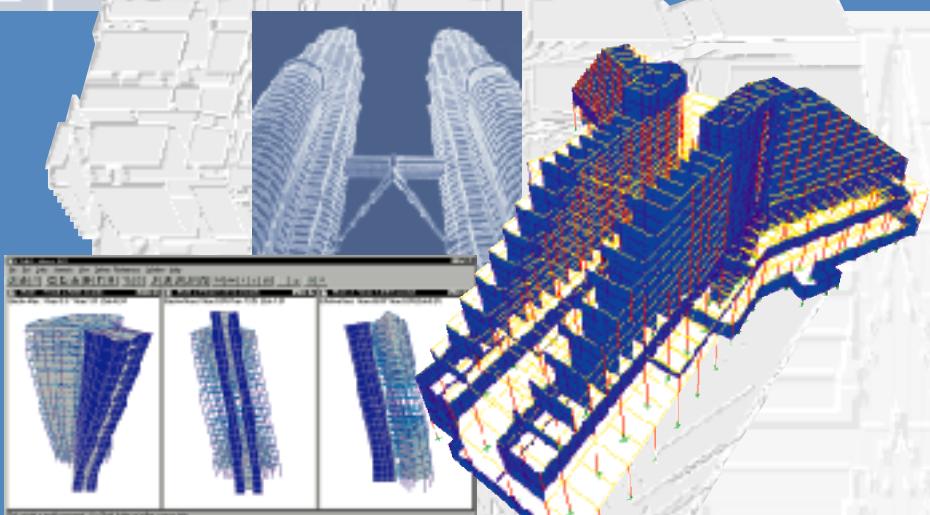


INTEGRATED DESIGN AND ANALYSIS
SOFTWARE FOR
BUILDING SYSTEMS

FOR WINDOWS® 95/98/NT/2000



ETABS®

LINEAR AND NONLINEAR
STATIC AND DYNAMIC
ANALYSIS AND DESIGN
OF BUILDING SYSTEMS

USER'S MANUAL Volumes 1 and 2

COMPUTERS &
STRUCTURES
INC.



STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

Keyboard Shortcuts for Making Selections of Objects Onscreen

<u>Keystroke</u>	<u>Purpose</u>
E	Puts you in a mode to select edges of area objects
Spacebar	Removes you from the mode where you can select area object edges
Ctrl key + left click	Pops up a dialog box where you choose which overlapping item you would like to select
Ctrl key + right click	Pops up a dialog box where you choose which overlapping item's right click information you would like to see

Keyboard Shortcuts for Various ETABS Menu Items

<u>Keystroke</u>	<u>Corresponding Menu Command</u>
Ctrl+N	File menu > New Model
Ctrl+O	File menu > Open
Ctrl+S	File menu > Save
Ctrl+P	File menu > Print Graphics
Alt+F4	File menu > Exit
Ctrl+X	Edit menu > Cut
Ctrl+C	Edit menu > Copy
Ctrl+V	Edit menu > Paste
Del	Edit menu > Delete
Ctrl+A	Select menu > Select All
F5	Analysis menu > Run
F1	Help menu > Search for Help On

Keyboard Shortcuts for Making Selections in List Boxes

First left click to select an item in the list box. Then:

<u>Keystroke</u>	<u>Purpose</u>
Ctrl key + left click	Select more items in list box that are either adjacent or non-adjacent to the first selection
Shift key + left click	Select all items in list box from the first one clicked on to the last one clicked on, inclusive (Note: You can also hold down the left mouse button and drag the mouse to select a block of items in a list box)

Keyboard Shortcuts for Constraints Used While Drawing Objects

<u>Keystroke</u>	<u>Purpose</u>
X	Constrain current line or edge to have a constant X coordinate
Y	Constrain current line or edge to have a constant Y coordinate
Z	Constrain current line or edge to have a constant Z coordinate
A	Constrain current line or edge to a specified angle
Spacebar	Remove current constraint option

Keyboard Shortcuts for Constraints Used While Reshaping Objects

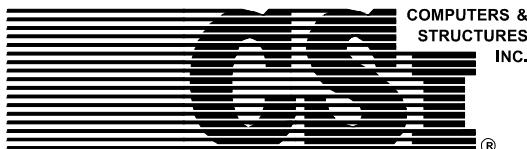
<u>Keystroke</u>	<u>Purpose</u>
X	Constrain object or edge movement to have a constant X coordinate
Y	Constrain object or edge movement to have a constant Y coordinate
Z	Constrain object or edge movement to have a constant Z coordinate
A	Constrain object or edge movement to a specified angle
Spacebar	Remove current constraint option

Main (Top) Toolbar Buttons		Side Toolbar Buttons	
	New Model		Pointer
	Open .EDB File		Reshaper
	Save Model		Draw Point Objects (displays flyout button)
	Undo		Create Points (plan, elev, 3D)
	Redo		Draw Line Objects (displays flyout buttons)
	Refresh Window		Draw Lines (plan, elev, 3D)
	Lock/Unlock Model		Create Lines in Region or at Clicks (all views)
	Run Analysis		Create Columns in Region or at Clicks (plan)
	Rubber Band Zoom		Create 2ndary Beams in Region or at Clicks (plan)
	Restore Full View		Create Braces in Region or at Clicks (elev)
	Restore Previous Zoom		Draw Area Objects (displays flyout buttons)
	Zoom In One Step		Draw Areas (plan, 3D)
	Zoom Out One Step		Draw Rectangular Areas (plan, elev)
	Pan		Create Areas at Click (plan, elev)
	3D View		Draw Walls (plan)
	Plan View		Create Walls in Region or at Clicks (plan)
	Elevation View		Select All
	Rotate 3D View		Restore Previous Selection
	Perspective Toggle		Clear Selection
	Move Up in List		Set Intersecting Line Select Mode
	Move Down in List		Snap to Points
	Object Shrink Toggle		Snap to Middle and Ends
	Set Building View Options		Snap to Intersections
	Show Undeformed Shape		Snap to Perpendicular
	Display Static Deformed Shape		Snap to Lines and Edges
	Display Mode Shape		Snap to Invisible Grid
	Display Member Force Diagram		
	Display Output Tables		

ETABS®

Three Dimensional Analysis and Design of Building Systems

ETABS USER'S MANUAL Volume 1



**Computers and Structures, Inc.
Berkeley, California, USA**

First Edition
December 1999

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., 1978-1999.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
ETABS is a registered 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.

ACKNOWLEDGMENT

Thanks are due to all of the numerous structural engineers, who over the years have given valuable feedback that has contributed toward the enhancement of this product to its current state.

Special recognition is due Dr. Edward L. Wilson, Professor Emeritus, University of California at Berkeley, who was responsible for the conception and development of the original ETABS and whose continued originality has produced many unique concepts that have been implemented in this version.



Volume 1 Contents



Tip:

If you are just getting started with ETABS Version 7 we suggest that you read Chapters 1 through 6 in Volume 1 and then use the rest of the manual (Volumes 1 and 2) as a reference guide on an as-needed basis. If you are not responsible for installing ETABS then you can skip Chapter 2.

The Table of Contents for Volume 1 of this manual consists of a chapter list that covers both Volumes 1 and 2 followed by an expanded table of contents for Volume 1 only. The chapter list devotes one line to each chapter. It shows you the chapter number (if applicable), chapter title and the pages that the chapter covers. Subheadings are provided in the chapter list section to help give you a sense of how this manual is divided into several different parts.

Following the chapter list is the expanded table of contents for Volume 1. Here all section headers and subsection headers are listed along with their associated page numbers for each chapter in Volume 1.

When searching through the manual for a particular chapter, the highlighted tabs at the edge of each page may help you locate the chapter more quickly.

If you are new to ETABS we suggest that you read Chapters 1 through 6 in Volume 1 and then use the rest of the manual (Volumes 1 and 2) as a reference guide on an as-needed basis. If you are not responsible for installing ETABS then you can skip Chapter 2.

ETABS User's Manual Chapter List

Volume 1

Volume 1 Contents

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	Chapter List.....	i to vi
N. A.	Expanded Table of Contents.....	vi to xx

 **Note:**

Chapter 4 provides a comprehensive description of the various parts of the ETABS graphic interface.

Introduction and Getting Started Information

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
1	Introduction	1-1 to 1-10
2	Installation	2-1 to 2-14
3	Getting Help	3-1 to 3-7

 **Note:**

Chapter 6 provides information on how to create an ETABS model.

