as the automatic scaling. A factor of 2 gives twice the apparent deformation and so on.

  The program calculates the default deformation (scale factor of 1 deformation) as a multiple of the default font size, which it-

self is determined as a multiple of the average story height. The advantage to determining the scale factor as a multiple of the default font size is that the default font size is limited by a specified minimum and maximum size that is specified in the preferences. This helps keep the automatically determined deformations for the mode shapes within a reasonable range most, but not necessarily all, of the time. If the automatic scale factor seems to cause a display problem, specify your own factor.

Note:

You can animate mode shapes in this program.

- **Cubic Curve:** Refer to the bullet item entitled "Cubic Curve" in the previous "Deformed Shape" subsection for a description of the cubic curve item.

When a mode shape is displayed, click on the **Start Animation** button on the status bar to animate the mode shape. Sometimes this makes it easier to recognize the deformed shape. Click the **Stop Animation** button on the status bar to stop the animation. Note that the program includes sound with the animation. You can use the **Options menu > Sound** command to toggle this sound on and off.

Also, when a mode shape is displayed, use the left arrow key on the status bar, , to display the previous mode shape and the right arrow key on the status bar, , to display the next mode shape.

Show Member Forces/Stress Diagram Command

Click the **Display menu > Show Member Forces/Stress Diagram** command or click the **Display Member Force Diagram** button to display Support/Spring Reactions; Frame/Pier/Spandrel Forces; Shell Stresses/Forces; and Link Forces. These items are described in detail in subsequent subsections.

Show Member Forces/Stress Diagram > Support/Spring Reactions Command

Display support and spring reactions directly on the program model by clicking the **Display menu > Show Member Forces/Stress Diagram > Support/Spring Reactions** command. Using this command brings up the Point Reaction Forces form. The following bullet items describe the areas in this form:

- **Load:** From the drop-down box choose the load case for which you want to display support or spring reactions. Note that you can plot support or spring reactions for any static load case, response spectrum case, time history case, static nonlinear case, or load combination. For time history cases, also specify a time for which you want to see the reactions. For static nonlinear cases, also specify a step at which you want to see the reactions.

  Following is a description of what is plotted for each of those cases.

  - ✓ **Static load case (linear):** The force values are displayed along with arrows showing the direction of the force. The arrows indicate the direction of the force acting from the support (spring) onto the elements connected to the support (spring). When reactions are displayed for a static load case, use the left and right arrow keys on the status bar, , to quickly display reactions for other static load cases.

  - ✓ **Response spectrum case:** The reaction values are displayed along with arrows showing the direction of the force. The arrows indicate the direction of the force acting from the support (spring) onto the elements connected to the support (spring). Note that the response spectrum reactions show the maximum value obtained for each component of each reaction and also note that the values may not occur at the same time in an earthquake.

When reactions are displayed for a response spectrum case, use the left and right arrow keys on the status bar, , to quickly display reactions for other response spectrum cases.

- ✓ **Time history case:** When you choose a time history case, a box appears where you specify the time step in the time history analysis for which you want to display the reactions. Choose a time before clicking the **OK** button to plot the reactions.

  If you specify a time that is before the time history starts, the first step of the time history is displayed. If you specify a time that is after the time history finishes, the last step of the time history is displayed. If you specify a time during the time history that is not exactly the same as one of the output time step times, the nearest time step is displayed.

  The reaction values are displayed along with arrows showing the direction of the force. The arrows indicate the direction of the force acting from the support (spring) onto the elements connected to the support (spring).

  When reactions are displayed, use the left arrow key on the status bar, , to display the reactions for the previous time step in the analysis and the right arrow key on the status bar, , to display the reactions for the next time step in the analysis.

- ✓ **Static nonlinear case:** When you choose a static nonlinear case, a box appears where you specify the step in the static nonlinear analysis for which you want to display the reactions. Choose a step number before clicking the **OK** button to plot the deformed shape. To get an idea of the force and deformation associated with any step in the pushover, click the **Display menu > Show Static Pushover Curve** command, click the File menu at the top of the resulting form and click the **Display Tables** command. This displays a

table that among other things includes the force and deformation for each step of the nonlinear static analysis.

The reaction values are displayed along with arrows showing the direction of the force. The arrows indicate the direction of the force acting from the support (spring) onto the elements connected to the support (spring).

When reactions are displayed, use the left arrow key on the status bar, <<, to display the previous step in the analysis and the right arrow key on the status bar, >>, to display the next step in the analysis. In this manner, you can easily step through the support or spring reactions for the entire pushover analysis if desired.

- ✓ **Load combination:** The reaction values are displayed along with arrows showing the direction of the force. The arrows indicate the direction of the force acting from the support (spring) on to the elements connected to the support (spring).

  If a load combination is a single-valued load combination, its reaction values are displayed. If a load combination is multi-valued, the displayed results are those with the largest absolute value. For example, if the minimum value at point 1 is -3 and the maximum value is +2, the -3 value is displayed. If an adjacent point has a minimum value of -1 and a maximum value of +2, then +2 is displayed at that point. This process continues for every component at every reaction point.

  When reactions are displayed for a load combination, use the left and right arrow keys on the status bar, << >>, to quickly display reactions for other load combinations.

- **Type:** Indicate whether to display support reactions or spring forces.

When support or spring reactions are displayed, right click on any point object to see its support or spring forces in a tabular form. Sometimes it is easier to read the values using this method.

Show Member Forces/Stress Diagram > Frame/Pier/ Spandrel Forces Command

Display column, beam, brace, pier and spandrel forces directly on your program model by clicking the **Display menu > Show Member Forces/Stress Diagram > Frame/Pier/Spandrel Forces** command. Using this command brings up the Member Force Diagram for Frames form. The following bullet items describe the areas in that form:

- **Load:** From the drop-down box, choose the load case for which you want to display the member forces. Note that you can plot member forces for any static load case, response spectrum case, time history case, static nonlinear case, or load combination. For time history cases, also specify a time for which you want to see the forces. For static nonlinear cases, also specify a step at which you want to see the forces.

- **Component Type:** Specify which component of force you want to see. You can choose any one (at a time) of the following:

  - ✓ Axial Force
  - ✓ Shear 2-2
  - ✓ Shear 3-3
  - ✓ In-Plane Shear
  - ✓ Torsion
  - ✓ Moment 2-2
  - ✓ Moment 3-3
  - ✓ In-Plane Moment

Note:

The column, beam, brace, pier and spandrel forces are all displayed at the same time. If you want to see forces for just one of those types of objects, make the other objects invisible using the ***View menu > Set Building View Options*** *command.*

The in-plane moment and shear items are only available if the currently active window (the one where the forces are going to be displayed) is a 2-D view. This view is useful for looking at two-dimensional frames when the local 2-axis of the columns is in the plane of the frame.

If the local axes of a member are rotated such that neither the local 2- nor 3-axis is in the 2-D plane, the force displayed when the in-plane option is chosen is made up of appropriate components from the local 2 and 3 axes.

- **Scaling:** By default the program automatically determines a scaling factor for the plotted forces. If you want the program to automatically determine a scaling factor, leave the Auto option selected. Otherwise, select the Scale Factor option and specify your own scale factor.

  The program calculates the scale factor as a multiple of the default font size, which itself is determined as a multiple of the average story height. The advantage to determining the scale factor as a multiple of the default font size is that the default font size is limited by a specified minimum and maximum size that is specified in the preferences. This helps keep the automatically determined scale factor within a reasonable range most, but not all, of the time. If the automatic scale factor seems to cause a display problem, specify your own factor.

- **Fill diagram and Show Values on Diagram Check Boxes:** You can display the force diagrams filled with no text values, unfilled with no text values or unfilled with text values.

  - ✓ **To display force diagrams filled with no text values:** Check the Fill Diagram check box. Note that if the Show Values on Diagram check box is currently checked, you must uncheck it before you can check the Fill Diagram check box.

  - ✓ **To display force diagrams unfilled with no text values:** Leave both diagrams' check boxes unchecked.

Tip:

When forces are displayed on the model, right click on any column, beam, brace, pier or spandrel to pop up a window where you can run your mouse pointer over the element and see the force value at any location.

- ✓ **To display force diagrams unfilled with text values:** Check the Show Values on Diagram check box. Note that if the Fill Diagram check box is currently checked, you must uncheck it before you can check the Show Values on Diagram check box.

When forces are displayed on the model note the following:

- You can right click on a frame element, wall pier or wall spandrel to pop up a window where you can run your mouse pointer over the element and see the force value at any location.
- When forces are displayed, use the left and right arrow keys on the status bar, , to quickly display forces for other load cases.
  - ✓ If you are currently viewing a static load case (linear), the arrow keys step you through all of the other static load cases (linear).
  - ✓ If you are currently viewing a response spectrum case, the arrow keys step you through all of the other response spectrum cases.
  - ✓ If you are currently viewing a time step in a time history case, the arrow keys step you through all of the other time steps in the time history case.
  - ✓ If you are currently viewing a step in a static nonlinear case, the arrow keys take you through all of the other steps in the static nonlinear case.
  - ✓ If you are currently viewing a load combination, the arrow keys step you through all of the other load combinations.
- Multi-valued load combinations plot a range of values. See the subsection titled "Color > Output Command" in Chapter 14 for more information.

- For frame elements, exact force values are plotted at all output station locations. Those exact force values are then connected by straight-line segments to complete the force diagram. See the section entitled "Frame/Line > Frame Output Stations Command" in Chapter 10 Assign Menu for more information.

- For wall pier and spandrel elements, exact force values are plotted at the ends of the element. Those exact force values are then connected by a straight-line segment to complete the force diagram.

  Note that because of the above described method of reporting pier and spandrel forces, the forces reported for spandrel elements may not be refined enough if your design is governed by gravity load. In cases where your design is governed by gravity load, we recommend that you model the spandrels with frame elements.

Show Member Forces/Stress Diagram > Shell Stresses/ Forces Command

Note:

Shell element internal forces are reported at the element midsurface in force per unit length. Shell element internal stresses are reported at both the top and bottom of the element in force per unit area.

Display internal shell element forces and stresses directly on your the program model by clicking the **Display menu > Show Member Forces/Stress Diagram > Shell Stresses/ Forces** command. Using this command brings up the Element Force/Stress Contours for Shells form.

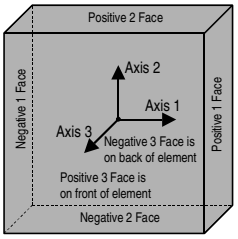
Important note: The internal shell element forces are forces per unit length acting along the midsurface of the shell element (area object). The internal shell element stresses are stresses acting on the edges (not positive 3-axis face and negative 3-axis face) of the shell element (area object).

The following bullet items describe the areas in the Element Force/Stress Contours for Shells form:

- **Load** - From the drop-down box, choose the load case for which you want to display shell element forces or stresses. Note that you can plot shell element forces or stresses for any

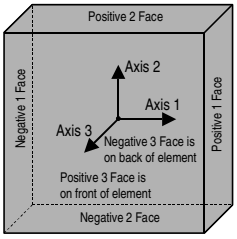
static load case, response spectrum case, time history case, static nonlinear case, or load combination. For time history cases, also specify a time for which you want to see the forces or stresses. For static nonlinear cases, also specify a step at which you want to see the forces or stresses.

- **Component Type** - Specify whether you want to see the shell element internal Forces or the internal Stresses.



  - ✓ **Forces** - For shell element internal *forces,* the possible components are as follows:
    - ❑ **F11:** Direct force per unit length acting at the mid-surface of the element on the positive and negative 1 faces in the 1-axis direction.
    - ❑ **F22:** Direct force per unit length acting at the mid-surface of the element on the positive and negative 2 faces in the 2-axis direction.
    - ❑ **F12:** Shearing force per unit length acting at the mid-surface of the element on the positive and negative 1 faces in the 2-axis direction, and acting on the positive and negative 2 faces in the 1-axis direction.
    - ❑ **FMAX:** Maximum principal force per unit length acting at the mid-surface of the element. Note that by definition, principal forces are oriented such that the associated shearing force per unit length is zero.
    - ❑ **FMIN:** Minimum principal force per unit length acting at the mid-surface of the element. Note that by definition, principal forces are oriented such that the associated shearing force per unit length is zero.
    - ❑ **M11:** Direct moment per unit length acting at the mi-surface of the element on the positive and negative 1 faces about the 2-axis.

- ❑ **M22:** Direct moment per unit length acting at the mid-surface of the element on the positive and negative 2 faces about the 1-axis.
- ❑ **M12:** Twisting moment per unit length acting at the mid-surface of the element on the positive and negative 1 faces about the 1-axis, and acting on the positive and negative 2 faces about the 2-axis.
- ❑ **MMAX:** Maximum principal moment per unit length acting at the mid-surface of the element. Note that by definition, principal moments are oriented such that the associated twisting moment per unit length is zero.
- ❑ **MMIN:** Minimum principal moment per unit length acting at the mid-surface of the element. Note that by definition, principal moments are oriented such that the associated twisting moment per unit length is zero.
- ❑ **V13:** Out-of-plane shear per unit length acting at the mid-surface of the element on the positive and negative 1 faces in the 3-axis direction.
- ❑ **V23:** Out-of-plane shear per unit length acting at the mid-surface of the element on the positive and negative 2 faces in the 3-axis direction.
- ❑ **VMAX:** Maximum principal shear per unit length acting at the mid-surface of the element. Note that by definition, principal shears are oriented on faces of the element such that the associated shears per unit length on perpendicular faces are zero.



✓ Stresses - For shell element internal *stresses,* the possible components are the following:

- ❑ **S11:** Direct stress (force per unit area) acting on the positive and negative 1 faces in the 1-axis direction.
- ❑ **S22:** Direct stress (force per unit area) acting on the positive and negative 2 faces in the 2-axis direction.

- **S12:** Shearing stress (force per unit area) acting on the positive and negative 1 faces *in the 2-axis direction* and acting on the positive and negative 2 faces *in the 1-axis direction*.

- **SMAX:** Maximum principal stress (force per unit area). Note that by definition, principal stresses are oriented such that the associated shearing stress is zero.

- **SMIN:** Minimum principal stress (force per unit area). Note that by definition, principal stresses are oriented such that the associated shearing stress is zero.

- **S13:** Out-of-plane shearing stress (force per unit area) acting on the positive and negative 1 faces in the 3-axis direction.

- **S23:** Out-of-plane shearing stress (force per unit area) acting on the positive and negative 2 faces in the 3-axis direction.

- **SMAXV:** Maximum principal shearing stress (force per unit area). Note that by definition principal shearing stresses are oriented on faces of the element such that the associated shears per unit length on perpendicular faces are zero.

▪ **Contour Range** - The shell element internal forces and stresses are displayed on your screen as colored contours. Ten different contour colors are used. Specify the actual colors used by clicking the **Options menu > Colors > Output** command and editing the colors in the Contours area of the Assign Output Colors form.

In the Contour Range area of the Element Force/Stress Contours for Shells form, you can specify two values:

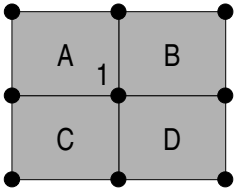
✓ **Min:** Any element with a force or stress less than the value shown here is displayed in the color associated with Min in

the Contours area of the Assign Output Colors form. Note that the color associated with Min is the top color in the form.

✓ **Max:** Any element with a force or stress greater than or equal to the value shown here is displayed in the color associated with Max in the Contours area of the Assign Output Colors form. Note that the color associated with Max is the bottom color in the form.

When you specify the Min and the Max values, the program spaces the intermediate range values equally between the specified Min and Max values.

If you set both the Min and the Max values to zero, the program creates its own range. In that case, the program creates a stress range with rounded off (even) values within which the actual maximum and minimum stresses fit. Note that setting Min and Max to zero is the default.



▪ **Stress Averaging** - Specifies if stress averaging is to be used when displaying the shell element forces or stresses. Consider the four shell elements labeled A, B, C and D shown in the sketch to the left. Those four shell elements all have a common point, labeled 1, in the sketch.

Each of the shell elements has an associated internal force or stress at joint 1. Typically, the forces or stresses at common points in different shell elements are different. The finer your mesh, the closer those values become.

If the force or stress contours are plotted with no stress averaging at the common points, you will typically see abrupt changes in force or stress from element to element. Stress averaging tends to get rid of those abrupt changes in the plot and smoothes the contours.

The program averages the stresses at a point by averaging the stresses from all shell elements that both connect to the point *and are visible in the active window*. Then when the program

plots the stress for a particular shell element, it plots that average stress at the point considered, instead of the actual stress calculated for that shell element at the point.

Do not overlook the implications of the italicized text in the previous paragraph. Assume you are looking at stresses in a location where a wall intersects a floor. Further assume that you are looking at averaged stresses in the floor. If you are viewing the averaged stresses in the floor in a 2D plan view of the floor, only the shell elements that are in the floor, and thus visible in the window, are included in the stress averaging.

If you view the same averaged stresses in a 3D view, where both the wall and the floor are visible, the shell elements from both the floor *and the wall* are included in the stress averaging. Thus the averaged stresses in the floor at the intersection of the floor and the wall will appear differently, depending on whether you are looking at them in a 2D plan view or in a 3D view.

- **Stress Averaging** - In this program, you have the following Stress Averaging option: None, no stress averaging; at All Joints, stress averaging at all joints; or at Selected Joints, stress averaging at specific points you have selected just before plotting the shell forces or stresses.

- **Display on Deformed Shape** - If this check box is checked, the program will display the results on the deformed shape. If it is unchecked, it will not.

Miscellaneous Notes about Shell Element Forces and Stresses

Note that shell element stresses (not forces) actually have different values at the top and bottom of the shell elements (area objects). Thus, depending on which side of the object you are looking at you may see different stresses. Two-dimensional views always look at area objects from the same side. If you want to see stresses on the other side of the area object, view them in a 3D view.

Finally, when shell element forces and stresses are plotted for multi-valued load combinations, the program displays the maximum or minimum value that has the largest absolute value.

Show Member Forces/Stress Diagram > Link Forces

Display link forces directly on your program model by clicking the **Display menu > Show Member Forces/Stress Diagram > Link Forces** command. This command brings up the Member Force Diagram form. This form works as described in the previous subsection entitled "**Show Member Forces/Stress Diagram > Frame/Pier/Spandrel Forces Command**."

Note the following about displayed link element forces:

- When multiple link elements are defined at the same location, the force value displayed on the model is the sum of the forces in all the link elements that exist at that location.
- When link forces are displayed on the model, you can right click on a link element to bring up a tabular display of the forces on that element. When multiple elements are defined in the same location, the tabular display shows the forces for each element separately.
- When forces are plotted on link elements for multi-valued load combinations, the program displays the maximum or minimum values that has the largest absolute value.

Show Energy/Virtual Work Diagram Command

Click the **Display menu > Show Energy/Virtual Work Diagram** command or the button to display energy/virtual work diagrams that can be used as an aid to determine which elements should be stiffened to most efficiently control the lateral displacements of your structure. Following is a little background information.

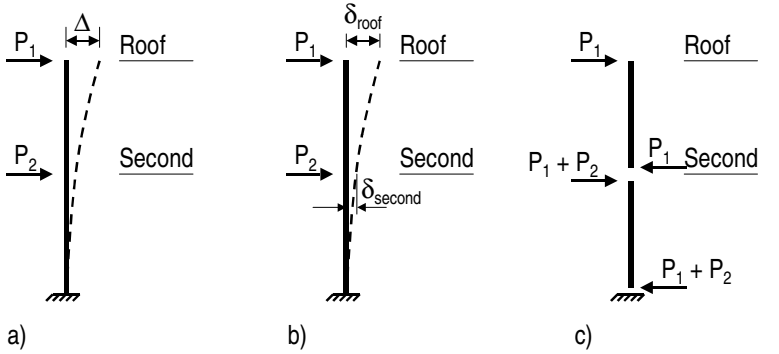


Figure 12-1: *Example of the theory used for energy diagrams*

Consider the two story structure shown in Figure 12-1a that has lateral loads P_1 and P_2 at the Roof and Second story levels, respectively. Also note the displaced shape, Δ, associated with this structure and loading, which is shown by a dashed line.

Note:

The energy diagram is an aid to help you determine which elements should be stiffened to most efficiently control the lateral displacements of your structure.

Now consider the same structure, shown in Figure 12-1b, with a single load P (typically a unit load) applied to it and a resulting displaced shape, δ, shown as a dashed line. Maxwell's Reciprocal Theorem states that:

$$P\Delta = P_1\delta_{roof} + P_2\delta_{second} \qquad \text{Eqn. 12-1}$$

See a structural analysis textbook for details on Maxwell's Reciprocal Theorem.

In this very simple example, Equation 12-1 could be reduced to an element level where the elements are illustrated in Figure 12-1c as shown in Equation 12-2.

$$P\Delta = [P_1\delta_{roof} - P_1\delta_{second}] + [(P_1 + P_2)\delta_{second} - (P_1 + P_2)\delta_{base}] \qquad \text{Eqn. 12-2}$$

Noting that δ_{base} is equal to zero, Equation 12-2 reduces to that shown in Equation 12-3.

$$P\Delta = [P_1\delta_{roof} - P_1\delta_{second}] + [(P_1 + P_2)\delta_{second}] \qquad \text{Eqn. 12-3}$$

In Equation 12-3 the first term in brackets is the energy in the top element and the second term is the energy in the bottom ele-

ment. The energy in both of these elements sums to the total PΔ energy.

When you request that the program shows the energy/virtual work diagram, it reports the equivalent of the values shown in brackets in Equation 12-3 for each element in the structure. Note the following about the energy values that the program reports:

- The energy/virtual work values that the program reports are based on all six degrees of freedom of the element, not just the one degree of freedom described in Figure 12-1 and Equations 12-1 through 12-3.
- The energy/virtual work values that the program reports are determined as follows:
  - ✓ The program determines the energy/virtual work per unit volume associated with each element in the structure.
  - ✓ The program normalizes all of the calculated energy/virtual work values such that the largest one has a value of 100.

Note:

Energy diagrams are normalized.

As previously stated, the energy/virtual work diagrams are helpful as an aid to determine which elements should be stiffened to control lateral displacements in your structure. When you request that the program shows an energy diagram, the Energy Diagram form appears. In this form, input a load case associated with forces and one associated with displacements.

The load case associated with forces is the load case for which you want to control displacements. In the previous discussion, it is the load case shown in Figure 12-1a. The load case associated with displacements is the one associated with Figure 12-1b in the previous example. Typically, this load case consists of one or more unit loads.

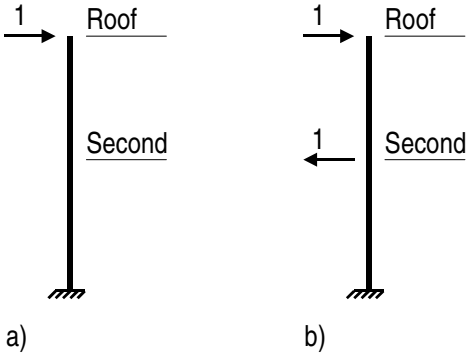


Figure 12-2: *Example displacement load cases for energy diagram*

Figure 12-2 shows a couple of load cases you might use for your displacement load cases. Figure 12-2a shows a load case that is appropriate if you are interested in controlling the roof displacement. Figure 12-2b shows a load case that is appropriate if you are interested in controlling the interstory displacement between the roof and the second story level.

Show Response Spectrum Curves Command

After running a time history analysis, select one or more point objects and click the **Display menu > Show Response Spectrum Curves** command or the button to bring up the Response Spectrum Generation form. In this form, specify the appropriate data to plot various response spectra.

Important Note: The response spectrum curve that you plot using this command is based on a *time history* that you have previously run. It has ***nothing*** to do with any response spectrum analysis that you may have run.

The Response Spectrum Generation form has five separate tabs and two buttons. The two buttons are described first, followed by an explanation of the tabs.

- **Display button:** Click this button to display the plot you have defined in this form.

- **Done button:** Click this button to close the Response Spectrum Generation form.

Define Tab

The Define tab on the Response Spectrum Generation form has the following three areas:

- **Time History Case:** This is the name of a previously run time history case for which you want to create the response spectrum.
- **Choose a Point:** This area displays the labels of the point objects you had selected when you clicked the **Display menu > Show Response Spectrum Curves** command. Highlight a point in this area. The response spectrum is generated based on the time history absolute acceleration response at this highlighted point.
- **Vector Direction:** This is the direction associated with the response spectrum. The choices are X, Y or Z, which are the global axes directions. The response spectrum is based on time history absolute acceleration response in the specified vector direction at the point that is highlighted in the Choose a Point area of the Define tab.

Axes Tab

Te Axes tab of the Response Spectrum Generation form has the following areas:

- **Abscissa:** This is the horizontal axis of the response spectrum. It can be frequency, f, or period, T, where f = 1/ T.
- **Ordinate:** This is the vertical axis of the response spectrum. It can be SD (spectral displacement), SV (spectral velocity), PSV (pseudo-spectral velocity), SA (spectral acceleration) or PSA (pseudo-spectral acceleration).

SD, SV and SA for a given period (frequency) are calculated as the displacement, velocity and acceleration, respectively, of a single degree of freedom system subjected to the output time history acceleration at the highlighted joint (on the Define tab) in the specified vector direction (on the Define tab).

PSV and PSA are defined by Equations 12-4a and 12-4b, respectively.

$$PSV = \frac{2\pi}{T} SD \qquad \text{Eqn. 12-4a}$$

$$PSA = \left(\frac{2\pi}{T}\right)^2 SD \qquad \text{Eqn. 12-4b}$$

Options Tab

The Options tab of the Response Spectrum Generation form has the following areas:

- **Abscissa:** This is the horizontal axis of the response spectrum. It can be plotted with either an Arithmetic scale or a Log scale.

  The Spectrum Widening item widens the plotted peaks of the spectrum. When you specify a percentage value for spectrum widening, each peak in the response spectrum is widened by the specified percentage of the frequency at the peak on both sides of the peak value.

  Important note: The spectrum widening is based on a percentage of the frequency at the peak regardless of whether your abscissa is frequency or period.

  As an example, consider the response spectrum plot shown in Figure 12-3a. Note that this is a plot of period versus spectral acceleration. Note that a peak occurs at 2 seconds in the plot. Now suppose that you create the same response spectrum but this time specify 10% spectrum widening. In that case, the program would artificially widen the peak such that there would be a constant peak value from 1.82 seconds to 2.22 sec-

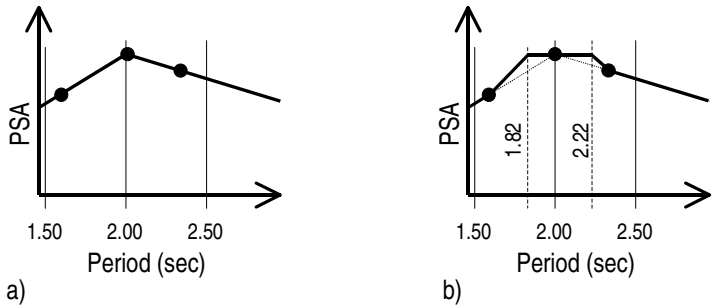


Figure 12-3: *Example of spectrum widening*

Calculation for widened spectrum range shown in (b)
T = 2.00 sec, therefore f = 1 / T = 1 / 2.00 = 0.5 Hz
Calculate 10% spectrum widening frequency: 0.10 f = 0.10 * 0.5 = 0.05 Hz
Calculate frequency range: f_{max} = 0.5 + 0.05 = 0.55 Hz, f_{min} = 0.5 - 0.05 = 0.45 Hz
Calculate period range: T_{min} = 1/ f_{max} = 1/ 0.55 = 1.82 sec, T_{max} = 1/ f_{min} = 1/ 0.45 = 2.22 sec

onds. The response spectrum with spectrum widening is shown in Figure 12-3b. The calculations are shown in the figure.

- **Ordinate:** This is the vertical axis of the response spectrum. It can be plotted with either an Arithmetic scale or a Log scale.

  The Scale Factor item linearly scales the ordinates of the response spectrum. This scale factor can be useful if, for example, you have run your analysis in kip and inch units and you want to see a PSA response spectrum with the acceleration in g (acceleration of gravity) instead of inches/second^2. If this were the case, you would specify the scale factor as 1 / 386.4 = 0.002588.