General Overview of ETABS

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
4	The ETABS Graphical User Interface	4-1 to 4-16
5	Overview of an ETABS Model.....	5-1 to 5-5
6	ETABS Modeling Tips	6-1 to 6-10

The ETABS Menus

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
7	Overview of the ETABS Menus	7-1 to 7-2
8	The ETABS File Menu	8-1 to 8-30
9	The ETABS Edit Menu.....	9-1 to 9-43
10	The ETABS View Menu	10-1 to 10-30
11	The ETABS Define Menu.....	11-1 to 11-65
12	The ETABS Draw Menu.....	12-1 to 12-22
13	The ETABS Select Menu.....	13-1 to 13-6
14	The ETABS Assign Menu	14-1 to 14-64
15	The ETABS Analyze Menu	15-1 to 15-9
16	The ETABS Display Menu	16-1 to 16-40
17	The ETABS Design Menu.....	17-1 to 17-6
18	The ETABS Options Menu.....	18-1 to 18-23
19	The ETABS Help Menu.....	19-1 to 19-2

**Note:**

Chapters 8 through 19 document most of the menu commands and toolbar button shortcuts available in ETABS.

Other Information

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	References.....	References-1 to References-4
N. A.	Index	Index-1 to Index-12

Volume 2***Volume 2 Contents***

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	Chapter List.....	i to vi
N. A.	Expanded Table of Contents.....	vi to xxi

Detailed Information on Selected ETABS Topics

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
20	Units	20-1 to 20-5
21	Coordinate Systems	21-1 to 21-7
22	Story Level Data.....	22-1 to 22-5
23	Area Objects	23-1 to 23-21
24	Line Objects	24-1 to 24-31
25	Point Objects.....	25-1 to 25-12
26	Groups and Section Cuts	26-1 to 26-12
27	Load Cases, Load Combinations and Mass.....	27-1 to 27-13
28	Automatic Seismic Loads.....	28-1 to 28-37
29	Automatic Wind Loads	29-1 to 29-19
30	Automatic Meshing of Area and Line Objects ..	30-1 to 30-12
31	Manual Meshing of Area Objects	31-1 to 31-16
32	Transformation of Loads into the ETABS Analysis Model	32-1 to 32-32
33	Overview of ETABS Analysis Techniques.....	33-1 to 33-24

Note:

Volume 2 provides detailed information on a variety of ETABS subjects.



ETABS Analysis Output***Note:***

Chapters 34 through 43 document the ETABS analysis output. Design information is documented in the design manuals.

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
34	Point Object Output Conventions.....	34-1 to 34-5
35	Frame Element Output Conventions.....	35-1 to 35-5
36	Shell Element Output Conventions	36-1 to 36-14
37	Link Element Output Conventions.....	37-1 to 37-7
38	Wall Pier and Spandrel Output Conventions.....	38-1 to 38-6
39	Section Cut Output Conventions.....	39-1 to 39-4
40	Printed Input Tables.....	40-1 to 40-3
41	Printed Output Tables	41-1 to 41-14
42	Database Input/Output Tables	42-1 to 42-2
43	The ETABS Log and Out Files.....	43-1 to 43-6
44	Inserting ETABS Output into Written Reports	44-1 to 44-3

Introduction to the ETABS Design Postprocessors***Note:***

Chapters 45 through 48 provide information on how to use the ETABS design postprocessors

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
45	Steel Frame Design	45-1 to 45-17
46	Concrete Frame Design.....	46-1 to 46-15
47	Composite Beam Design	47-1 to 47-19
48	Shear Wall Design	48-1 to 48-28

Other Information

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	References.....	References-1 to References-4
N. A.	Appendix 1 - The ETABS Menu Structure.....	A1-1 to A1-13
N. A.	Index	Index-1 to Index-12

ETABS User's Manual - Volume 1 - Expanded Contents

CHAPTER 1: INTRODUCTION

Welcome to ETABS!	1-1
Organization of This Manual	1-3
ETABS: A Special Purpose Program	1-4
Significant Features of ETABS	1-5
Modeling Features	1-5
Analysis Features	1-6
Design Features	1-8
Advantages of ETABS over Other Programs	1-9

CHAPTER 2: INSTALLATION

What Your ETABS Package Includes	2-2
System Requirements	2-2
Installation of the ETABS Program	2-3
Single User Installation	2-4
Network Server Installation	2-5
Network Workstation Installation	2-6

Installing From Your ETABS CD	2-8
Installing from a Network Server	2-8
Removing ETABS from Your System	2-9
Installing the Sentinel Driver	2-9
Using the Hardware Key Device	2-11
Using the Hardware Key Device on a Single Computer	2-11
Using the Hardware Key Device on a Local Area Network	2-12
The NSRVGX Program	2-13
Troubleshooting ETABS Installation Problems	2-14
Upgrading from ETABS 6	2-14

CHAPTER 3: GETTING HELP

User Manuals Provided With ETABS	3-1
Reference Book Provided With ETABS	3-2
On-Line Help	3-2
Technical Notes on Our Web Site	3-3
Phone, Fax and E-Mail Technical Support	3-3
Phone Support	3-4
Fax Support	3-4
E-Mail Support	3-5
Help Us to Provide You Technical Support	3-5
"Hands On" Training	3-6
Seminars	3-7

CHAPTER 4: THE ETABS GRAPHICAL USER INTERFACE

Features of the Graphical User Interface	4-1
Main Window	4-1
Main Title Bar	4-2
Menu Bar	4-2

- Main (Top) Toolbar 4-3
- Side Toolbar 4-3
- Display Windows 4-4
- Display Title Bar 4-5
- Status Bar 4-5
- Mouse Pointer Position Coordinates 4-6
- Plan View Drawing and Assignment Similarity 4-6
- Current Units 4-7
- The ETABS Aerial View 4-8
- Using the Mouse 4-9
- Starting a Model 4-11
- The Two Modes of ETABS 4-12
- Locking and Unlocking a Model 4-13
- Undo Features in ETABS 4-14
- Tips for Using the Graphical User Interface 4-15

CHAPTER 5: OVERVIEW OF AN ETABS MODEL

- The Concept of Objects 5-2
- The ETABS Analysis Model 5-4

CHAPTER 6: ETABS MODELING TIPS

- Modeling Process 6-1
- Modeling Tips 6-5

CHAPTER 7: OVERVIEW OF THE ETABS MENUS

- General 7-1

CHAPTER 8: THE ETABS FILE MENU

- General 8-1

- Starting a New Model 8-1
 - Initialization of a New Model 8-2
 - Defining a Grid System 8-4
 - Defining Story Data 8-5
 - Adding Structural Objects from a Template 8-6
 - Steel Floor System Template 8-8
 - Flat Slab Template 8-10
 - Flat Slab with Perimeter Beams Template 8-12
 - Two-Way Slab Template 8-13
 - Waffle Slab Template 8-15
 - Ribbed Slab Template 8-17
 - Opening an Existing Model 8-19
 - Saving Models 8-20
 - Importing Files 8-21
 - Exporting Files 8-22
 - Creating Videos 8-26
 - Printing from ETABS 8-27
 - Printing Graphics 8-27
 - Printing Text Input and Output Tables 8-28
 - Printing Analysis Input Data 8-28
 - Printing Analysis Output Data 8-29
 - User Comments and Session Log 8-29
 - Displaying Input/Output Text Files 8-29
 - Exiting ETABS 8-30

CHAPTER 9: THE ETABS EDIT MENU

- General 9-1
- Cut, Copy and Paste 9-2
- Point Object Headings in Spreadsheet 9-3

- Line Object Headings in Spreadsheet 9-4
- Area Object Headings in Spreadsheet 9-6
- Delete 9-7
- Add to Model From Template 9-7
 - Two-Dimensional Frame 9-8
 - Three-Dimensional Frame 9-8
- Replicating Objects 9-9
 - Linear Replication 9-9
 - Radial Replication 9-9
 - Mirror Replication 9-11
 - Story Replication 9-11
 - Assignments that are Replicated 9-13
- Editing Coordinate System Grid Line Data 9-14
- Editing Story Data 9-17
 - Inserting a New Story Level 9-17
 - Deleting a Story Level 9-18
- Reference Planes and Reference Lines 9-18
- Merging Points 9-19
- Aligning Points, Lines and Edges 9-20
 - Coordinate System 9-22
 - Align Options 9-22
 - Align to X, Y or Z-Coordinate 9-23
 - Align to X or Y Grid Lines 9-23
 - Trim or Extend Selected Lines 9-25
 - Align Selected Points 9-27
 - Align Tolerance 9-28
- Moving Point, Line and Area Objects 9-29
 - Moving Objects in the Z Direction 9-30
- Expanding and Shrinking Areas 9-30

- Merging Areas 9-32
- Joining Lines 9-33
- Dividing Lines 9-35
- Reshaper Tool 9-37
 - Reshaping Area Objects 9-38
 - Reshaping Line Objects 9-39
 - Reshaping Dimension Lines 9-40
 - Reshaping Point Objects 9-40
 - Moving/Reshaping Objects in the Z Direction 9-41
- The ETABS Nudge Feature 9-42