- **Grid Overlay:** This check box toggles on and off the display of gridlines on the response spectrum plot.

Frequency/Period Tab

The name of this tab is either Frequency or Period, depending on which option is chosen in the Abscissa area of the Axes tab. The following bullet items describe the two areas on this tab.

- **Include Frequencies/Periods:** There are three choices for included frequencies/periods. You can choose any combination of these three choices. The choices are:

- ✓ **Default:** The default frequencies in Hz are: 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9, 1, 1.1, 1.2, 1.3, 1.4, 1.5, 1.6, 1.8, 2, 2.2, 2.4, 2.6, 2.8, 3, 3.3, 3.6, 4, 4.4, 4.7, 5, 5.5, 6, 6.5, 7, 7.5, 8, 8.5, 9, 10, 11, 12, 13, 14, 15, 16.5, 18, 20, 22, 25, 28 and 33. The default periods are equal to one divided by the default frequencies.

  The default frequencies range from 0.2 Hz to 33 Hz. The default periods range from 0.0303 seconds to 5 seconds.

- ✓ **Structural:** The structural periods and frequencies are those calculated by the program for your building.

- ✓ **User:** The user periods and frequencies are those specified by you in the User Frequencies/Periods area of the tab.

- **User Frequencies/Periods:** Specifies user frequencies or periods. To add a user frequency/period, type it in the edit box and click the **Add Value** button.

  To change the value of an existing *user frequency/period,* highlight it in the list box. Note that it appears in the edit box when you highlight it. Change its value in the edit box and click the **Change Value** button.

  To delete an existing *user frequency/period,* highlight it in the list box. Note that it appears in the edit box when you highlight it. Click the **Delete Value** button.

Damping Tab

On the Damping Tab, specify as many different levels of damping as you want. One response spectrum curve is then created for each value of damping. To add a damping level, type it in the edit box and click the **Add Value** button.

To change the value of an existing damping level, highlight it in the list box. Note that it appears in the edit box when you highlight it. Change its value in the edit box and click the **Change Value** button.

To delete an existing damping level, highlight it in the list box. Note that it appears in the edit box when you highlight it. Click the **Delete Value** button.

Show Time History Traces Command

After running a time history analysis, click the **Display menu > Show Time History Traces** command or the button to bring up the Time History Display Definition form. In this form specify the appropriate data to plot various time history curves.

A time history trace is simply a plot of a vertical time history function versus a horizontal time history function. The vertical time history function can be any defined time history function. Although the horizontal time history function defaults to Time, it can be any defined time history function.

Tip:

*You can use the **File menu > Create Video** command to create videos (movies) of your building's real-time deformations in an earthquake. The video is saved as a Windows .avi file.*

You can define the following types of time history functions: input functions, various types of energy functions, base reaction functions, point displacement functions, frame element force functions, link element force or deformation functions and section cut force functions. The time history curves can be plotted as a function versus time (e.g., displacement versus time) or as a function versus another function (e.g., force versus displacement).

Important note: Do not confuse the terminology of time history function implied in the Time History Display Definition form and used in this description with the time history function that is defined using the **Define menu > Time History Function** command. The Define menu command defines a time history input function that is simply one of the types of functions (see previous paragraph) referred to here as a time history function.

When you click the **Display menu > Show Time History Traces** command, the program automatically creates time history display functions for all selected objects. When inside the Time History Display Definition form, you can define additional

time history display functions as desired. Often, if you want to see a time history trace for, say a particular point, it is easiest to select that joint before clicking the command. The program creates the time history display function automatically for the point. You can easily modify the component of displacement displayed if the program did not default to the one you want.

The following bullet items describe various items in the Time History Display Definition form:

- **Time History Case:** This is the name of the time history case for which you want to display a time history trace.

- **Choose Functions:** Specify the vertical and horizontal time history functions that make up your time history trace. A time history trace is simply a plot of a vertical function versus a horizontal function. You can specify multiple vertical functions for a trace but only one horizontal function.

  This area has a number of buttons and list and edit boxes that are described as follows:

  - ✓ **Define Functions button:** This button brings up the Time History Functions form where add new functions, delete existing functions, or modify the components of existing functions.

    Most of the definitions and component information you encounter using this button are self-explanatory. Standard energy formulas are used for the program energy measures. All energies are integrated over the full structure. Following are equations describing each type of energy available in the program that can be plotted as a time history function:

$$IE = \int F(t)\, v(t)\, dt \qquad \text{Eqn. 12-5a}$$

$$KE = \int m\, a(t)\, v(t)\, dt \qquad \text{Eqn. 12-5b}$$

$$PE = \int k\ u(t)\ v(t)\ dt \qquad \text{Eqn. 12-5c}$$

$$MDE = \int c\ v^2(t)\ dt \qquad \text{Eqn. 12-5d}$$

$$NDE = \int [D_{force}\ v_{avg}(t) - D_{stiff}\ u(t)\ v(t)]\ dt \qquad \text{Eqn. 12-5e}$$

$$LE = \int [L_{force}\ v_{avg}(t) - L_{stiff}\ u(t)\ v(t)]\ dt \qquad \text{Eqn. 12-5f}$$

$$EE = IE - KE - PE - MDE - NDE - LE \qquad \text{Eqn. 12-5g}$$

where,

IE = Input energy

KE = Kinetic energy

PE = Potential energy

MDE = Modal damping energy

NDE = Nonlinear damping energy from link elements that are dampers. Note that this excludes the potential energy already accounted for in PE (the subtracted term).

LE = Link element energy (not including dampers). Note that this excludes the potential energy already accounted for in PE (the subtracted term).

EE = Energy error

a = Acceleration

c = Modal damping

D_{force} = Force in link elements that are dampers

D_{stiff} = Stiffness of link elements that are dampers

F = External force

k = Stiffness

L_{force} = Force in link elements (not including dampers)

L_{stiff} = Stiffness of link elements (not including dampers)

m = mass

t = time

u = displacement

v = Velocity

v_{avg} = Average velocity over a time step

Note that for ground-acceleration input, all displacements, velocities and accelerations above are *relative* to the ground motion. The external force is the ground acceleration times the mass of the structure.

- ✓ **List of Functions** - The list box entitled List of Functions lists all of the currently defined time history functions. Highlight a function in this list box and click the **Add** button to move the function into the Vertical Functions list box.

- ✓ **Vertical Functions** - The Vertical Functions list box lists all of the time history functions that will be plotted versus the single specified horizontal function in the current time history trace. To remove a function from the Vertical Functions list box, highlight the function and click the **Remove** button.

- ✓ **Horizontal Function** - The horizontal time history function is selected from the Horizontal Function drop-down box. This box includes all defined time history functions and Time. The default is Time.

Note:

The Selected Function Line Options can be different for each vertical function

- **Selected Function Line Options:** The individual line functions apply separately to each vertical function that is defined. To specify individual line options for a function, first highlight the function in the Choose Functions area and then specify the line options.

  The vertical scale factor is a scale factor for the vertical function only. It does not scale the horizontal function in any way.

- **Time Range:** Specifies any time range to be plotted. This can be useful if you only want to plot a portion of a full time history. This item applies whether you are plotting a function versus time or a function versus another function.

- **Axes Range Override:** This area allows you to change the range plotted for both axes. If you are plotting function versus time, the horizontal axis range override overrides the specified Time Range.

  For example, assume you have 10 seconds of earthquake output. Further assume that you have adjusted the time range to be from 0 to 5 seconds. Now if you override the horizontal axis to be from 0 to 3, only three seconds of the time history output plots. If you instead override the horizontal axis to be from 0 to 10, the time history plots from 0 to 5, as specified from in the Time Range area, but the horizontal axis plots from 0 to 10 seconds. Thus, the earthquake plot fills half of the horizontal axis length.

- **Axes Labels:** The axes labels that you type here appear on the screen plots and on printed plots.

- **Grid Overlay:** This check box toggles on and off the display of gridlines on the time history plot.

- **Display button:** Click this button to display the plot you have defined in this form.

- **Done button:** Click this button to close the Time History Display Definition form.

- **File Menu** - There is a File menu at the top of the window that displays the traces. This File menu allows you to print graphics of the plot or to print tables either to a file or to a printer. Printing time history functions to a file is useful if you then want to take the data in the file and plot it in another program, for example, a spreadsheet.

Set Output Table Mode Command

Click the **Display menu > Set Output Table Mode** command or the button to open the Display Output Tables form. In this form, select the type of information you want to include in the output tables. The options available are self-explanatory. Selecting an option will include it in the output table.

Note that some of the options may be inactive. This is because the current model type does not include those options. For example, the Spring Forces option will be inactive if no springs have been defined in the model.

Select Loads Button

Click the Select Loads button to indicate for which load case you want to display or print the output data. You can select one or more load cases at the same time.

Section Cut Button

You can select the type of Section Cuts that have been defined previously using the **Define menu> Section Cuts** command to be included in the output tables. This can be accomplished by clicking the Section Cut button. This will open the Select Section Cuts dialogue box. Click on the Section Cuts to be included in the output and click the **OK** button.

Design Menu

Overview

The Design menu serves as your gateway to the integrated design postprocessors that are a part of your program package. The design postprocessors available are:

- Steel Frame Design
- Concrete Frame Design
- Composite Beam Design
- Steel Joist Design
- Shear Wall Design

For each of the design postprocessors, you can access the following types of commands from the Design menu.

- Review and/or select design load combinations.
- Review and/or select overwrites.

- Start the design or check of the structure.
- Perform interactive design.
- Display input and output design information on the model.
- Perform various other tasks specific to the various design post-processors.

Note that you use the **File menu > Print Tables** command to print design output from the various design postprocessors in a tabular form.

Background

The frame design procedure identifies the design post processor that will design a particular line object. The design procedure is reported in the Line Information form, which appears when you right click on a line object.

The design procedure is always one of the following four items:

- Steel Frame Design
- Concrete Frame Design
- Composite Beam Design
- Null (no design)

See the subsequent section entitled "Overwrite Frame Design Procedure Command" for more information.

Default Design Procedure Assignments

The default design procedure assignments are determined by the program as follows:

- **Null:** If the line object is *not* assigned a frame section property, its default design procedure is Null.
- **Concrete Frame Design:** If the line object is assigned a frame section property that has a *concrete* material property, its default design procedure is Concrete Frame Design.

Tip:

If you have modeled a floor or roof with slightly sloping beams and you want to design the beams using the Composite Beam Design postprocessor, first make all of the beams horizontal. Do this using the ***Edit menu > Align Points/ Lines/ Edges*** *command and selecting the "Align to Z Coordinate of" option.*

- **Composite Beam Design:** If the line object is assigned a frame section property that has a *steel* material property and it meets all of the criteria listed in the bullets that follow, its default design procedure is Composite Beam Design.

  - ✓ The line type is Beam; that is, the line object is horizontal.
  - ✓ The frame element is oriented with its positive local 2-axis in the same direction as the positive global Z-axis (vertical upward).
  - ✓ The frame element has I-section or channel section properties.
  - ✓ The M3 (major) moment is released at each end of the frame element; that is, it is pinned.
  - ✓ A deck property (not slab property) is assigned to an area object located on top of the beam. The beam and the area object must be in the same plane.

- **Steel Joist Design:** If the line object is assigned a frame section property that has a *steel* material property and it meets all of the criteria listed in the bullets that follow, its default design procedure is Steel Joist Design.

  - ✓ The line type is Beam; that is, the line object is horizontal.
  - ✓ The frame element is oriented with its positive local 2-axis in the same direction as the positive global Z-axis (vertical upward).
  - ✓ The frame element has joist section properties.
  - ✓ The M3 (major) moment is released at each end of the frame element; that is, it is pinned.

- **Steel Frame Design:** If the line object is assigned a frame section property that has a *steel* material property and it does not qualify to have its design procedure set to Composite Beam Design, its default design procedure is Steel Frame Design.

Steel Frame Design Command

Clicking the **Design menu > Steel Frame Design** command or the button activates a drop-down menu of the commands that control the Steel Frame Design postprocessor. This section describes those menu commands.

Steel Frame Design > Select Design Group Command

Note:

Frame elements designed as a group are all given the same section size

In steel frame design, you have the option of grouping elements for design. When you specify a group for design, all elements in the group are given the same section. Note the following information related to using groups for steel frame design.

- Define the groups in the usual way; that is, by selecting the frame elements and clicking the **Assign menu > Group Names** command.
- After the group has been defined, use the **Design menu > Steel Frame Design > Select Design Group** command to designate that the group is to be used as a design group.
- Designing with groups only works if you have assigned auto select sections to the frame elements. Typically, you would assign the same auto select section to each frame element in the group, although this is not absolutely necessary. Any frame elements in a design group not assigned an auto select section are ignored for group design and are designed individually.

Tip:

Frame elements designed as a part of a group must be assigned auto select section lists.

Steel Frame Design > Select Design Combo Command

Click the **Design menu > Steel Frame Design > Select Design Combo** command to open the Design Load Combinations Selection form. Here you can review the default steel frame design load combinations defined by the program, or you can designate your own design load combinations.

In the form all of the available design load combinations are listed in the List of Combos list box. The design load combinations actually used in the design are listed in the Design Combos list box. You can use the **Add** button and the **Remove** button to move load combinations into and out of the Design Combos list box. Use the **Show** button to see the definition of a design load

combination. All three buttons act on the highlighted design load combination. You can use the Ctrl and Shift keys to make multiple selections in this form for use with the **Add** and **Remove** buttons, if desired.

The default steel frame design load combinations have names such as DSTL1.

Steel Frame Design > View/Revise Overwrites Command

Tip:

The steel frame design overwrites only apply to the frame elements to which they are specifically assigned.

Use the **Design menu > Steel Frame Design > View/Revise Overwrites** command to review or change the steel frame overwrites. You may not need to assign any steel frame overwrites; however, the option is available.

The steel frame design overwrites are basic properties that apply only to the frame elements to which they are specifically assigned. Some of the default overwrite values are based on steel frame preferences. Thus, you should define the preferences before defining the overwrites (and, of course, before designing or checking any steel frame members).

Select one or more frame elements for which you want to specify overwrites. To change an overwrite, check the check box to the left of the overwrite name and then click in the cell to the right of the overwrite name to change the overwrite.

You ***must*** check the box to the left of an overwrite item for that item to be changed in the overwrites. If the check box for an item is not checked when you click the **OK** button to exit the overwrites form, no changes are made to the item. This is true whether you have one frame element selected or multiple frame elements selected.

Steel Frame Design > Set Lateral Displacement Targets Command

Click the **Design menu > Steel Frame Design > Set Lateral Displacement Targets** command to open the Lateral Displacement Targets form where you can specify displacement targets for various load cases. Our intent is that you pick a point, typically at the roof level of your building, and specify a maximum displacement target (in any direction) for one or more load cases.

In the Loads area of the form, specify the Load Case for which you want to optimize the displacement, the location where you are specifying the displacement (by point ID and story level) and the target displacement. When those parameters have been specified to your satisfaction in the Loads area, click the **Add** button. Specify as many load cases and displacements as you want.

Tip:

Frame elements must have auto select list assignments to be designed using displacement optimization.

If you want to modify an existing displacement target specification, highlight it in the Loads area. Note that the data for the specification appears in the edit boxes at the top of the Loads area. Modify the data in the boxes as desired. Then click the **Modify** button.

If you want to delete an existing displacement target specification, highlight it in the Loads area. Note that the data for the specification appears in the edit boxes at the top of the Loads area. Then click the **Delete** button.

Note the following about the displacement optimization performed by the program:

- The program predicts which members should be increased in size to control the displacements based on the energy per unit volume in the members. The members with more energy per unit volume are increased in size a larger percentage than those with smaller energies per unit volume. Some members with small energy per unit volume may be decreased in size if they are still acceptable for strength considerations.

- You must have auto select lists assigned to the frame elements for the displacement optimization to do anything. When the program goes to increase or decrease a section size, it uses the available sizes in the auto select list.

- There is no guarantee that you will reach your displacement target just because you specified this option. You always need to rerun your *analysis* with the new section sizes to see what your new displacements are. Displacement optimization is an iterative process that typically requires you to rerun the analysis and design multiple times.

- If the appropriate sections are not in your auto select lists, you may never reach your displacement target, regardless of how many times you iterate by rerunning the analysis and design.

Steel Frame Design > Start Design/Check of Structure Command

To run a steel frame design, click **Design menu > Steel Frame Design > Start Design/Check of Structure**. This option will not be available if you have not first run a building analysis. It will also be unavailable if no frame elements with the Steel Frame design procedure are in the model.

If you have selected frame elements when you click this command, only the selected frame elements are designed. If no frame elements are selected when you click this command, all steel frame elements with the Steel Frame design procedure are designed. The results are displayed on screen in the active window.

Steel Frame Design > Interactive Steel Frame Design Command

Interactive steel frame design allows you to review the design results for any frame element and to interactively change the design overwrites and immediately view the results again.

Right click on a frame element while the design results are displayed on it to enter the interactive design mode and interactively design the element. If you are not currently displaying design results, click the **Design menu > Steel Frame Design > Interactive Steel Frame Design** command and then right click a frame element to enter the interactive design mode for that element.

Steel Frame Design > Display Design Info Command

You can review some of the results of the steel frame design directly on the program model using the **Design menu > Steel Frame Design > Display Design Info** command. The types of things you can display include design sections, unbraced lengths, effective length factors, allowable stresses, and stress ratio information. Clicking this command brings up the Display Design Results form, which allows you to choose Design Output or Design Input data types.

Steel Frame Design > Make Auto Select Section Null Command

Tip:

You normally use the Make Auto Select Section Null feature near the end of the iterative design process.

The **Design menu > Steel Frame Design > Make Auto Select Section Null** command is used to remove auto select section lists from selected frame elements. Typically you should remove auto select lists from all frame elements near the end of the iterative design process so that your final design iteration is performed using the actual frame sections assigned, not auto select sections.

Setting the auto select section to null does not change the current design section for the frame element.

The **Make Auto Select Section Null** command only works on a selection that you make. Thus you should select the elements for which you want to make the auto select sections null before executing this command. If you do not select any elements, this command will not be available. Often you may want to select all elements before executing this command.

The **Make Auto Select Section Null** command is not active until the first design has been run. If you have not yet run a design and you want to remove the auto select property, use the **Assign menu > Frame/Line > Frame Section** command to change the section property.

Steel Frame Design > Change Design Section Command

Note:

You can change the element design sections and rerun the design as many times as you want without rerunning the analysis.

After you have run a steel frame design, you may want to change the design section property assigned to one or more frame elements and then rerun the design without first rerunning the analysis. You can use the **Design menu > Steel Frame Design > Change Design Section** command to change the design section property, and then use the **Design menu > Steel Frame Design > Start Design/Check of Structure** to rerun the design.

The **Change Design Section** command only works on a selection that you make. Thus you should select the elements for which you want to change the design sections before executing this command. If you do not select any elements, this command will not be available.

The **Change Design Section** command only changes the design section for the frame element. The forces used in the design are

not based on this new section size but are instead based on whatever section was used in the last analysis.

Recall, however, that the design section property is used for the next analysis section property. Thus changing the design section property also changes the next analysis section property. If an auto select section is assigned to a frame element, you can control the section property used for that frame element in the next analysis by setting the design section property to the desired section using the **Change Design Section** command and then rerunning the analysis.

Steel Frame Design > Reset Design Section to Last Analysis Command

In some instances you may change your design section several times and then decide that you want to set the design section for one or more frame elements back to the last used analysis section. The **Design menu > Steel Frame Design > Reset Design Section to Last Analysis** command gives you a quick and easy way of doing this.

The **Reset Design Section to Last Analysis** command only works on a selection that you make. Thus, before implementing this command, select the elements for which you want to reset the design sections. If you do not select any elements, this command will not be available.

Steel Frame Design > Verify Analysis vs Design Section Command

When the iterative design process is complete, the last used analysis section property for a frame element and the current design section property for that frame element should be the same. If this is not the case, the frame element may not have been designed for the correct forces. The **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command is useful for verifying that the last used analysis section and the current design section are the same for all steel frame elements in the model.

When you execute the **Verify Analysis vs Design Section** command, the program tells you how many frame elements with the Steel Frame design procedure have different analysis and design sections, and then selects those frame elements if you tell it to do so. Typically you might use this command after you have run what you believe is your last design iteration just to verify that the analysis and design properties used are consistent.

It is not necessary to make a selection before using the **Verify Analysis vs Design Section** command. This command automatically checks all frame sections with the Steel Frame design procedure.

Steel Frame Design > Verify All Members Passed Command

The **Verify All Members Passed** command works in conjunction with the **Start Design/Check of Structure** command. Thus, you must first run an analysis and use the **Start Design/Check of Structure** command before using the **Verify All Members Passed** command. When this command is used, the program reports if structural members have passed the stress/capacity check.

Steel Frame Design > Reset All Steel Overwrites Commands

The **Design menu > Steel Frame Design > Reset All Steel Overwrites** command resets the overwrites for all frame sections with the Steel Frame design procedure to their default values. It is not necessary to make a selection before using the **Reset All Steel Overwrites** command. This command automatically applies to all frame sections with the Steel Frame design procedure.

Resetting your overwrites will reduce the size of your program database (*.edb) file.

Steel Frame Design > Delete Steel Design Results Command

The **Design menu > Steel Frame Design > Delete Steel Design Results** command deletes all of the steel frame design results. It

is not necessary to make a selection before using the **Delete Steel Design Results** command. This command automatically applies to all frame sections with the Steel Frame design procedure.

Deleting your steel frame design results will reduce the size of your program database (*.edb) file. Note that deleting your steel design results does not delete your current design section (next analysis section).

Concrete Frame Design Command

Clicking the **Design menu > Concrete Frame Design** command or the button activates a drop-down menu of the commands that control the Concrete Frame Design postprocessor. This section describes those menu commands.

Concrete Frame Design > Select Design Combo Command

Click the **Design menu > Concrete Frame Design > Select Design Combo** command to open the Design Load Combinations Selection form. In this form you can review the default concrete frame design load combinations defined by the program or you can designate your own design load combinations.

In the form all of the available design load combinations are listed in the List of Combos list box. The design load combinations actually used in the design are listed in the Design Combos list box. Use the **Add** button and the **Remove** button to move load combinations into and out of the Design Combos list box. Use the **Show** button to see the definition of a design load combination. All three buttons act on the highlighted design load combination. Use the Ctrl and Shift keys to make multiple selections in this form for use with the **Add** and **Remove** buttons, if desired.

The default concrete frame design load combinations have names such as DCON1.

Concrete Frame Design > View/Revise Overwrites Command

Tip:

The concrete frame design overwrites only apply to the frame elements to which they are specifically assigned.

Use the **Design menu > Concrete Frame Design > View/ Revise Overwrites** command to review or change the concrete frame overwrites. You may not need to assign any concrete frame overwrites; however, the option is available.

The concrete frame design overwrites are basic properties that apply only to the frame elements to which they are specifically assigned. Some of the default overwrite values are based on concrete frame preferences. Thus, you should define the preferences before defining the overwrites (and, of course, before designing or checking any concrete frame members).

Select one or more frame elements for which you want to specify overwrites. To change an overwrite, check the check box to the left of the overwrite name and then click in the cell to the right of the overwrite name to change the overwrite.

You must check the box to the left of an overwrite item for that item to be changed in the overwrites. If the check box for an item is not checked when you click the **OK** button to exit the overwrites form, no changes are made to the item. This is true whether you have one frame element selected or multiple frame elements selected.

Concrete Frame Design > Start Design/Check of Structure Command

To run a concrete frame design simply click **Design menu > Concrete Frame Design > Start Design/Check of Structure**. This option will not be available if you have not first run a building analysis. It will also be unavailable if there are no frame elements with the Concrete Frame design procedure in the model.

If you have selected frame elements when you click this command, only the selected frame elements are designed. If no frame elements are selected when you click this command, all concrete frame elements with the Concrete Frame design procedure are designed.

Concrete Frame Design > Interactive Concrete Frame Design Command

Interactive concrete frame design allows you to review the design results for any frame element and to interactively change the design overwrites and immediately view the results.

Right click on a frame element while the design results are displayed on it to enter the interactive design mode and interactively design the element. If you are not currently displaying design results, click the **Design menu > Concrete Frame Design > Interactive Concrete Frame Design** command and then right click a frame element to enter the interactive design mode for that element.

Concrete Frame Design > Display Design Info

You can review some of the results of the concrete frame design directly on the program model using the **Design menu > Concrete Frame Design > Display Design Info** command. The types of things you can display include design sections, unbraced lengths and longitudinal reinforcing.

Concrete Frame Design > Change Design Section

After you have run a concrete frame design, you may want to change the design section property assigned to one or more frame elements and then rerun the design without first rerunning the analysis. You can use the **Design menu > Concrete Frame Design > Change Design Section** command to change the design section property and then use the **Design menu > Concrete Frame Design > Start Design/Check of Structure** to rerun the design.

Note:

You can change the element design sections and rerun the design as many times as you want without rerunning the analysis.

The **Change Design Section** command only works on a selection that you make. Thus you should select the elements for which you want to change the design sections before executing this command. If you do not select any elements, this command will not be available.

The **Change Design Section** command only changes the design section for the frame element. The forces used in the design are

not based on this new section size but are instead based on the section used in the last analysis.

Recall, however, that the design section property is used for the next analysis section property. Thus changing the design section property also changes the next analysis section property. If an auto select section is assigned to a frame element, control the section property used for that frame element in the next analysis by setting the design section property to the desired section using the **Change Design Section** command and then rerunning the analysis.

Concrete Frame Design > Reset Design Section to Last Analysis Command

In some instances you may change your design section several times and then decide that you want to set the design section for one or more frame elements back to the last used analysis section. The **Design menu > Concrete Frame Design > Reset Design Section to Last Analysis** command gives you a quick and easy way of doing this.

The **Reset Design Section to Last Analysis** command only works on a selection that you make. Thus, before implementing this command, select the elements for which you want to reset the design sections. If you do not select any elements, this command will not be available.

Concrete Frame Design > Verify Analysis vs Design Section Command

When the iterative design process is complete, the last used analysis section property for a frame element and the current design section property for that frame element should be the same. If this is not the case, the frame element may not have been designed for the correct forces. The **Design menu > Concrete Frame Design > Verify Analysis vs Design Section** command is useful for verifying that the last used analysis section and the current design section are the same for all concrete frame elements in the model.

When you execute the **Verify Analysis vs Design Section** command the program tells you how many frame elements with the Concrete Frame design procedure have different analysis and design sections and then selects those frame elements, if you ask it to do so. Typically you might use this command after you have run what you believe is your last design iteration to verify that the analysis and design properties used are consistent.

It is not necessary to make a selection before using the **Verify Analysis vs Design Section** command. This command automatically checks all frame sections with the Concrete Frame design procedure.

Concrete Frame Design > Reset All Concrete Overwrites Command

The **Design menu > Concrete Frame Design > Reset All Concrete Overwrites** command resets the overwrites for all frame sections with the Concrete Frame design procedure to their default values. It is not necessary to make a selection before using the **Reset All Concrete Overwrites** command. This command automatically applies to all frame sections with the Concrete Frame design procedure.

Resetting your overwrites will reduce the size of your program database (*.edb) file.

Concrete Frame Design > Delete Concrete Design Results Command

The **Design menu > Concrete Frame Design > Delete Concrete Design Results** command deletes all of the concrete frame design results. It is not necessary to make a selection before using the **Delete Concrete Design Results** command. This command automatically applies to all frame sections with the Concrete Frame design procedure.

Deleting your concrete frame design results will reduce the size of your program database (*.edb) file. Note that deleting your concrete design results does not delete your current design section (next analysis section).

Composite Beam Design Command

Clicking the **Design menu > Composite Beam Design** command or the button activates a drop-down menu of the commands that control the Composite Beam Design postprocessor. This section describes those menu commands.

Composite Beam Design > Select Design Group Command

Note:

Beams designed as a group are all given the same beam size; however, each beam in the group may have a different number of shear connectors and different camber.