CHAPTER 10: THE ETABS VIEW MENU

- General 10-1
- Types of Views 10-1
 - Three Dimensional (3D) Views 10-2
 - Plan Views 10-5
 - Elevation Views 10-6
 - Perspective Views 10-8
 - Perspective Toggle in a Plan View 10-8
 - Perspective Toggle in an Elevation View 10-8
 - Perspective Toggle in a Three-Dimensional View 10-9
 - Custom Views 10-9
- Viewing Tools Available in ETABS 10-10
 - View Limits 10-10
 - Show Selection Only and Show All 10-11
 - Zoom Features 10-11
 - Pan Feature 10-13
 - Refresh Views and Windows 10-14
 - Change Axes Location 10-15

Make Measurements in your ETABS Model 10-15

Building View Options 10-16

View by Colors 10-17

Special Effects 10-18

Object Visibility 10-19

Object View Options 10-22

Piers and Spandrels 10-23

Other Visibility Options 10-24

Special Frame Items 10-26

Other Special Items 10-29

CHAPTER 11: THE ETABS DEFINE MENU

General 11-1

Material Properties 11-1

Frame Section Properties 11-6

Importing Sections from a Database 11-7

Adding User-Defined Frame Section Properties 11-9

Adding Frame Section Properties using Section Designer 11-11

Nonprismatic Sections 11-13

Segment Lengths 11-15

Starting and Ending Sections 11-15

Variation of Properties 11-16

Effect upon End Offsets Along the Length of Frame Elements 11-17

Reinforcing for Concrete Frame Section Properties 11-17

Reinforcing Information for Beams 11-17

Reinforcing Information for Columns 11-19

Wall/Slab/Deck Section Properties 11-21

Defining Wall and Slab Sections 11-21

Defining Deck Sections 11-23

Link Properties	11-26
Frame Nonlinear Hinge Properties	11-27
Section Cuts	11-27
Defining Section Cuts	11-28
Response Spectrum Functions	11-29
Response Spectrum Functions from a File	11-30
User-Defined Response Spectrum Functions	11-32
Code Specific Response Spectrum Functions	11-33
1994 UBC Parameters for a Response Spectrum Function	11-34
1997 UBC Parameters for a Response Spectrum Function	11-34
1996 BOCA Parameters for a Response Spectrum Function	11-35
1995 NBCC Parameters for a Response Spectrum Function	11-35
IBC2000 Parameters for a Response Spectrum Function	11-36
1997 NEHRP Parameters for a Response Spectrum Function	11-36
1998 Eurocode 8 Parameters for a Response Spectrum Function	11-37
1992 NZS 4203 Parameters for a Response Spectrum Function	11-37
Modifying and Deleting Response Spectrum Functions	11-38
Time History Functions	11-38
Time History Functions from a File	11-38
User-Defined Time History Functions	11-41
ETABS Template Time History Functions	11-42
Sine Time History Function Template Parameters	11-43
Cosine Time History Function Template Parameters	11-44
Ramp Time History Function Template Parameters	11-44
Sawtooth Time History Function Template Parameters	11-45
Triangular Time History Function Template Parameters	11-46
Static Load Cases	11-46

Response Spectrum Cases	11-50
Spectrum Case Name	11-50
Structural and Function Damping	11-50
Modal Combination	11-52
Directional Combination	11-53
Input Response Spectra	11-55
Excitation Angle	11-55
Time History Cases	11-56
History Case Name	11-56
Options	11-56
Load Assignments	11-60
Static Nonlinear/Pushover Cases	11-63
Load Combinations	11-63
Mass Source	11-64

CHAPTER 12: THE ETABS DRAW MENU

General	12-1
The ETABS Similar Stories Feature	12-2
Drawing Point Objects	12-3
Drawing Line Objects	12-3
Floating Properties of Object Window for Line Objects	12-8
Drawing Area Objects	12-9
Floating Properties of Object Window for Area Objects	12-12
Developed Elevations	12-12
Dimension Lines	12-17
Special Drawing Controls	12-18
ETABS Snap Options	12-18
Drawing Constraints in ETABS	12-21

CHAPTER 13: THE ETABS SELECT MENU

- General 13-1
- Basic Methods of Selecting Objects 13-1
- Other Methods of Selecting Objects 13-4
- Deselecting Objects 13-6

CHAPTER 14: THE ETABS ASSIGN MENU

- General 14-1
 - Assignments to Point Objects 14-1
 - Rigid Diaphragm Assignments to Point Objects 14-2
 - Panel Zone Assignments to Point Objects 14-3
 - Properties 14-4
 - Connectivity 14-6
 - Local 2-Axis 14-8
 - Options 14-9
 - Restraint (Support) Assignments to Point Objects 14-9
 - Point Spring Assignments to Point Objects 14-10
 - Coupled Springs 14-12
 - Link Property Assignments to Point Objects 14-13
 - Additional Point Mass Assignments to Point Objects 14-14
 - Force Loads to Point Objects 14-16
 - Ground Displacement Assignments to Point Objects 14-18
 - Temperature Loads Assignments to Point Objects 14-20
 - Assignments to Line Objects 14-22
 - Frame Section Assignments to Line Objects 14-22
 - Frame Releases and Partial Fixity Assignments to Line Objects 14-23
 - Unstable End Releases 14-24
 - Frame Rigid Offset Assignments to Line Objects 14-24
 - Rigid End Offsets Along the Length of Frame Elements 14-25

Automatically Calculated End Offset Lengths	14-25
End Offset Properties and the Rigid-Zone Factor	14-26
Rigid Frame Joint Offsets	14-27
Frame Output Station Assignments to Line Objects	14-28
Local Axes Assignments to Line Objects	14-29
Frame Property Modifier Assignments to Line Objects	14-31
Link Property Assignments to Line Objects	14-32
Frame Nonlinear Hinge Assignments to Line Objects	14-32
Pier Label Assignments to Line Objects	14-34
Spandrel Label Assignments to Line Objects	14-35
Line Spring Assignments to Line Objects	14-36
Additional Line Mass Assignments to Line Objects	14-38
Automatic Frame Mesh/No Mesh Assignments to Line Objects	14-39
Point Load Assignments to Line Objects	14-40
Distributed Load Assignments to Line Objects	14-42
Temperature Load Assignments to Line Objects	14-46
Assignments to Area Objects	14-48
Wall, Slab and Deck Section Assignments to Area Objects	14-48
Opening Assignments to Area Objects	14-49
Rigid Diaphragm Assignments to Area Objects	14-49
Local Axes Assignments to Area Objects	14-50
Shell Stiffness Modifiers Assignments to Area Objects	14-51
Pier Label Assignments to Area Objects	14-52
Spandrel Label Assignments to Area Objects	14-53
Area Spring Assignments to Area Objects	14-54
Additional Area Mass Assignments to Area Objects	14-56
Automatic Membrane Floor Mesh/No Mesh Assignments to Area Objects	14-57
Uniform Surface Load Assignments to Area Objects	14-58

- Temperature Load Assignments to Area Objects 14-60
- Group Name Assignments 14-63
- Clear Display of Assigns 14-64

CHAPTER 15: THE ETABS ANALYZE MENU

- Analysis Options 15-1
 - Building Active Degrees of Freedom 15-1
 - Dynamic Analysis Parameters 15-3
 - P-Delta Analysis Parameters 15-5
- Run Analysis 15-8
 - Analyze Window 15-9
- Run Static Nonlinear Analysis 15-9

CHAPTER 16: THE ETABS DISPLAY MENU

- General 16-1
- Undeformed Shape 16-1
- Loads 16-2
 - Joint/Point Loads 16-2
 - Frame/Line Loads 16-3
 - Shell/Area Loads 16-5
- Input Table Mode 16-6
- Deformed Shape 16-7
- Mode Shape 16-12
- Member Force and Stress Diagrams 16-14
- Support and Spring Reactions 16-14
- Frame Element, Pier and Spandrel Forces 16-17
- Shell Forces and Stresses 16-20
- Load 16-21
- Component Type 16-21

- Component 16-21
- Contour Range 16-24
- Stress Averaging 16-24
- Miscellaneous Notes about Shell Element Forces and Stresses 16-26
- Link Element Forces 16-26
- Energy Diagram 16-27
- Response Spectrum Curves 16-29
 - Define Tab 16-29
 - Axes Tab 16-30
 - Options Tab 16-31
 - Frequency/Period Tab 16-32
 - Damping Tab 16-33
- Time History Traces 16-34
- Static Pushover Curve 16-39
- Section Cut Forces 16-39
- Output Table Mode 16-40