In composite beam design, you have the option of grouping elements for design. When you specify a group for design, all elements in the group are given the same beam size. Note the following information related to using groups for design of composite beams.

- Define the groups in the usual way; that is, by selecting the beam elements and clicking the **Assign menu > Group Names** command.
- After the group has been defined, use the **Design menu > Composite Beam Design > Select Design Group** command to designate that the group is to be used as a design group.
- Designing with groups only works if you have assigned auto select sections to the beams. Typically you would assign the same auto select section to each beam in the group, although this is not absolutely necessary. Any beams in a design group not assigned an auto select section are ignored for group design and are designed individually.

Tip:

Beams designed as a part of a group must be assigned auto select section lists.

Note that when beams are designed in a group, they will all have the same beam size, but the shear connectors and camber may be different.

Composite Beam Design > Select Design Combo Command

Clicking the **Design menu > Composite Beam Design > Select Design Combo** command brings up the Design Load Combinations Selection form. In this form, review the default composite beam design load combinations defined by the program or designate your own design load combinations. Note that for com-

posite beam design, separate design load combinations are specified for construction loading, final loading considering strength, and final loading considering deflection. Each of those types of design load combinations is specified on a separate tab of the form.

In the form, all of the available design load combinations are listed in the List of Combos list box. The design load combinations actually used in the design are listed in the Design Combos list box. You can use the **Add** button and the **Remove** button to move load combinations into and out of the Design Combos list box. Use the **Show** button to see the definition of a design load combination. All three buttons act on the highlighted design load combination. Use the Ctrl and Shift keys to make multiple selections in this form for use with the **Add** and **Remove** buttons, if desired.

The default composite beam design load combinations have names such as DCMPC1. The naming conventions is described as follows:

- **DCMPCn:** The D stands for Design. The CMP stands for composite. The last C stands for construction. The n item is a number. Design load combinations with this type of designation are the program default for construction loading in composite design.

- **DCMPSn:** The D stands for Design. The CMP stands for composite. The S stands for strength. The n item is a number. Design load combinations with this type of designation are the program default for strength considerations under final loading in composite design.

- **DCMPDn:** The D stands for Design. The CMP stands for composite. The last D stands for deflection. The n item is a number. Design load combinations with this type of designation are the program default for deflection considerations under final loading in composite design.

Composite Beam Design > View/Revise Overwrites Command

Use the **Design menu > Composite Beam Design > View/Revise Overwrites** command to review or change the composite beam overwrites. You may not need to assign any composite beam overwrites; however, the option is available. If you are using cover plates or user-defined shear connector patterns, you must assign them through the overwrites. This is the only place available to assign those items.

Tip:

The composite beam design overwrites only apply to the beams to which they are specifically assigned.

The composite beam design overwrites are basic properties that apply only to the beams to which they are specifically assigned. Some of the default overwrite values are based on composite beam preferences. Thus define the preferences before defining the overwrites (and before designing or checking any composite beam).

Select one or more beams for which you want to specify overwrites. To change an overwrite, check the check box to the left of the overwrite name and then click in the cell to the right of the overwrite name. When you click in the cell, you activate a drop down box where you can select a choice or you are able to type data into the cell. Note that information about each item in the overwrites is provided at the bottom of the form when you click on the item.

You must check the box to the left of an overwrite item for that item to be changed in the overwrites. If the check box for an item is not checked when you click the **OK** button to exit the overwrites form, no changes are made to the item. This is true whether you have one beam selected or multiple beams selected.

Composite Beam Design > Start Design Using Similarity Command

If you are designing using similarity, the program assumes that if a composite beam is located at a story designated as similar to a master story, then that composite beam has the same composite beam size as the composite beams of the master story (set a story Similar To a master story in the Story Data; see the **Edit menu > Edit Story Data > Edit Story** command). Thus, the program designs the composite beam at the master story and then copies that

design to the other stories designated as similar to the master story. Note that in the composite beam overwrites, you can specify that similarity be ignored for a composite beam; in which case the composite beam will be designed, not copied from a master story level.

To run a composite beam design and take advantage of similarity, click the **Design menu > Composite Beam Design > Start Design Using Similarity** command. It is **not** necessary to run the full building analysis (i.e., use the **Analyze menu > Run Analysis** command) before using the **Start Design Using Similarity** command. However, if a full building analysis has not been run when the **Start Design Using Similarity** command is used, the program will run an analysis on a story-by-story basis, analyzing only those stories for which analysis is needed to perform the design.

The story-by-story analysis is much quicker than the full building analysis. However, if, for example, columns from one story frame down onto beams in another story, the loads in those columns will not be accounted for in a single-story analysis.

Note that the final design should always be based on a full building analysis, not single-story analyses. Also, final design should use the **Design menu > Composite Beam Design > Start Design Without Similarity** command.

Composite Beam Design > Start Design Without Similarity Command

To run a composite beam design that excludes the similarity features described in the previous section, click the **Design menu > Composite Beam Design > Start Design Without Similarity** command. Note that for this option to be available, you must first use the **Analyze menu > Run Analysis** command to run a full building analysis.

If composite beams have been selected when you click this command, only the selected beams are designed. If no beams are selected when you click this command, all composite beams are designed.

Composite Beam Design > Interactive Composite Beam Design Command

Interactive composite beam design is a powerful feature that allows you to review the design results for any composite beam and to interactively change the design assumptions and immediately view the results.

Right click on a beam while the design results are displayed on it to enter the interactive design mode and interactively design the beam. If you are not currently displaying design results you can click the **Design menu > Composite Beam Design > Interactive Composite Beam Design** command and then right click a beam to enter the interactive design mode for that beam.

Composite Beam Design > Display Design Info Command

Review some of the results of the composite beam design directly on the program model using the **Design menu > Composite Beam Design > Display Design Info** command. The types of things you can display are beam labels and associated design group names; design sections together with connector layout, camber and end reactions; and stress ratio information.

Composite Beam Design > Make Auto Select Section Null Command

The **Design menu > Composite Beam Design > Make Auto Select Section Null** command is used to remove auto select section lists from selected beams. Typically you should remove auto select lists from all beams near the end of the iterative design process so that your final design iteration is performed using actual beam sections assigned, not auto select sections.

Tip:

You normally use the Make Auto Select Section Null feature near the end of the iterative design process.

Setting the auto select section to null does not change the current design section for the beam.

The **Make Auto Select Section Null** command only works on a selection that you make. Thus, select the elements for which you wan to make the auto select sections null before executing this command. If you do not select any elements, this command will not be available. Often you may want to select all elements before executing this command.

The **Make Auto Select Section Null** command is not active until the first design has been run. If you have not yet run a design and you want to remove the auto select property, use the **Assign menu > Frame/Line > Frame Section** command to change the section property.

Composite Beam Design > Change Design Section Command

After you have run a composite beam design, you may want to change the design section property assigned to one or more beams and then rerun the design without first rerunning the analysis. You can use the **Design menu > Composite Beam Design > Change Design Section** command to change the design section property and then rerun the design.

The **Change Design Section** command only works on a selection that you make. Thus you should select the elements for which you want to change the design sections before executing this command. If you do not select any elements, this command will not be available.

Note:

You can change the element design sections and rerun the design as many times as you want without rerunning the analysis.

The **Change Design Section** command only changes the design section for the beam. The forces used in the design are not based on this new section size but are instead based on the section used in the last analysis.

Recall, however, that the design section property is used for the next analysis section property. Thus, changing the design section property also changes the next analysis section property. If an auto select section is assigned to a beam, control the section property used for that beam in the next analysis by setting the design section property to the desired beam size using the **Change Design Section** command and then rerunning the analysis.

Composite Beam Design > Reset Design Section to Last Analysis Command

In some instances you may change your design section several times and then decide that you want to set the design section for one or more beams back to the last used analysis section. The

Design menu > Composite Beam Design > Reset Design Section to Last Analysis command gives you a quick and easy way of doing this.

The **Reset Design Section to Last Analysis** command only works on a selection that you make. Thus, before implementing this command, select the elements for which you want to reset the design sections. If you do not select any elements, this command will not be available.

Composite Beam Design > Verify Analysis vs Design Section Command

When the iterative design process is complete, the last used analysis section property for a beam and the current design section property for a beam should be the same. If this is not the case, the beam may not have been designed for the correct forces. The **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command is useful for verifying that the last used analysis section and the current design section are the same for all composite beams in the model.

When you execute the **Verify Analysis vs Design Section** command, the program tells you how many beams have different analysis and design sections and then selects those beams, if you tell it to do so. Typically you might use this command after you have run what you believe is your last design iteration to verify that the analysis and design properties used are consistent.

It is not necessary to make a selection before using the **Verify Analysis vs Design Section** command. This command automatically checks all composite beam sections.

Composite Beam Design > Verify All Members Passed Command

The **Verify All Members Passed** command reports how many structural members have passed the stress/capacity check.

Composite Beam Design > Reset All Composite Beam Overwrites Command

The **Design menu > Composite Beam Design > Reset All Composite Beam Overwrites** command resets the composite beam overwrites for all composite beam sections back to their default values. It is not necessary to make a selection before using the **Reset All Composite Beam Overwrites** command. This command automatically applies to all composite beam sections.

Resetting your composite beam overwrites will reduce the size of your program database (*.edb) file.

Composite Beam Design > Delete Composite Beam Design Results Command

The **Design menu > Composite Beam Design > Delete Composite Beam Design Results** command deletes all of the composite beam results. It is not necessary to make a selection before using the **Delete Composite Beam Design Results** command. This command automatically applies to all composite beam sections.

Deleting your composite beam results will reduce the size of your program database (*.edb) file. Note that deleting your composite beam design results does not delete your current design section (next analysis section).

Steel Joist Design Command

Clicking the **Design menu > Steel Joist Design** command or the button activates a drop-down menu of the commands that control the Steel Joist Design postprocessor. This section describes those menu commands.

Steel Joist Design > Select Design Group Command

In steel joist design, you have the option of grouping elements for design. When you specify a group for design, all elements in the group are given the same joist size. Note the following information related to using groups for design of steel joists.

Tip:

Joists designed as a part of a group must be assigned auto select section lists.

- Define the groups in the usual way; that is, by selecting the joist elements and clicking the **Assign menu > Group Names** command.
- After the group has been defined, use the **Design menu > Steel Joist Design > Select Design Group** command to designate that the group is to be used as a design group.
- Designing with groups only works if you have assigned auto select sections to the joists. Typically, you would assign the same auto select section to each joist in the group, although this is not absolutely necessary. Any joists in a design group not assigned an auto select section are ignored for group design and are designed individually.

Steel Joist Design > Select Design Combo Command

Clicking the **Design menu > Steel Joist Design > Select Design Combo** command brings up the Design Load Combinations Selection form. In this form, review the default steel joist design load combinations defined by the program or designate your own design load combinations.

In the form, all of the available design load combinations are listed in the List of Combos list box. The design load combinations actually used in the design are listed in the Design Combos list box. You can use the **Add** button and the **Remove** button to move load combinations into and out of the Design Combos list box. Use the **Show** button to see the definition of a design load combination. All three buttons act on the highlighted design load combination. Use the Ctrl and Shift keys to make multiple selections in this form for use with the **Add** and **Remove** buttons.

Steel Joist Design > View/Revise Overwrites Command

Use the **Design menu > Steel Joist Design > View/Revise Overwrites** command to review or change the steel joist overwrites. You may not need to assign any steel joist overwrites; however, the option is available.

Select one or more beams for which you want to specify overwrites. To change an overwrite, check the check box to the left of the overwrite name and then click in the cell to the right of the

overwrite name. When you click in the cell, a drop-down box is activated where you can select a choice or type data into the cell. Note that information about each item in the overwrites is provided at the bottom of the form when you click on the item.

You must check the box to the left of an overwrite item for that item to be changed in the overwrites. If the check box for an item is not checked when you click the **OK** button to exit the overwrites form, no changes are made to the item. This is true whether you have one joist selected or multiple joists selected.

Steel Joist Design > Start Design Using Similarity Command

When you design using similarity, the program assumes that if a joist is located at a story designated as similar to a master story, that joist has the same joist size as the joists of the master story (set a story Similar To a master story in the Story Data; see the **Edit menu > Edit Story Data > Edit Story** command). Thus, the programs designs the joist at the master story and then copies that design to the other stories designated as similar to the master story. Note that in the steel joist overwrites, you can specify that similarity be ignored for a joist, in which case the joist will be designed, not copied from a master story level.

To run a steel joist design and take advantage of similarity, click the **Design menu > Steel Joist Design > Start Design Using Similarity** command. It is not necessary to run a full building analysis (i.e., use the **Analyze menu > Run Analysis** command) before using the **Start Design Using Similarity** command. However, if a full building analysis has not been run when the **Start Design Using Similarity** command is used, the program will run the analysis on a story-by-story basis, analyzing only those stories for which analysis is needed to perform the design.

The story-by-story analysis is much quicker than the full building analysis. However, if, for example, columns from one story frame down onto beams in another story, the loads in those columns will not be accounted for in the single-story analysis.

Note that final design should always be based on a full building analysis, not single-story analyses. Also, final design should use

the **Design menu > Steel Joist Design > Start Design Without Similarity** command.

Steel Joist Design > Start Design Without Similarity Command

To run a steel joist design that excludes the similarity features described in the previous section, click the **Design menu > Steel Joist Design > Start Design Without Similarity** command. Note that for this option to be available, you must first run a full building analysis using the **Analyze menu > Run Analysis** command.

If steel joists have been selected when you click this command, only the selected joists are designed. If no joists are selected when you click this command, all steel joists are designed.

Steel Joist Design > Interactive Steel Joist Design Command

Interactive steel joist design is a powerful feature that allows you to review the design results for any steel joist and to interactively change the design assumptions and immediately view the results.

Right click on a joist while the design results are displayed on it to enter the interactive design mode and interactively design the beam. If you are not currently displaying design results click the **Design menu > Steel Joist Design > Interactive Steel Joist Design** command and then right click a joist to enter the interactive design mode for that joist.

Steel Joist Design > Display Design Info Command

Review some of the results of the composite beam design directly on the program model using the **Design menu > Steel Joist Design > Display Design Info** command. The types of things you can display are beam labels and associated design group names; design sections together with end reactions; and design ratio information.

Steel Joist Design > Make Auto Select Section Null Command

The **Design menu > Steel Joist Design > Make Auto Select Section Null** command is used to remove auto select section lists from selected joists. Typically, you should remove auto select lists from all joists near the end of the iterative design process so that your final design iteration is performed using actual joist sections assigned, not auto select sections.

Tip:

You normally use the Make Auto Select Section Null feature near the end of the iterative design process.

Setting the auto select section to null does not change the current design section for the beam.

The **Make Auto Select Section Null** command only works on a selection that you make. Thus, select the elements for which you want to make the auto select sections null before executing this command. If you do not select any elements, this command will not be available. Often you may want to select all elements before executing this command.

The **Make Auto Select Section Null** command is not active until the first design has been run. If you have not yet run a design and you want to remove the auto select property, use the **Assign menu > Frame/Line > Frame Section** command to change the section property.

Steel Joist Design > Change Design Section Command

Note:

You can change the element design sections and rerun the design as many times as you want without rerunning the analysis.

After you have run a steel joist design, you may want to change the design section property assigned to one or more joists and then rerun the design without first rerunning the analysis. Use the **Design menu > Steel Joist Design > Change Design Section** command to change the design section property and then rerun the design.

The **Change Design Section** command only works on a selection that you make. Thus you should select the elements for which you want to change the design sections before executing this command. If you do not select any elements, this command will not be available.

The **Change Design Section** command only changes the design section for the joist. The forces used in the design are not based

on this new section size but are instead based on the section used in the last analysis.

Recall, however, that the design section property is used for the next analysis section property. Thus, changing the design section property also changes the next analysis section property. If an auto select section is assigned to a joist, control the section property used for that joist in the next analysis by setting the design section property to the desired beam size using the **Change Design Section** command and then rerunning the analysis.

Steel Joist Design > Verify Analysis vs Design Section Command

When the iterative design process is complete, the last used analysis section property for a joist and the current design section property for a joist should be the same. If this is not the case, the joist may not have been designed for the correct forces. The **Design menu > Steel Joist Design > Verify Analysis vs Design Section** command is useful for verifying that the last used analysis section and the current design section are the same for all steel joists in the model.

When you execute the **Verify Analysis vs Design Section** command, the program tells you how many joists have different analysis and design sections and then selects those joists, if you tell it to do so. Typically you might use this command after you have run what you believe is your last design iteration to verify that the analysis and design properties used are consistent.

It is not necessary to make a selection before using the **Verify Analysis vs Design Section** command. This command automatically checks all steel joist sections.

Steel Joist Design > Verify All Members Passed Command

The **Design menu > Steel Joist Design > Verify All Members Passed** command reports how many of the steel joists have passed the design check.

Steel Joist Design > Reset All Steel Joist Overwrites Command

The **Design menu > Steel Joist Design > Reset All Steel Joist Overwrites** command resets the steel joist overwrites for all steel joist sections back to their default values. It is not necessary to make a selection before using the **Reset All Steel Joist Overwrites** command. This command automatically applies to all steel joist sections.

Resetting your composite beam overwrites will reduce the size of your program database (*.edb) file.

Steel Joist Design > Delete Steel Joist Design Results Command

The **Design menu > Steel Joist Design > Delete Steel Joist Design Results** command deletes all of the steel joist results. It is not necessary to make a selection before using the **Delete Steel Joist Design Results** command. This command automatically applies to all steel joist sections.

Deleting your steel joist results will reduce the size of your program database (*.edb) file. Note that deleting your steel joist design results does not delete your current design section (next analysis section).

Shear Wall Design Command

Clicking the **Design menu > Shear Wall Design** command or the button activates a drop-down menu of the commands that control the Shear Wall Design postprocessor. This section describes those menu commands.

Shear Wall Design > Select Design Combo Command

Clicking the **Design menu > Shear Wall Design > Select Design Combo** command brings up the Design Load Combinations Selection form. In this form, review the default shear wall design load combinations defined by the program or designate your own design load combinations.

In the form all of the available design load combinations are listed in the List of Combos list box. The design load combinations actually used in the design are listed in the Design Combos list box. You can use the **Add** button and the **Remove** button to move load combinations into and out of the Design Combos list box. Use the **Show** button to see the definition of a design load combination. All three buttons act on the highlighted design load combination. Use the Ctrl and Shift keys to make multiple selections in this form for use with the **Add** and **Remove** buttons, if desired.

The default shear wall design load combinations have names such as DWAL1.

Shear Wall Design > View/Revise Pier Overwrites Command

Use the **Design menu > Shear Wall Design > View/Revise Pier Overwrites** command to review or change the wall pier overwrites. You may not need to assign any wall pier overwrites; however, the option is available.

The wall pier design overwrites are basic properties that apply only to the piers to which they are specifically assigned. Note that inputting 0 for most pier overwrite items means to use the program default value for that item.

Select one or more piers for which you want to specify overwrites. In the overwrites form, there is a checkbox to the left of each item. You must check this box for any item you want to change in the overwrites. If the check box for an overwrite item is not checked when you click the **OK** button to exit the overwrites form, no changes are made to the pier overwrite item. This is true whether you have one pier selected or multiple piers selected.

Shear Wall Design > View/Revise Spandrel Overwrites Command

Use the **Design menu > Shear Wall Design > View/Revise Spandrel Overwrites** command to review or change the wall spandrel overwrites. You may not need to assign any wall spandrel overwrites; however, the option is available.

The wall spandrel design overwrites are basic properties that apply only to the spandrels to which they are specifically assigned. Note that inputting 0 for most spandrel overwrite items means to use the program default value for that item.

Select one or more spandrels for which you want to specify overwrites. In the overwrites form, there is a checkbox to the left of each item. You must check this box for any item you want to change in the overwrites. If the check box for an item is not checked when you click the **OK** button to exit the overwrites form, no changes are made to the item. This is true whether you have one spandrel selected or multiple spandrels selected.

Shear Wall Design > Define General Pier Sections Command

To define a pier section with reinforcing for checking, click the **Design menu > Shear Wall Design > Define General Pier Sections** command. This command allows you to define a pier section using the Section Designer utility.

Shear Wall Design > Assign Pier Sections Type Command

Use the **Design menu > Shear Wall Design > Assign Pier Sections Type** command to assign a pier a section that has been defined using the Section Designer utility.

Shear Wall Design > Start Design/Check of Structure Command

To run a shear wall design, click **Design menu > Shear Wall Design > Start Design/Check of Structure**. This option will not be available if you have not first run a building analysis. It will also be unavailable if there are no piers or spandrels in the model.

If you have selected piers and/or spandrels when you click this command, only the selected piers and/or spandrels are designed. If no piers and/or spandrels are selected when you click this command, all piers and spandrels are designed.

Shear Wall Design > Interactive Wall Design Command

Right click on a pier or spandrel while the design results are displayed on it to enter the interactive design mode and interactively design the pier or spandrel in the Wall Design form. If you are not currently displaying design results, click the **Design menu > Shear Wall Design > Interactive Wall Design** command and then right click a pier or spandrel to enter the interactive design mode for that pier or spandrel.

Shear Wall Design > Display Design Info Command

You can review some of the results of the shear wall design directly on the program model using the **Design menu > Shear Wall Design > Display Design Info** command. The types of things you can display are reinforcing requirements, capacity ratios and boundary element requirements.

Shear Wall Design > Reset All Pier/Spandrel Overwrites

The **Design menu > Shear Wall Design > Reset All Pier/Spandrel Overwrites** command resets the pier and spandrel overwrites for all pier and spandrel elements to their default values. It is not necessary to make a selection before using the **Reset All Pier/Spandrel Overwrites** command. This command automatically applies to all pier and spandrel elements.

Shear Wall Design > Delete Wall Design Results Command

The **Design menu > Shear Wall Design > Delete Wall Design Results** command deletes all of the shear wall results. It is not necessary to make a selection before using the **Delete Wall Design Results** command. This command automatically applies to all pier and spandrel elements.

Deleting your shear wall results will reduce the size of your program database (*.edb) file.

Overwrite Frame Design Procedure Command

Tip:

*If you do not want a frame element to be designed, use the **Design menu > Overwrite Frame Design Procedure** command to assign it a Null (no design) design procedure.*

The **Design menu > Overwrite Frame Design Procedure** command allows you to change the design postprocessor for a frame element as follows:

- A *concrete frame element* can be switched between the Concrete Frame Design and the Null design procedures. Assign a concrete frame element the Null (no design) design procedure if you do not want it designed by the Concrete Frame Design postprocessor.
- A *steel frame element* can be switched between the Steel Frame Design, Composite Beam Design (if it qualifies), and the Null design procedures. In this form, a steel frame element qualifies for the Composite Beam Design procedure if it meets all of the following criteria.

  ✓ The line type is Beam; that is, the line object is horizontal.

  ✓ The frame element is oriented with its positive local 2-axis in the same direction as the positive global Z-axis (vertical upward).

  ✓ The frame element has I-section or channel section properties.

  Note that these are the same as the criteria used by the program to determine the default design procedure, except that the last two criteria checked when determining the default design procedure are not checked here.

  Assign a steel frame element the Null (no design) design procedure if you do not want it designed by either the Steel Frame Design or the Composite Beam Design postprocessor.

Tip:

Force the design procedure to be updated for a line object while you are creating your model simply by clicking on the Design menu. This automatically updates the design procedure for all objects.

When you click the **Design menu > Overwrite Frame Design Procedure** command, the Overwrite Frame Design Procedure form appears. Five options are available in this box. Depending on the frame section assignment made to the line object, several of the options may not be available. Following is a description of each of the five options:

- **Steel Frame Design:** This option is available for all line objects that have steel frame section properties. It means that the frame section is designed using the Steel Frame Design postprocessor.

- **Concrete Frame Design:** This option is available for all line objects that have concrete frame section properties. It means that the frame section is designed using the Concrete Frame Design postprocessor.

- **Composite Beam Design:** This option is available for all line objects that have steel frame section properties and that meet the three requirements listed earlier in this subsection for composite beams (horizontal, 2-axis up, I-section or channel section). It means that the frame section is designed using the Composite Beam Design postprocessor.

- **No Design:** This option is available for all line objects. It means that the line object is not designed by any design postprocessor. These line objects are given the Null design procedure.

- **Default:** This option is available for all line objects. It means that the line object is to be given the default design procedure (postprocessor) as described in the section entitled "Default Design Procedure Assignments" earlier in this chapter.

The design procedure for a line object is determined when the analysis is run. It is not necessarily determined when the line object is drawn. The design procedure is not automatically updated as you modify your model. It is automatically updated when you click on the Design menu and when you run the analysis. Thus if you draw a line object and later right click on it, the Design Procedure item in the Line Information form may be Null or outdated. You can always update the design procedure by clicking on the design menu.

If at any time while you are creating your model you want to know what the current default design procedure is for a line object, click on the Design menu then click somewhere else to close the design menu. This automatically updates the design procedures. Now you can see the design procedure assigned to the line object at the current instant in time by right clicking on the line object.

Chapter 14

14

Options Menu

General

The Options menu provides you with control over some of the basic features of this program. It also allows you to specify values for various items that, to some degree, control the look and feel of the graphic interface as well as the behavior of the program. This chapter describes all of the items on the Options menu except for the Preference items related to the design postprocessors. The design postprocessors are documented separately.

Preferences Command

The program preferences control a variety of items that affect the look and feel of the program, the default behavior of the design postprocessors, and how the program considers live load reduction.

Preferences > Dimensions and Tolerances Command

The **Options menu > Preferences > Dimensions/Tolerances** command brings up the Preferences form where you can specify preferences for various dimension and tolerance items. To modify any of the preferences, enter the number desired in the edit box next to each of these items in the Preferences form.

The following items can be specified:

Note:

The auto merge tolerance is used internally by the program to determine such things as when two point objects are in the same location or when a point object lies on a line object.

14

- **Auto Merge Tolerance:** This item is used as a basic tolerance check in the model. It is entered in length units. The program default for this item is 0.1 inches in English units and 1 mm in metric units. Following are some typical uses of the tolerance.

  - ✓ When a point object is drawn, or generated, that is within this distance of another point object, the drawn point object is merged into the original point object.

  - ✓ If a point object is within this distance of a line object, the point object is assumed to be supported by the line object. When the analysis is run, the line object is broken up at the point object and connected to the point object. Note that this might put a small kink in the line object if the point object is not exactly on the line object.

  - ✓ If a point object is within this distance of being in a plane, it is assumed to be in the plane.

- **Plan Fine Grid Spacing:** This is the spacing of invisible grid points that are used by the **Draw menu > Snap to > Fine Grid** command and the button. This item is entered in length units. The program default is 48 inches in English units or 1 meter in metric units. See the subsection entitled "Snap To Command" in Chapter 8 Draw Menu for more information.

- **Plan Nudge Value:** This is the distance that a nudged object moves after you have pressed the appropriate key on the keyboard. This item is entered in length units. The program default is 48 inches in English units or 1 meter in metric units.

See the section titled "Nudge Feature" in Chapter 5 Edit Menu for more information.

- **Screen Selection Tolerance:** When clicking on an object to select it, your mouse pointer must be within this distance of the object to select it. This item is entered in pixels. The screen selection tolerance has no affect on selection by windowing. The program default is 3 pixels.

Note:

A pixel is the smallest graphic unit (dot) that can be displayed on the screen. A typical screen resolution is 1024 pixels by 768 pixels.

- **Screen Snap To Tolerance:** When using the snap features in this program, your mouse pointer must be within this distance of a snap location to snap to it. This item is entered in pixels. The program default is 12 pixels.

- **Screen Line Thickness:** This parameter controls the thickness of lines on the screen. All lines are affected except for the bounding plane line. The thickness is entered in pixels. This item has no affect on text fonts. It also does not affect the aerial view. The program default is 1 pixel.

- **Printer Line Thickness:** This parameter controls the thickness of lines and fonts that are output to the printer. All lines are affected. The thickness is entered in pixels. The program default is 4 pixels.

Note:

In printing a point is a unit of type equal to 0.01384 inch or approximately $\frac{1}{72}$ inch.