CHAPTER 17: THE ETABS DESIGN MENU

- Overview 17-1
- Overwrite Frame Design Procedure 17-2
 - Background 17-2
 - ETABS Default Design Procedure Assignments 17-3
 - The Overwrite Frame Design Procedure Command 17-4

CHAPTER 18: THE ETABS OPTIONS MENU

- General 18-1
- Preferences 18-1
 - Dimensions and Tolerances 18-2
 - Output Decimals 18-6

- Reinforcement Bar Sizes 18-7
 - Overview 18-7
 - Reinforcing Bar Sizes Dialog Box 18-9
- Live Load Reduction 18-10
 - General 18-10
 - Method Area in the Live Load Reduction Factor Dialog Box 18-10
 - No Live Load Reduction 18-10
 - Tributary Area Live Load Reduction 18-11
 - Influence Area Live Load Reduction 18-11
 - User-Defined Live Load Reduction 18-12
 - Minimum Factor Area in the Live Load Reduction Factor Dialog Box 18-13
 - Application Area in the Live Load Reduction Factor Dialog Box 18-13
 - Application to Columns Area in the Live Load Reduction Factor Dialog Box 18-14
 - Tributary Area 18-14
- Colors 18-14
 - Display Colors 18-15
 - Output Colors 18-17
- Other Option Items 18-20
 - Windows 18-20
 - Startup Tips 18-20
 - Bounding Plane 18-21
 - Moment Diagrams on Tension Side 18-22
 - Sound 18-22
 - Lock Model 18-22
 - Aerial View Window 18-23
 - Floating Property Window 18-23
 - Crosshairs 18-23

CHAPTER 19: THE ETABS HELP MENU

The ETABS Help File 19-1

About ETABS 19-1

REFERENCES

INDEX



Chapter 1

Introduction

ETABS is a special purpose computer program developed specifically for building systems. The concept of special purpose programs for building type structures was introduced over 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.

Welcome to ETABS!

ETABS version 7 is by far the most sophisticated and user-friendly release of the ETABS series of programs. This is the first version of ETABS that is completely integrated within Microsoft Windows. It features a powerful graphical user interface that is unmatched in terms of ease-of-use and productivity.

Creating and modifying a model, executing the analysis, design, and optimizing the design are all done through this single interface. 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:

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

The ETABS program is comprised of the following modules all 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.
- Finite element based linear static and dynamic analysis module.
- Finite element based nonlinear static and dynamic analysis module (available in ETABS Nonlinear version only).
- 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.

ETABS Version 7 is available in the following two versions:

Note:

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



- **ETABS Plus** - Includes all of the capabilities of ETABS except that nonlinear static and dynamic analysis is not included. The steel frame design, concrete frame design, composite beam design and shear wall design modules are all included.
- **ETABS Nonlinear** - Includes all of the capabilities of ETABS including nonlinear static and dynamic analysis. The steel frame design, concrete frame design, composite beam design and shear wall design modules are all included.

Organization of This Manual

Tip:

If you are just getting started with ETABS Version 7 we suggest that you begin by reading Chapters 1 through 6 in Volume 1 of this manual. If you are not responsible for installing ETABS then you can skip Chapter 2. We further suggest that you use the rest of the manual (Volumes 1 and 2) as a reference guide on an as-needed basis.



We have tailored the content of all of the ETABS manuals more toward a design engineer than a computer analyst. This manual is divided into six parts in two separate volumes that are described below:

Volume 1

- **Chapters 1 through 3:** General introduction and information on installation and getting help. This is the "Getting Started" portion of the manual.
- **Chapters 4 through 6:** A general overview of ETABS. Chapter 6 provides useful information about how to create models in ETABS.
- **Chapters 7 through 19:** Detailed discussion of each of the ETABS menus.

Volume 2

- **Chapters 20 through 33:** Additional detailed information on selected ETABS topics.
- **Chapters 34 through 44:** Documentation of the analysis output for ETABS.

- **Chapters 45 through 48:** Introduction to the ETABS design postprocessors. This includes the Steel Frame Design, Concrete Steel Frame Design, Composite Beam design and Shear Wall Design postprocessors.

We suggest that you start by reading Chapters 1 through 6 in Volume 1 of this manual. If you are not responsible for installing ETABS then you can skip Chapter 2. We further suggest that you use the rest of the manual (Volumes 1 and 2) as a reference guide on an as-needed basis. Refer to Chapter 3, "Getting Help", for information on additional ETABS documentation.

ETABS: A Special Purpose Program

A wide variety of general-purpose computer software is currently available for the static and dynamic structural analysis of complex frame structures. Most of these programs can be used for the analysis of multistory frame and shear wall buildings. However, from an analytical point of view, building systems represent a unique class of structures that deserve special treatment.

ETABS is a special purpose computer program for the linear and nonlinear, static and dynamic analysis of buildings. Special purpose computer programs for addressing such problems, such as ETABS, have proven to be very practical and efficient, resulting in significant savings in the time associated with data preparation, output interpretation and execution throughput over general purpose computer programs for the following reasons:

- The input and output conventions of the user interfaces correspond to common building terminology. The models are defined logically floor-by-floor, column-by-column, bay-by-bay and wall-by-wall and not as a stream of non-descript nodes and elements as in general purpose computer programs. Thus the structural definition is simple, concise and meaningful.
- The results produced by the programs are in a form directly usable by the engineer. General-purpose computer programs produce results in a general form that may need additional processing before they are usable in structural design.

Significant Features of ETABS

Modeling Features

The ETABS building is idealized as an assemblage of area, line and point objects. These objects are used to represent column, beam, brace, wall, floor and link/spring objects. The basic frame geometry is defined with reference to a simple three-dimensional grid system. With relatively simple modeling techniques very complex framing situations may be considered.



Tip:

Simple yet sophisticated drawing tools are available in ETABS to help you create your model.

The buildings may be unsymmetrical and non-rectangular in plan. Torsional behavior of the floors and interstory compatibility of the floors are accurately reflected in the results. The solution enforces complete three-dimensional displacement compatibility, making it possible to capture tubular effects associated with the behavior of tall structures having relatively closely spaced columns.

Semi-rigid floor diaphragms may be modeled to capture the effects of in-plane floor deformations. Floor elements may span between adjacent levels to create sloped floors (ramps). This is useful for modeling parking garage structures.

Modeling of partial diaphragms, such as in mezzanines, setbacks, atriums and floor openings is possible. It is also possible to model situations with multiple independent diaphragms at each level thereby allowing the modeling of buildings consisting of several towers rising from a combined structure below or vice-versa.

The column, beam and brace elements may be non-prismatic, and they may have partial fixity at their end connections. They may also have uniform, partial uniform or trapezoidal load patterns, and they may have temperature loads. The effects of the finite dimensions of the beams and columns on the stiffness of a frame system are included using end offsets that can be automatically calculated. It is possible to define shear, moment, axial and PMM nonlinear hinges at any location on a column, beam or brace element for use in a static nonlinear pushover analysis.

The floors and walls can be modeled as membrane elements with in-plane stiffness only, plate bending elements with out-of-plane stiffness only or full shell-type elements which combine both in-plane and out-of-plane stiffness. Floor and wall elements may have uniform load patterns in or out-of-plane, and they may have temperature loads. The required meshing of the membrane floor elements for gravity load transfer is handled automatically by the program. The user merely needs to define the outline of a floor, and define the outline of any openings, and the program will automatically create the required mesh for the floor elements. The column, beam, brace, floor and wall elements are all compatible with one another.

Special formulations of one-point nonlinear spring elements and two-point nonlinear link elements are included to allow the modeling of biaxial hysteretic and friction pendulum base isolation devices. Uniaxial gap, damper and plasticity options are also available. These elements may be used for the modeling of added stiffness and damping elements, slotted-bolted energy dissipaters, supplemental dampers and other passive energy devices and also for evaluating the effects of three dimensional structural pounding. A linear or nonlinear link element is also available for modeling panel zones in frame structures. The link elements can be used in static linear analysis, static nonlinear analysis (push-over), dynamic (time history) linear analysis and dynamic nonlinear analysis.



Note:

The nonlinear elements are only active in the nonlinear version of ETABS.

Analysis Features

Static analyses for user specified vertical and lateral floor or story loads are possible. If floor elements with plate bending capability are modeled, then vertical uniform loads on the floor are transferred to the beams and columns through bending of the floor elements. Otherwise, vertical uniform loads on the floor are automatically converted to span loads on adjoining beams, or point loads on adjacent columns, thereby automating the tedious task of transferring floor tributary loads to the floor beams without explicit modeling of the secondary framing. Lateral wind and seismic load patterns meeting the requirements of various building codes can be automatically generated by the program.

Three-dimensional mode shapes and frequencies, modal participation factors, direction factors and participating mass percentages are evaluated using either eigenvector or ritz-vector analysis.

**Tip:**

The theoretical basis for many of the numerical analysis techniques used in ETABS is discussed in Professor Wilson's book titled "Three Dimensional Static and Dynamic Analysis of Structures."

The P-delta effects are included in the basic formulation of the structural lateral stiffness matrix as a geometric correction. This causes equilibrium to be satisfied in the deformed position and the P-delta problem is solved exactly with minimal numerical effort. Also, as the correction is on the lateral stiffness matrix, the P-delta effects appear in the static analysis and filter into the eigen, response spectrum and time history analyses.

Response spectrum analysis is based upon the mode superposition method using either the complete quadratic modal combination (CQC) technique [E. L. Wilson, et al., 1981b and A. K. Gupta, 1990], the square root of the sum of the squares (SRSS) technique, the absolute sum (ABS) technique, or the general modal combination (GMC) technique. The structure may be excited from three different directions in any one run with independent spectra. The direction combination can be by either the SRSS or the ABS technique. Composite modal damping effects from supplemental dampers are included in the analysis.

The linear time history analysis uses a variable time step closed form integration technique for the evaluation of the modal coordinates [E. L. Wilson, et al., 1981a]. Time-dependent ground accelerations or load cases can excite the structure concurrently in any three orthogonal directions with independent excitations. The nonlinear time history analysis is based upon a very efficient iterative vector superposition integration scheme [E. L. Wilson, 1993 and E. L. Wilson, et al., 1989]. The time history results may be displayed as time-functions (such as displacement vs. time) or as function-function (such as force vs. deformation). Response spectrum curves may be created from acceleration time histories generated by ETABS.

Thermal stress analyses for user specified distributions of temperature are possible.

Three-dimensional static nonlinear (pushover) analysis for uniform load patterns, load patterns based on mode shapes, and any arbitrarily defined load pattern is possible. Nonlinear hinge property definition data is set up such that both user-defined hinge properties and the hinge properties designated in the ATC-40 and FEMA-273 documents can be easily assigned. Capacity spectrum analysis is automatically performed and graphical as well as printed output is provided.



Tip:

The **File menu > Print Tables** command is a very powerful way to get output printed to a printer or to a file.

The static nonlinear analysis capabilities of ETABS allow you to perform incremental construction analysis where the structure is analyzed accounting for forces that arise as a result of the sequence of construction.

Results from the various static load conditions may be combined with each other or with the results from the dynamic response spectrum or time history analyses.

The output can be viewed graphically, displayed in a tabular form on the screen, printed to a printer, or printed to an ASCII file. Types of output available include mode shapes and participation factors, static and dynamic story displacements and story shears, inter-story drifts, and joint displacements, reactions and member forces, time history traces, and more. For static nonlinear analysis the types of output available include force-deformation (pushover) and capacity spectrum curves, step-by-step deformation, step-by-step member forces, and step-by-step hinge state.

Design Features



Tip:

The building code used for design is specified or viewed using the **Options menu > Preferences** command.

The ETABS program includes a fully integrated set of design post processors for steel and concrete design. Design post processors included in the package are for steel frame design, concrete frame design, composite beam design and shear wall design.

Many different building codes are included in the design modules. Not all of these design codes are included for each design module. To see the specific codes available for a given design module click the **Options menu > Preferences** command. For steel design the codes considered include:

- AISC ASD89 (American)
- AISC LRFD93 (American)
- BS 5950-90 (British)
- CISC 95 (Canadian)
- Eurocode 3-1992 (European)

For concrete design the building codes included are:

- ACI 318-99 (American)
- 1997 UBC (American)
- BS 8110-89 (British)
- CAN3-A23.3-M94 (Canadian)
- Eurocode 2-1991 (European)
- NZS 3101-95 (New Zealand)

ETABS can automatically create the load combinations, with the appropriate load factors, required for each of these codes. The user can modify these load combinations or add to them.

Advantages of ETABS over Other Programs

Using ETABS can result in a significant decrease in required man-hours to complete the model, a significant decrease in processing time and possibly a significant increase in solution accuracy because ETABS takes advantage of the characteristics that are inherent in the basic nature of a building type structure that a general structural analysis program may not recognize. These characteristics of buildings include:

- Most buildings are of simple geometry with horizontal beams and vertical columns. A simple grid system defined by horizontal floor lines and vertical column lines can establish such geometry with minimal effort.

- Many of the floor levels in buildings are typical. Most general programs do not recognize this fact; therefore, for typical regions of the structure many of the internal calculations may be unnecessarily duplicated.
- In most buildings the dimensions of the members are large in relation to the bay widths and story heights. These dimensions have a significant effect on the stiffness of the frame. Corrections for these effects must be included in the formulation of the member stiffness. Most general-purpose programs work on centerline dimensions and such stiffness corrections are usually very tedious to implement.
- In the analysis of buildings the member forces need to be produced at the outer faces of the supports of the members. Such transformations are not automatic in general-purpose programs.
- The loading in building systems is of a restricted form. Loads, in general, are either vertically down (dead or live) or lateral (wind or seismic). The vertical loads are usually applied on the floors and beams and the lateral loads are generated at the story levels. Tributary floor loads need to be automatically transferred to the building frames. Also, various code-loading requirements need special options that allow convenient generation and combination of the vertical and lateral static and dynamic loading.
- It is desirable to have a building analysis computer output printed in a special format, such as, in terms of a particular story, column, beam, brace or wall. Also, special output, such as lateral story displacements and inter-story drifts are required.

All of the above mentioned characteristics of building systems are recognized by ETABS, making it ideally suited for the specific application of building systems.



Chapter 2

Installation

If you are just getting started with ETABS Version 7 we suggest that you first read Chapter 1 of this manual. Then you can follow the instructions in this chapter to install the ETABS program and, if necessary, the Sentinel Driver.

This chapter covers the following topics:

- What Your ETABS Package Includes
- System Requirements
- Installation of the ETABS Program
- Removing ETABS from Your System
- Installing the Sentinel Driver
- Using the Hardware Key Device
- Troubleshooting ETABS Installation Problems
- Upgrading from ETABS 6

What Your ETABS Package Includes

Your ETABS package includes the following:



Tip:

Check your ETABS package carefully as soon as you open it to verify that you received all of these items.

- A single Compact Disk (CD) containing the Setup program, executable files, support files, and sample data files for the version you ordered (ETABS Plus, or ETABS Nonlinear).
- Three Dimensional Static and Dynamic Analysis of Structures, by Edward L. Wilson.
- The ETABS users manuals.
- A Hardware key device.

System Requirements

ETABS will work on any Windows-based personal computer with at least the following configuration:



Tip:

Although ETABS runs using an 800 by 600 resolution monitor, for optimum performance a monitor supporting 1024 by 768 resolution or better is recommended.

- Intel Pentium, or higher, processor.
- A minimum of 64 MB of RAM.
- At least 200 MB of free hard disk space. Program files require about 20 MB. The remainder is needed for analytical scratch (temporary) files. Large projects may require much more disk space.
- Microsoft Windows 95/98/NT-4.0/2000 or higher operating system.
- Windows-compatible graphics card and monitor supporting at least 800 by 600 resolution and 256 colors. Note that 1024 by 768 resolution is recommended as a practical minimum.

Installation of the ETABS Program

If you already have ETABS installed on your machine, please uninstall it first before installing the new version. To do this, follow the directions in the next topic entitled “Removing ETABS from Your System.”



Tip:

A Network Server installation does not actually install the ETABS program. It only copies the files to the network for use in Network Workstation installations.

Three types of installation are available:

- **Single User installation** installs the entire ETABS program on your local computer. Use this type of installation if you are not connected to a network or you want your installation to be independent of a network.
- **Network Server installation** copies the entire ETABS program to a network server. This would typically be performed by a network administrator to make ETABS available for subsequent installation and execution by network workstations.
- **Network Workstation installation** installs ETABS on a network workstation using a minimum amount of local disk space. This requires that ETABS already be installed on a network server that is available to the workstation whenever the program is used.

If you are not sure what to do, choose Single-User installation.

The type of program installation you choose is *independent* of how you access the hardware key device. For example, a single-user installation can access the key device over the network. Alternatively, a network-workstation installation can access the key on the local workstation. See the topic “Using the Hardware Key Device” later in this chapter for more information.