- **Maximum Graphic Font Size:** The default text font size used in this program is determined based on the average story height of your model. As you zoom into your model, the font size becomes proportionately larger. However, the font size is never made larger than the specified maximum graphic font size. The maximum graphic font size is entered in points. The program default is 12 points.

  This font size does not apply to the grid line identification labels; the size of the labels is determined by the specified size of the bubble that displays the text.

- **Minimum Graphic Font Size:** The default text font size used in this program is determined based on the average story height of your model. As you zoom out of your model, the font

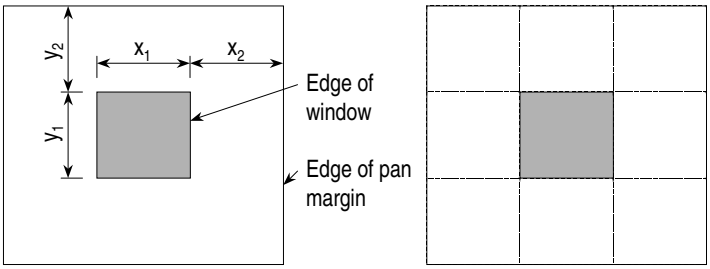


Figure 14-1: *Example of pan margin*

a) Illustration showing a 100% pan margin

b) Illustration showing that a 100% pan margin covers nine times the area of the window

size becomes proportionately smaller. However, the font size is never made smaller than the specified minimum graphic font size. The minimum graphic font size is entered in points. The program default is 3 points. In some cases, you may need to increase this minimum font size to be able to read what is on your screen.

This font size does not apply to the grid line identification labels; the size of the labels is determined by the specified size of the bubble that displays the text.

Tip:

Do not make your pan margin too large. It will eat up all of your computer's memory. The default value of 50% is normally adequate.

- **Pan Margin:** This is the distance beyond the edge of a view that you can pan. It is entered as a percent of the window size. The program default is 50%. This is a recommended value.

  Figure 14-1 shows an example of the pan margin. In the figure, the window is shown shaded. Figure 14-1a shows an example of 100% pan margin. Note that the dimension x2 is equal to 100% of x1 and similarly y2 is equal to 100% of y1. Figure 14-1b illustrates that setting the pan margin to 100% allows you to potentially cover nine times more screen area than when the pan margin is set to 0%. If the pan margin is set to 0%, you cannot pan.

  Note that setting the pan margin to 100% also requires nine times more memory than when the pan margin is set to 0%, because nine times more screen area must be saved in memory! Thus, you need to be very careful with this control or you

may use up all of your available memory and have a difficult time getting your program model to run.

See the subsection titled "Pan Feature" in Chapter 6 View Menu for additional information on panning.

- **Auto Zoom Step:** This is the size of the step used for the **View menu > Zoom In One Step** command and the **View menu > Zoom Out One Step** command as well as their associated toolbar buttons. This parameter is entered in percent. The magnification of all objects in a view is increased or decreased by this percentage. The program default is 10%.

  See the subsection entitled "Zoom Command" in Chapter 6 View Menu for additional information on the auto zoom step.

Note:

The shrink factor typically only applies to line objects, not area objects.

- **Shrink Factor:** When you shrink the objects in a view using the **Shrink Object Toggle** button , all line objects are shrunken by this specified percentage. The program default for this item is 70%. Note that you can also shrink objects by selecting the **View menu > Set Building View Options** command, or the button, and checking the Object Shrink check box in the Special Effects area of the Set Building View Options form.

  Typically, area objects are not shrunken by this percentage. Instead, they are shrunken by a set number of pixels that is built into the program. In the unusual case where shrinking the area object by the set number of pixels causes it to become an illegal object (because an edge has zero length), the area object reverts to using the percentage specified here.

The **Reset to Defaults** resets the entire dimension and tolerance values to their program default values.

Preferences > Output Decimals Command

Click the **Options menu > Preferences > Output Decimals** command to bring up the Output Decimals Preferences form

where you can specify preferences for the number of decimal places desired in the numeric output for various items. To modify any of the output decimal preferences, enter the number of decimal places desired in the edit box next to each of the items in the Preferences form.

You can control the number of decimal places for the following types of items:

- **Displacements:** These are translational displacements that are reported in the current length units (e.g., inches). The program default for displacements is four decimal places.
- **Rotations:** These are output rotations that are always reported in radians. The program default for rotations is five decimal places.
- **Forces:** These are reported in the current force units (e.g., kips). The program default for forces is two decimal places.
- **Moments:** These are reported in the current force-length units (e.g., kip-inch). The program default for moments is three decimal places.
- **Forces per length:** These are reported in the current force/length units (e.g., kip/inch). The program default for forces per length is three decimal places.
- **Moments per length:** These are reported in the current force-length/length units (e.g., kip-inch/inch). The program default for moments per length is three decimal places.
- **Stresses:** These are reported in the current force/length^2 units (e.g., kip/inch^2 also known as ksi). The program default for stresses is three decimal places.
- **Lengths:** These are reported in the current length units (e.g., inches). The program default for lengths is three decimal places.

- **Rebar Areas:** These are reported in the current length^2 units (e.g., inches^2). The program default for rebar areas is three decimal places.

- **Dimension Line Text:** Options are available to report the dimensions decimal form in the current length units (e.g., inches) or in feet and inches. If you elect to report the dimensions in feet and inches, the further option of reporting to either the nearest 1/8 inch or 1/16 inch also is available. The program default for dimension line text is feet and inches if your database units are English or in decimal form in the current units to two decimal places if your database units are Metric.

The **Reset to Defaults** button resets all of the output decimal values to their program default values. It also sets the dimension line text to decimal form, not feet and inches.

If the ETABS defaults do not work well for the units that you typically use, set your own output decimal preferences in an .edb file that you use for model initialization. Note that if you do this, the **Reset to Defaults** button resets the values to the built-in program defaults, not the values that were in your initialization file. See the section entitled "New Model Command" in Chapter 4 for more information on the initialization file.

Preferences > Steel Frame Design Command

Clicking the **Options menu > Preferences > Steel Frame Design** command brings up the Steel Frame Design Preferences form where you can set the preferences. Descriptions of the preferences and their associated defaults are provided in the separate Steel Frame Design Manual.

Preferences > Concrete Frame Design Command

Clicking the **Options menu > Preferences > Concrete Frame Design** command brings up the Concrete Frame Design Preferences form where you can set the preferences. Descriptions of

the preferences and their associated defaults are provided in the separate Concrete Frame Design Manual.

Preferences > Composite Beam Design Command

Clicking the **Options > Preferences > Composite Beam Design** command brings up the Composite beam Preferences where you can specify preferences. Descriptions of the preferences and their associated defaults are provided in the separate Composite Beam Design Manual.

Preferences > Shear Wall Design Command

Clicking the **Options > Preferences > Shear Wall Design** command brings up the Wall Pier/Spandrel Design Preferences form where you can set your preferences. Descriptions of the preferences and their associated defaults are provided in the separate Shear Wall Design Manual.

Preferences > Reinforcement Bar Sizes Command

You can define the reinforcing bar (rebar) ID, Bar Area, and Bar Diameter in the Reinforcing Bar Sizes form. To access this form, click the **Options menu > Preferences > Reinforcement Bar Sizes** command.

The program default reinforcing bars include the following:

- **ASTM standard bar sizes:** #2, #3, #4, #5, #6, #7, #8, #9, #10, #11, #14, and #18.
- **ASTM metric bar sizes:** 10M, 15M, 20M, 25M, 30M, 35M, 45M and 55M.
- **European (metric) bar sizes:** 6ϕ, 8ϕ, 10ϕ, 12ϕ, 14ϕ, 16ϕ, 20ϕ, 25ϕ, 26ϕ and 28ϕ.

You can change the bar ID, area or diameter for any of these reinforcing bars, and you can Add additional reinforcing bar definitions or Modify existing definitions. You can also Delete rein-

forcing bar definitions, including the defaults, as long as they are not being used somewhere by the program.

Reinforcing Bar Sizes Form

The following bullet items describe how to modify the bar sizes shown in the Reinforcing Bar Sizes form.

Note:

You can define your own rebar sizes if desired.

- **To add a new bar:** Type the bar ID, area and diameter in the Bar ID, Bar Area and Bar Diameter edit boxes located at the top of the Rebar area. Be sure to enter the area and the diameter in the current units. Click the **Add New Bar Size** button. If desired, make other additions, changes or deletions in the form. Click the **OK** button.

- **To change a bar ID, area or diameter:** Click on the bar ID that you want to change in the Rebar area of the form. Note that the data for the bar is highlighted and that it appears in the edit boxes at the top of the Rebar area. Modify the bar ID, area and diameter as desired in the Bar ID, Bar Area and Bar Diameter edit boxes. Be sure to enter the area and the diameter in the current units. Click the **Change Bar Size** button. If desired, make other additions, changes or deletions in the form. Click the **OK** button.

- **To delete a bar:** Click on the bar ID that you want to delete in the Rebar area of the form. Note that the data for the bar is highlighted and that it appears in the edit boxes at the top of the Rebar area. Click the **Delete Bar** button. If desired, make other additions, changes or deletions in the form. Click the **OK** button.

The **Reset to Defaults** button resets all of the area and diameter values for the program default rebar to their default values. It also adds back in any default rebar sizes that were deleted. This button has no affect on other user-defined rebar you may have added.

Preferences > Live Load Reduction Command

Clicking the **Options menu > Preferences > Live Load Reduction** command brings up the Live Load Reduction Factor form where you can specify your live load reduction preferences. Note that for live load to be reduced, it must be defined as a reducible type live load. See the section entitled "Static Load Cases Command" in Chapter 7 Define Menu for more information. The following subsections describe the four areas in this form. Do not overlook the upcoming subsection entitled "Application Area in the Live Load Reduction Factor Form" that contains crucial information about the application of live load reduction in this program.

Note:

Live load reduction does not apply to floor and ramp-type area objects.

Important Note: This program applies live load reduction to line objects (frames and links) and wall-type area objects only. It does not apply live load reduction to floor-type and ramp-type area objects; that is, the RLLF factor described in the subsequent subsection en titled "Live Load Reduction Formulas" is always 1 for floor-type and ramp-type area objects.

Method Area of the Live Load Reduction Factor Form

In the Method area of the Live Load Reduction Factor form, choose the formula or method, if any, to be used for live load reduction. The options available in the Methods area of the form are described in the following bullets.

Tip:

You can overwrite the RLLF factor on an element-by-element basis in the design overwrites.

- **No Live Load Reduction -** In this case, no live load reduction is performed, even if you have defined static load cases as reducible live load type load cases.
- **Tributary Area (UBC 97) -** The tributary area live load reduction method is based on Section 1607.5 of the 1997 UBC. The basic formula used is shown in Equation 14-1.

$$\text{RLLF} = \frac{1.0 - 0.08(A - 150)}{100} \qquad \text{Eqn. 14-1}$$

where,

RLLF = The reduced live load factor for an element, unitless. The RLLF is multiplied times the unreduced live load to get the reduced live load.

A = Tributary area for the element, ft^2. If A does not exceed 150 ft^2, then no live load reduction is used. See the subsequent subsection entitled "Tributary Area" for more information.

The RLLF factor can not be less than the minimum factor described in the section below titled "Minimum Factor Area in the Live Load Reduction Factor Form."

Note that no check is preformed to limit the RLLF based on Equation 7-2 in Section 1607.5 of the 1997 UBC.

- **Influence Area** - The influence area live load reduction method is based on Section 4.8.1 of the ASCE 7-95 Standard. The basic formula used is shown in Equation 14-2.

$$RLLF = \left(0.25 + \frac{15}{\sqrt{A_I}} \right) \qquad \text{Eqn. 14-2}$$

where,

RLLF = The reduced live load factor for an element, unitless. The RLLF is multiplied times the unreduced live load to get the reduced live load.

A_I = Influence area for the element, ft^2. The influence area for a column is taken as four times the tributary area. The influence area for a beam, brace or wall is taken as two times the tributary area. See the subsequent subsection entitled "Tributary Area" for more information.

The RLLF factor is limited to a minimum value as described in the section below titled "Minimum Factor Area in the Live Load Reduction Factor Form."

- **User-Defined Live Load Reduction** - The user-defined live load reduction method is similar to that described in Section 1607.5 of the 1997 UBC. The basic formula used is shown in Equation 14-3.

$$RLLF = \frac{100 - r(A - 150)}{100} \qquad \text{Eqn. 14-3}$$

where,

RLLF = The reduced live load factor for an element, unitless. The RLLF is multiplied times the unreduced live load to get the reduced live load.

r = Rate of live load reduction, 1/length^2. The default value is 0.08 in 1/ft^2 units.

A = Tributary area for the element or reaction, length^2. If A does not exceed A_{min} then no live load reduction is used. See the subsection below titled "Tributary Area" for more information.

A_{min} = User specified minimum tributary area for the element or reaction, length^2. The default for this item is 150 ft^2.

The RLLF factor is limited to a minimum value as described in the section below titled "Minimum Factor Area in the Live Load Reduction Factor Form."

Note:

Live loads are not reduced for basic analysis output. Reduced live loads are only used for determining design forces in the design postprocessors.

Minimum Factor Area of the Live Load Reduction Factor Form

Two minimum reduced live load factors (RLLF in the subsections above) are specified. One applies to elements receiving load from one story level only and the other applies to elements receiving load from more than one story level. Default values are provided for these minimum reduced live load factors or you can specify your own values.

The default values for the minimum reduced live load factors for the three different live load reduction methods are:

- **Tributary area method:** 0.6 for elements with load from only one story level (Single Story) and 0.4 for elements with load from more than one story level (Multi Story).
- **Influence area method:** 0.5 for elements with load from only one story level (Single Story) and 0.4 for elements with load from more than one story level (Multi Story).
- **User-defined method:** 0.6 for elements with load from only one story level (Single Story) and 0.4 for elements with load from more than one story level (Multi Story).

Application Area of the Live Load Reduction Factor Form

Important Note: In this program, the live load is currently only reduced for design forces that are used in the program design postprocessors. Live loads are *not* reduced in the basic analysis output, even if the live load is specified as a reducible-type live load when the static load case is defined and live load reduction is enabled in the preferences. Thus, when live load reduction is enabled, it is possible that you will see different live load forces for the exact same item in the basic analysis output and the design output because the live load is only reduced in the design output. Again, in the analysis output, it is unreduced.