Single User Installation

To install the entire ETABS program on your local computer:

- Turn on your computer and start Windows.
- **IMPORTANT!** No other applications should be running during the installation procedure. **Close all other applications before proceeding!**
- Follow the instructions below under either the subtopic “Installing From Your ETABS CD”, or the subtopic “Installing From a Network Server.”
- You will be asked to choose the destination folder or directory in which to store the program and support files on your local machine.
- When asked to select the type of Setup, choose “Single User.”
- Respond to the remaining prompts from SETUP to complete the installation.
- If you are going to attach the hardware key device to this computer then follow the instructions provided in the section titled “Installing the Sentinel Driver.”



Tip:

If you are unsure of which type of installation to choose, then select the Single User installation option.

The SETUP program will:

- Copy system files to your Windows folder.
- Copy program and support files to the folder or directory that you specify on your local machine.
- Optionally copy sample data files to a subfolder called EXAMPLES.
- Optionally copy program manual files to a subfolder called MANUALS.
- Register ETABS for use in Windows.
- Add ETABS to the Start menu for Windows.

If you have trouble with your ETABS installation please refer to the section titled “Troubleshooting ETABS Installation Problems.”



The Readme.txt File

IMPORTANT! After any installation, please read the Readme.txt file in the ETABS directory where you installed the program. This file contains important information that may be more current than the program manuals. You may use any editor or word-processor to review this file. If you download an intermediate update of ETABS from our web site then please read the Readme.txt file that came with the update.

Network Server Installation

To copy the entire ETABS program to a network server for subsequent installation and execution by network workstations:

- Turn on your computer and start Windows.
- You must perform the installation from a Windows machine, but you can install it onto any Windows, Novell, or other type of file server that can be accessed from Windows workstations.
- You must have sufficient rights to create files on the server.
- **IMPORTANT!** No other applications should be running during the installation procedure. **Close all other applications before proceeding!**
- Follow the instructions below under the subtopic “Installing From Your ETABS CD”, or the subtopic “Installing From a Network Server.”
- You will be asked to choose the destination folder or directory in which to store the setup, system, program, and support files on the network server.
- When asked to select the type of Setup, choose “Network Server.”



Tip:

A Network Server installation does not actually install the ETABS program. It only copies the files to the network for use in Network Workstation installations.

- Respond to the remaining prompts from SETUP to complete the installation.

The SETUP program will:

- Copy setup, system, program, and support files to the folder or directory that you specify on the network server.
- Copy sample data files to a subfolder called EXAMPLES.
- Copy program manual files to a subfolder called MANUALS.

IMPORTANT! You will not be able to run ETABS after this installation. You must still perform a single-user or network-workstation setup from the network server in order to use the program.

If you have trouble with your ETABS installation please refer to the section titled “Troubleshooting ETABS Installation Problems.”

Network Workstation Installation

To install ETABS on your network workstation to run from a network server:

- Turn on your workstation (computer) and start Windows.
- ***IMPORTANT!*** No other applications should be running during the installation procedure. **Close all other applications before proceeding!**
- Follow the instructions below under subtopic “Installing From a Network Server.” **You should not perform this installation from a CD.**

**Tip:**

Always perform a network workstation installation from your network server, not from the ETABS CD.

You must perform a network server installation before you perform a network workstation installation.

- You will be asked to choose the destination folder or directory in which to store small support files on your local workstation.
- When asked to select the type of Setup, choose “Network Workstation.”
- Respond to the remaining prompts from SETUP to complete the installation.
- If you are going to attach the hardware key device to this workstation then follow the instructions provided in the section titled “Installing the Sentinel Driver.”

The SETUP program will:

- Copy system files to the Windows folder on your workstation.
- Copy support files to the folder or directory that you specify on your local machine.
- Optionally copy sample data files to a subfolder called EXAMPLES.
- Optionally copy program manual files to a subfolder called MANUALS.
- Register ETABS for use with Windows.
- Add ETABS to the Start menu for Windows.

Whenever you run ETABS your workstation must have access to the network server from which you installed ETABS.

If you have trouble with your ETABS installation please refer to the section titled “Troubleshooting ETABS Installation Problems.”

Installing From Your ETABS CD

To install ETABS from the CD:

- Insert the ETABS CD into your CD-ROM drive.
- Wait for the ETABS setup program to start automatically. If the setup program does not automatically start then:
 - ✓ Select **Run** from the Windows Start menu.
 - ✓ Type “D:\SETUP.EXE” without the quotes in the command line of the Run dialog box. If your CD is in a drive other than D:, then substitute the appropriate drive letter for D.
 - ✓ Click **OK** in the Run dialog box to start the setup program.
- Follow the remaining instructions in the appropriate subtopic above (Single User Installation, Network Server Installation) for the type of installation you are performing.



Note:

*Do not perform
a network
workstation
installation
from the CD.*

Installing from a Network Server

To install ETABS from a network server:

- Ask your network administrator for the location of an existing ETABS network-server installation.
- Select **Run** from the Start menu.
- On the Command Line in the Run dialog box, type in the complete path to the ETABS Setup.exe program as given to you by your network administrator.
- Click **OK** in the Run dialog box to start the installation.

- Follow the remaining instructions in the subtopic above (Single User Installation, Network Server Installation, Network Workstation Installation) for the type of installation you are performing.

Removing ETABS from Your System

If you need to remove ETABS from your system, or before installing a new version of ETABS:

- Turn on your computer and start Windows.
- ***IMPORTANT!*** No other applications should be running during this procedure. **Close all other applications before proceeding!**
- From the Windows **Start** menu choose **Settings** and then choose **Control Panel** from the submenu to display the Control panel window. Double click **Add Remove Programs**, highlight ETABS in the scroll box and then click the Add/Remove button.
- Follow the prompts. When asked, you may safely remove all shared components that reside in the ETABS folder or directory.

Installing the Sentinel Driver

In order to use the hardware key device on a Windows machine, either in local mode or as a key server, you must install the Sentinel Driver for Windows. This driver is NOT automatically installed by the ETABS setup program and must be separately installed. **The driver is not required for machines accessing the key across the network; it is only required for machines with the key device attached.**

**Note:**

The Sentinel driver installation program automatically uninstalls any previous installation of a Sentinel driver. Thus it is not necessary for you to uninstall any previously installed Sentinel driver prior to implementing this installation process.

The installation of the Sentinel driver is a one-time process that needs to be performed on each machine that may have a hardware key device attached to its parallel port. After that, the Sentinel driver will automatically run every time you start your computer.

The Sentinel drivers for the hardware key device are located on the ETABS CD in a directory called NETDRIVE.

To install the Sentinel driver on any machine:

- Insert the ETABS CD in the CD drive on the machine where you want to install the Sentinel driver.
- Select **Run** from the Windows Start menu.
- Type “D:\NETDRIVE\SETUP.EXE” without the quotes in the command line of the Run dialog box. If your CD is in a drive other than D:, then substitute the appropriate drive letter for D.
- Click **OK** in the Run dialog box to start the Sentinel driver setup program.

The installation takes only a few seconds. **It proceeds quietly, requiring no input from you, and displaying no messages unless an error occurs.** After the installation is done (the hourglass disappears, this typically takes five seconds or less), you should restart your system.

If an error occurs during installation of the Sentinel driver check the following:

- If you are installing the Sentinel driver on a Windows NT machine make sure that you have administrative rights to the NT machine.
- Restart your computer, make sure all applications are closed, and try installing the Sentinel driver again.

If the Sentinel driver installation error persists then contact CSI technical support. See the section titled “Phone, Fax and E-Mail Technical Support” in Chapter 3 for information on contacting CSI.

Using the Hardware Key Device

The ETABS program is copy-protected with a hardware key device that is provided with the software. This hardware key device must always be accessible to ETABS whenever you use the program. This is done by attaching the key device to the parallel port of your local workstation or to that of another workstation on your local area network, as described below. The same key device may be used in either local or network access modes.

If ETABS cannot find the hardware key device while you are using the program, ETABS will enter display-only mode, with the following implications:

- You can save your current model.
- You cannot make changes to your model.
- You cannot perform analysis or design.

If the hardware key device inadvertently becomes unavailable while you are using ETABS, you should save your model, exit the program, and then re-attach the key device before restarting ETABS.

Using the Hardware Key Device on a Single Computer

Note:

The NSRVGX program does not need to be run if you are using the hardware key device on a single computer only. However, the Sentinel driver should be installed on that computer.

If you are normally going to use ETABS on a single computer, it is simplest to attach the hardware key device directly to that computer.

Attach the key device to any parallel printer port on your workstation. The key device should be directly attached to the computer port. Any printers, data switches, or other devices that use the port may then be attached to the other end of the key device. The hardware key device does not require a printer to be connected or, if connected, for it to be powered.

You may connect an extension cable between the computer port and the hardware key device, and/or between the key device and any printer or other devices. Use a straight-through DB-25 male to DB-25 female cable.

Hardware key devices for different programs can usually be attached to the same parallel port. Contact CSI technical support, if you are using multiple key devices and are experiencing conflicts. See the section titled “Phone, Fax and E-Mail Technical Support” in Chapter 3 for information on contacting CSI.

Using the Hardware Key Device on a Local Area Network

If you are going to use ETABS on multiple workstations, it may be more convenient to attach the hardware key device to one workstation and access it from other workstations across a local area network.

The hardware key device works with the IPX/SPX and/or the NETBEUI/NETBIOS network protocol. You need to have one of these protocols installed for the hardware device to work reliably over a network. The hardware device does not work with the TCP/IP protocol; the device does work if TCP/IP is installed concurrently with another network protocol. For Windows 95 it is very important that these protocols be set up exactly the same on all machines on the network. Many people have had success setting IPX/SPX as the default protocol for Windows 95 computers, and then enabling NetBIOS over IPX/SPX.

The workstation to which the hardware key device is attached is called the key server. The key device is attached to the key server as described above for a local workstation. The key server must be running Windows, and running the key-server program NSRVGX as described below. Also, the Sentinel driver must be installed on the key server as described in the section titled “Installing the Sentinel Driver.”

The standard, single-user key will allow different workstations to access ETABS at different times. Multiple-user keys are available that will allow simultaneous use of ETABS by more than one workstation. Several key devices can exist on the same network by using multiple key servers. Each key server may connect to one or more key devices on one or more ports. Concurrent usage of ETABS is allowed from different workstations up to the sum of the license limits of all key devices on all key servers.

The NSRVGX Program

Each key server must be running NSRVGX in order for the key device to be accessible across the network. Without NSRVGX, the key device is available locally only to the key-server workstation itself.

The key device should be attached to the parallel port before starting NSRVGX. Use the Start menu or Windows Explorer to start NSRVGX.EXE, which is located in the ETABS folder. After a few seconds of initialization, NSRVGX will run minimized as an icon. You may open the NSRVGX window to see how many other workstations are currently accessing the hardware key devices attached to the key server. You may minimize the window, but do not stop NSRVGX or shut down the key-server workstation while other workstations are accessing the attached key device.

Note that it may take a few moments for ETABS to access a hardware key device across a network, particularly if the network is busy or if the key server is performing other tasks. Finally, please read the important note in the box at the bottom of this page regarding NSRVGX.



Important Note Regarding NSRVGX

The present version of NSRVGX is unable to run if it is located under a folder (directory) whose name contains a space character. By default, ETABS is installed in an ETABS subfolder under the folder “Program Files”. Since this folder name contains a space character, NSRVGX will not run.

To remedy this situation, copy NSRVGX.EXE to another folder so that there are no space characters anywhere on the path, and run it from that folder. This problem should be fixed in a subsequent release of ETABS.

Troubleshooting ETABS Installation Problems

If you have trouble with your ETABS installation please check the following:



Tip:

If all else fails contact CSI technical support as described in Chapter 3.

- Verify that you have installed the latest service pack available for your particular Windows operating system. The service packs are available on Microsoft's web site.
- Restart your computer, make sure all applications are closed, and try installing the ETABS again.
- If you are performing a network server installation verify with your network administrator that you have sufficient rights to copy files to the network server.

If the ETABS installation problem persists then contact CSI technical support. See the section titled "Phone, Fax and E-Mail Technical Support" in Chapter 3 for information on contacting CSI.

Upgrading from ETABS 6

Most modeling and analysis features available in ETABS 6 are also present in ETABS 7, and many new features have been added.

ETABS 6 input data files can be imported directly into the ETABS version 7 graphical user interface. Note that only the Version 6 analysis input file can be imported. The Version 6 Steeler, Conker and Waller input files *cannot* be imported. The imported analysis models can be displayed, modified, analyzed and designed. Input data files for ETABS versions earlier than ETABS 6 need to be first converted to ETABS 6 files before they can be imported into ETABS 7.



Tip:

If you convert ETABS 6 files to ETABS 7, then check the conversion carefully.

WARNING! Some imported data may be interpreted differently by ETABS 7 than by ETABS 6. Be sure to check your imported model carefully! Compare the results of analyses using both ETABS 6 and ETABS 7 before making further use of the imported ETABS 6 model!



Chapter 3

Getting Help

There are multiple options available for obtaining help for the ETABS program. These options include the User's Manuals and Reference book provided with ETABS, on-line help that is included as a menu option in the program; technical notes that are provided on our web site; phone, fax and e-mail technical support; "hands on" training which is provided at our office in Berkeley, California; and occasional seminars held throughout the country and the world. Each of these options is described in detail in this chapter.

User Manuals Provided With ETABS

The ETABS users manuals are provided with your ETABS program. The main intent of these manuals is to provide you with detailed reference material regarding the use of ETABS that you can refer to as you use the program on an as-needed basis. The users manuals are not, in general, intended to be read from cover to cover in a single sitting.

Reference Book Provided With ETABS

3



Note:

Add Professor Wilson's book to your reference library.

The reference book titled "Three Dimensional Static and Dynamic Analysis of Structures", by Edward L. Wilson is provided with your ETABS package. The major purpose of Professor Wilson's book is to present the theoretical background required so that users of structural analysis computer programs such as ETABS can understand the basic assumptions and approximations used in the program and thus can adequately verify the results of the analyses.

This book provides a detailed overview of the theoretical basis for the analysis capabilities that are included in ETABS. It is currently used as a textbook in various universities.

On-Line Help



Shortcut:

From within the graphical user interface press the F1 function key on the keyboard at any time to activate the ETABS on-line help.

The ETABS program includes extensive on-line help. This help can be accessed any time the graphical user interface is open by either 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 dialog box is open you will bring up context-sensitive help related to that dialog box.

The data included in the on-line help is mainly focused on assisting you with the nuts and bolts of using the graphic interface. It is mainly intended to help you enter data into dialog boxes and to inform you what the data you enter into dialog boxes means. 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 does not in general address other items such as tips and tricks for creating models, the theoretical basis for the analysis engine in ETABS, the formulas and algorithms used in the design modules, in-depth descriptions of each element in ETABS, etc.

Technical Notes on Our Web Site

We expect to provide a series of technical notes for ETABS on our web site at www.csiberkeley.com. These notes are in *.pdf format and they can be downloaded and then viewed and/or printed using the Adobe Acrobat Reader. These notes are intended to be a supplement to the User's Manuals and are provided for the following purposes:

- Provide errata for the User's Manuals if needed.
- Provide documentation for new features that are added to the program.
- Provide additional information for selected topics in the User's Manuals on an as-needed basis. An example of this might be an in-depth explanation and example of the different geometric nonlinearity effects available for a static nonlinear analysis (pushover).
- Provide information on how some of the algorithms internal to the program work on an as-needed basis. An example of this might be a technical note describing exactly how ETABS calculates the unbraced length of a column.
- Provide additional theoretical information expanding on some topic discussed in Professor Wilson's book.



Tip:

Check our web site on a regular basis to see if new or updated ETABS technical notes have been added.

Check our web site on a regular basis to see if new or updated ETABS technical notes have been added.

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 is 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 ETABS help file. See Chapter 19.

If you can not find a solution then contact us as described below.

Phone 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 holidays.

When you send a fax with questions about your model please include a picture of your model if possible. This will often times be 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 general more efficient for you to email your entire model (*.edb and/or *.Set 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 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 *.Set 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 below. 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

**Shortcut:**

For a shortcut to 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 following information, as appropriate to your circumstances, we will be able to serve you better and faster.

- The name and the version number of the program you are using. See the section titled “About ETABS” in Chapter 19 for more information on this.
- If you are faxing us a description of your model then 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).

“Hands On” Training

CSI holds “hands on” training sessions for ETABS at our training facility in our office in Berkeley, California on a regular basis. The sessions focus on in-depth, hands-on training and covers modeling of simple as well as complicated structures using ETABS. These one-day sessions are geared toward training you in the use of the graphical user interface.

**Note:**

Many engineers are extremely pleased with the one-on-one attention they receive in these training sessions.

Two full-time instructors are provided for these classes. To guarantee individual attention the class size is limited to six (6) attendees, each provided with an individual computer. Class members are encouraged to bring their own projects to the training sessions, but familiarity with the program is not a pre-requisite.

In a typical training session the first quarter of the day the instructors provide an overview of the ETABS graphic interface and do some live demonstrations creating models, analyzing them and reviewing the results. For the remainder of the day the

student works on the computer either with training problem models developed by CSI or with their own projects which they have brought in. The two instructors are available at all times to answer questions and to provide helpful tips and comments.

If you are interested in attending a “hands on” training session at CSI, or if you would like more information please call us at (510) 845-2177 or e-mail us at info@csiberkeley.com.