Application to Columns Area of the Live Load Reduction Factor Form

Note:

For columns, the live load reduction can be specified to apply to the axial load only or to all force components.

For columns, the live load reduction can be specified to apply to the axial load only or to all force components. By default, the program assumes only the axial load component of columns receives the specified live load reduction. This is useful when you want to reduce the axial live load but not the moment due to live load in the column.

Note that live load reduction can be overwritten on an element-by-element basis in each of the design postprocessors.

Colors

Note:

The color of Null-type area and line objects is controlled by the Background item.

You can control the colors used for display of various items and for color-coding of output stress ratio ranges by using the **Options menu > Colors** command. You can separately specify colors to be used for screen display, color printer graphical output and non-color printer graphical output. The following two subsections describe display colors and output colors.

Colors > Display Command

Click the **Options menu > Colors > Display** command to bring up the Assign Display Colors form where you can set the display colors for various items in your model. The following bullet items describe the areas in that form.

- **Click to Change Color:** In this area you can left click on any of the color boxes to change the display color for the associated item. Following is a list of items for which you can change the display color.

Note:

*The color of the bounding plane line (cyan) is built into the program. You cannot change this. You can however use the **View menu > Set Building View Options** command, or the **Set Building View Options** button on the main (top) toolbar, to turn off the bounding plane line.*

  - ✓ **Columns:** These are column-type line objects.
  - ✓ **Beams:** These are beam-type line objects.
  - ✓ **Braces:** These are brace-type line objects.
  - ✓ **Links:** These are the color of the link element symbol. When a link property is assigned to a line object, the entire line object is not shown in this color. Only the link element symbol attached to the line object is in this color.
  - ✓ **Walls:** These are wall-type area objects. The color applies to the object edges and fill color.
  - ✓ **Floors:** These are floor-type area objects. The color applies to the object edges and fill color.
  - ✓ **Ramps:** These are wall-type area objects. The color applies to the object edges and fill color.

- ✓ **Openings:** These are area objects that are designated as openings. The color applies to the object edges and crossing lines. Note that openings are never filled, even when the Object Fill box is checked in the Set Building View Options form.

- ✓ **Springs:** This is the color of the spring symbol. This item also controls the color of restraints (supports).

- ✓ **Text:** This is the color of all of the text, including grid line IDs. It also controls the color of dimension lines and the global axes. In addition, it controls the color of dots used to show point objects when in a "Shrink Object" mode, end releases and nonlinear hinges. It also controls the color of the thickened line showing end offsets along the length of a frame member.

- ✓ **Grid Lines:** This is the default color of grid lines and grid line bubbles. It can be overwritten on a grid line by grid line basis using the **Edit menu > Edit Grid** command. See the section entitled "Edit Grid Data Command" in Chapter 5 Edit Menu for more information.

  This item also controls the color of reference planes and reference lines.

- ✓ **Background:** This is the background color. This item indirectly controls the color of null-type line objects, null-type area objects, and diaphragm extent lines because they are displayed in the opposite color from the background color.

- ✓ **Steel Dsgn:** This is the color assigned to members designated to be designed as steel frame members.

- ✓ **Comp Dsgn:** This is the color assigned to members designated to be designed as Composite Steel Beams.

- ✓ **Conc Dsgn:** This is the color assigned to members designated to be designed as Concrete Frame members.

- ✓ **Typ Member:** This is the color assigned to members on a level that has been designated as typical to another level.

- **Device Type:** Here you indicate whether the colors you are specifying are for screen display, output to a non-color printer or output to a color printer. Note that you can specify different display colors for each of these three device types.

- **Darkness:** This item controls the variation of color (intensity of shading) when extruded shapes are displayed. The darkness value can range from 0 to 1. A darkness value of 0 means there is no variation of color and the extruded shape will not really be distinguishable. A darkness value of 1 gives the maximum variation of color. The default value is 0.3. This value works well in most instances.

  Note that you can click the **View menu > Set Building View Options** command, or the **Set Building View Options** button to access the Set Building View Options form and toggle the display of extruded shapes on or off.

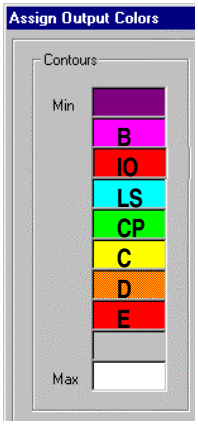
- **Reset Defaults button:** This button resets the colors to the built-in program default colors. The Reset Defaults button not only resets the colors for the currently chosen device type, it resets the colors for all three device types, regardless of which one is currently chosen.

Colors > Output Command

Click the **Options menu > Colors > Output** command to bring up the Assign Output Colors form where you can set the display colors for various output items. The following bullet items describe the areas in this form.

- **Contours:** In this area, you can specify ten different colors that are used to display shell stress contours.

  In addition, some of the colors in the Contours area are used to show the hinge state (B, IO, LS, CP, C, D or E) when display-



ing the deformed shape of a static nonlinear analysis. The color boxes used for this are shown in the sketch to the left.

- **Steel Ratios:** In this area, you can specify the colors that frame elements are displayed in when you display results from either the Steel Frame Design or the Composite Beam Design post-processor on the model. You also specify the range of values to which the colors apply. Note the following about the steel ratios:

  ✓ The five values used to define the stress ranges must always be input in increasing numerical order. The largest value defining the range does not necessarily have to be 1.0. It can be larger or smaller.

  ✓ Any frame element that does not have its Design Procedure the same as the design postprocessor that is currently displaying the results is shown in a color that is opposite of the background color.

  ✓ Any frame element with the correct Design Procedure that has not yet been designed is displayed in the "Not Yet Designed" color.

  In some cases, a frame element with the correct Design Procedure may not have a stress ratio calculated when it is run through the design postprocessor. One example where this will happen is in the Steel Frame Design postprocessor when the axial stress for the element exceeds the Euler buckling stress. In such cases, the element is displayed in the color associated with the highest range of stress values (i.e., the color of the fifth box down counting from the top).

- **Diagram Fill:** In this area you specify the colors used for filling force diagrams for frame elements. Three colors are specified here. They are colors for positive values, colors for negative values, and colors for ranges of values.

The positive and negative colors are simply based on the sign of the output force value (axial force, shear, torsion or moment). The range color is used when plotting the results of multi-valued load combinations. Consider the filled moment diagrams, for a beam element in a rigid frame, that are shown in Figure 14-2. (Note that these diagrams are plotted with positive values on the tension side of the element.)

Figure 14-2a shows the moment diagram for a load case designated 1, which is gravity load. Figure 14-2b shows the moment diagram for a load case designated 2 which is a lateral load. Figure 14-2c shows the moment diagram for a load *combination* designated 1. This load combination is created as an add-type load combination and it adds the results of load case 1 and load case 2. Note how the fill colors are different for positive and negative moment in these moment diagrams.

Figure 14-2d shows the moment diagram for a load *combination* designated 2. This load combination is created as an *envelope*-type load combination. The combination shows the envelope of load case 1 and load combination 1; that is, it shows the range of values, from maximum to minimum, of all load cases and load combinations that are included in the envelope-type load combination. As shown in Figure 14-2d, this range is displayed in a different color that you can specify.

- **Device Type:** Here you indicate whether the colors you are specifying are for screen display, output to a non-color printer or output to a color printer. Note that you can specify different display colors for each of these three device types.

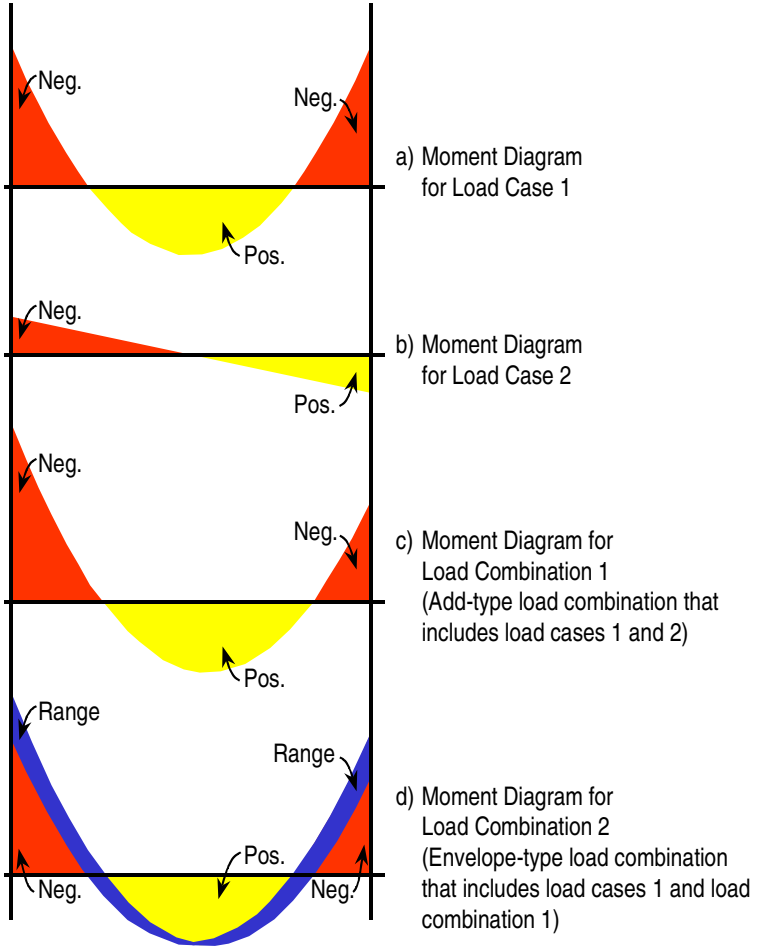


Figure 14-2: *Example of diagram fill colors where the diagrams are plotted with positive values on the tension side of the element*

- **Reset to Defaults button:** This button resets the colors to the built-in program default colors. The Reset to Defaults button not only resets the colors for the currently chosen device type, it resets the colors for all three device types, regardless of which one is currently chosen.

Other Option Menu Commands

This section describes the other option items that are available on the Options menu.

Windows Command

Tip:

You can display your model in from one to four windows. Each window can display a completely different view.

Display your model in from one to four windows. A different view can be displayed in each window. Use the **Options menu > Windows** command at any time to specify the number of windows that you want to use.

As a shortcut if you want to close a window you can click on the X in the upper right hand corner of the window. The remaining windows will automatically resize. You can not use this method to close the last window.

Show Tips at Startup Command

When you first open the program, the Startup Tips may appear. You can toggle whether these tips appear on and off using the **Options menu > Show Tips at Startup** command.

Note that the option you choose for this is saved in the ETABS.ini file in your Windows Or WinNT directory. If this file is deleted or moved, your Tips option is lost and the program defaults back to showing the tips.

Important Note: When you first start the graphical interface the Startup Tips appear. You do not have to click the **OK** button associated with the tip or click the "X" button in the upper right hand corner of the tip window to continue. Simply left clicking anywhere in the entire program window closes the Tip of the Day window. For example, as soon as you start the graphical interface, immediately click on the file menu and the Startup Tips window will close and the File menu will appear.

Show Bounding Plane Command

You can toggle the bounding plane feature on and off using the **Options menu > Show Bounding Plane** command. When this feature is active, a cyan line appears in some views showing you the location of a currently active plan or elevation view. For example, if a plan view *is currently active* and a three-dimensional view is also showing, then a cyan bounding plane appears in the

three-dimensional view around the story level associated with the plan view. As a second example, if an elevation (or developed elevation) view *is currently active* and a plan view is also showing, a cyan line appears in the plan view showing the location of the elevation. Table 14-1 lists the circumstances where the bounding plane (line) appears.

Table 14-1: Circumstances where bounding plane (line) appears

Window You Are Working In (i.e., Active Window)	Another Visible Window	Bounding Plane or Line Visible?
Plan	Plan	No
	Elevation	No
	3D	Yes
Elevation	Plan	Yes
	Elevation	No
	3D	Yes
3D	Plan	No
	Elevation	No
	3D	No

Moment Diagrams on Tension Side Command

You have the option of plotting moment diagrams for frame elements with positive values on the tension side of the member or on the compression side of the member. Click the **Options menu > Moment Diagrams on Tension Side** command to toggle this option one way or the other. See Figure 14-2 earlier in this chapter for an example of moment diagrams plotted on the tension side of a member.

Sound Command

Click the **Options menu > Sound** command to toggle the sound produced by the program when it is displaying animation of deformed shapes and mode shapes on or off.

Lock Model Command

Click the **Options menu > Lock Model** command or the **Lock/Unlock Model** button to toggle the model between locked and unlocked. When the model is locked, you cannot make any changes to it that will affect the analysis results, except as noted below.

Note:

The program automatically locks your model when you run an analysis.

Exception: While the model is locked, you can run one or more nonlinear static analyses.

When you run an analysis, the program automatically locks the model. This feature prevents changes to the model such that your analysis results would be invalidated. After you have run an analysis, if you want to make changes to your model, first unlock it. Unlocking the model deletes all of your analysis results. The results are deleted because, after you make changes to the model, the analysis results are no longer valid.

If you want to save analysis results and make changes to your model, use the **File menu > Save As** command to save the file with a new name after you have run the analysis but before you make any changes (i.e., unlock the model). You can then make changes to the file with the new name. The original file remains with its results.

Show Aerial View Window Command

Click the **Options menu > Show Aerial View Window** command to toggle the aerial view window on or off. You may find the aerial view to be a convenient tool for quickly zooming into various areas of your model. See the section entitled "The Aerial View" in Chapter 1 Graphical User Interface for more information.

Show Floating Property Window Command

Click the **Options menu > Show Floating Property Window** command to toggle the floating property window on or off. The

floating property window appears when you are drawing area and/or line objects. See Chapter 8 Draw Menu for more information.

Show Crosshairs Command

This toggle switch option controls whether crosshairs are visible when you are drawing objects in plan and elevation views. In plan view, the crosshairs are always oriented in the global X and Y directions. In elevation view, the crosshairs are always oriented in the horizontal and vertical directions.

Reset Toolbars Command

This option resets the default toolbars.