Seminars

CSI occasionally holds technical seminars throughout the country and sometimes the world. Often these seminars are jointly sponsored by CSI and a technical engineering association. Topics of recent seminars have included Dynamic Analysis, Nonlinear Dynamic Analysis, and Nonlinear Static Pushover Analysis. Generally these seminars are one-day events. Often they are scheduled to occur in conjunction with a technical convention. A schedule for these seminars is posted on our web site at www.csiberkeley.com.



Chapter 4

The ETABS Graphical User Interface

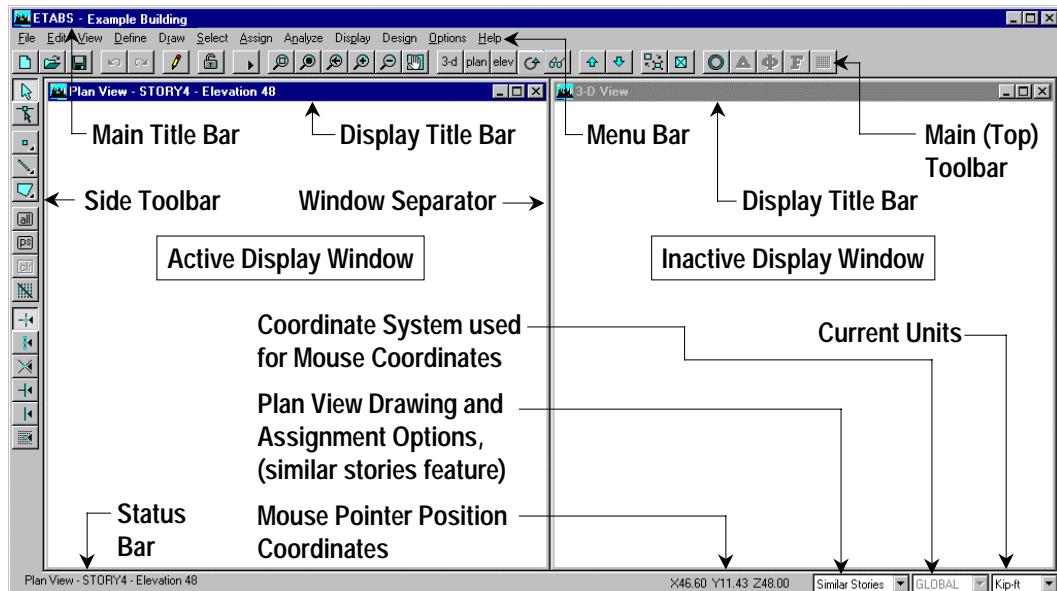
This chapter provides a brief overview of the ETABS graphical user interface.

Features of the Graphical User Interface

The ETABS graphical user interface is shown in Figure 4-1. In the figure several important features of the interface are labeled. These features include the main window, main title bar, display title bar, menu bar, main toolbar, side toolbar, display windows, status bar, mouse pointer position coordinates and the current units. Each of these items is discussed below.

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. You can refer to your Windows help, available on the Start menu, for additional information on these items.



Main Title Bar

(Above)

Figure 4-1:
The ETABS graphical user interface

The main title bar, located at the top of the main window, includes the program name and the model name. When ETABS is the active program the main title bar is highlighted. 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 menus for ETABS. The menus contain all of the operations that you can perform with ETABS. To access a menu simply left click on it. This will cause the menu to drop down giving you access to various commands on the menu.

You will notice that some of the commands on the menus have three dots after them like this, ..., and others have a filled triangular section adjacent to them on the right hand margin of the menu, like this, ▶. The three dots after a menu item indicates that a dialog box will appear when you click on the menu item. The triangle indicates that a submenu will appear when you click on



Tip:

The menu items can be accessed using shortcut keystrokes.

the menu item. The commands with neither of these items after them execute as soon as you click them. There are no submenus or dialog boxes for these commands.

Main (Top) Toolbar

The main (or top) toolbar provides quick access to some commonly used commands, particularly file, viewing and analysis output display options. You execute a main toolbar operation by left clicking on a main toolbar button. If you hold your mouse pointer over a main toolbar button for a few seconds without clicking or holding down any mouse buttons then a short description of the toolbar button function will pop up in a small text box. All of the operations available on the main toolbar can also be accessed from the menu bar. The main toolbar is not user customizable.

Side Toolbar

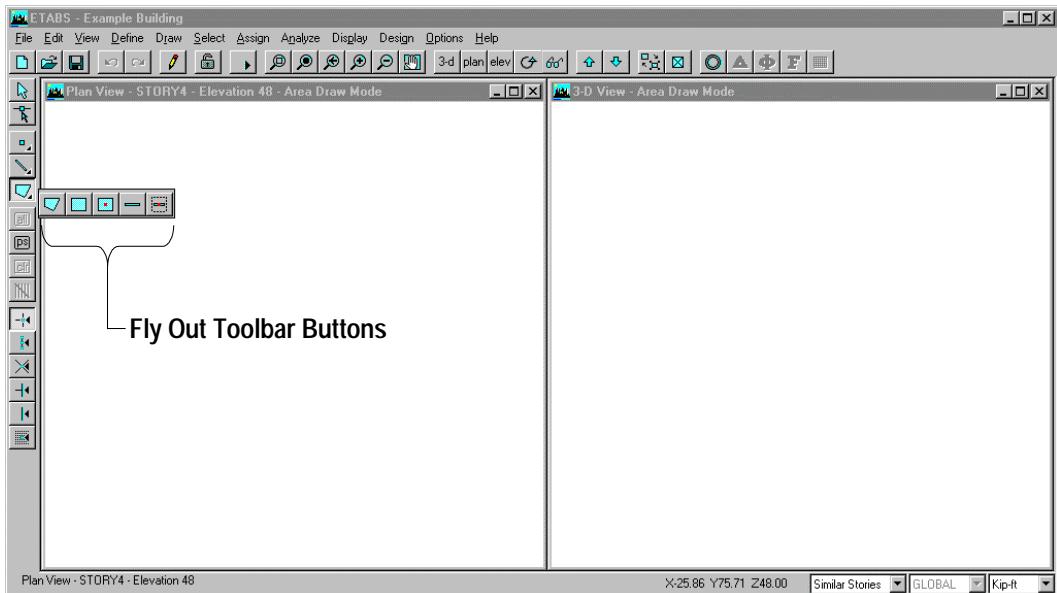
The side toolbar provides quick access to some commonly used drawing options. You execute a side toolbar operation by left clicking on a side toolbar button. If you hold your mouse pointer over a side toolbar button for a few seconds without clicking or holding down any mouse buttons then a short description of the toolbar button function will pop up in a small text box. All of the operations available on the side toolbar can also be accessed from the menu bar. The side toolbar is not user customizable.



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.

When you click the drawing buttons on the side toolbar that have a small triangle in the bottom right-hand corner of the button, like this, ▾, you will see additional toolbar buttons fly out from the original toolbar button. In these cases the additional fly out buttons are the ones that actually perform an ETABS operation. The function of the original button is to display the fly out buttons. You simply left click on one of the fly out buttons to perform an operation. Figure 4-2 shows an example of some fly out buttons.



Display Windows

(Above)

Figure 4-2:
Example of ETABS
fly out toolbar but-
tons

Display windows show the geometry of the model and may also include display of properties, loading and analysis or design results. You may have from one to four display windows present at any 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, you could have four different views of the undeformed shape or other type of display. These four different views might be a plan view, two elevations and a three-dimensional perspective view.

Only one display window is active at a time. Viewing and displaying actions only affect the active window. You may make any display window active by clicking on its display title bar or clicking anywhere within the window. You can always tell which display window is active because its title bar will be highlighted.

Shortcut:

You can close a display window by clicking the "X" in the upper right hand corner of the display window.



After performing certain operations the display window may need to be redrawn. Normally this is done automatically, but on some occasions you may have to manually refresh the display window. This sometimes occurs after you delete some elements. You can use the **View menu > Refresh Window** command, or the **Refresh Window** button, , on the main toolbar to redraw the window.

You can close a display window by left clicking the “X” button at the top of the window to the right of the display title bar. You must always have at least one display window open.

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 then the title bar typically also tells you what results are currently displayed.

Status Bar

The status bar is located at the bottom of the main ETABS 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 the screen the text may tell you what you can do. For example, when the deformed shape is displayed, this 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 similarity (only available when you are in plan view), and a drop-down box for setting the current units.



Tip:

Keep an eye on the status bar for useful information and messages.

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 case.

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 you can use the **Edit menu > Edit Grid Data** command to define alternate coordinate systems.

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 Similarity

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 what happens.

There are three options in this drop-down box. They are:

- **One Story:** An object drawn is only applied to the story level that you drew it on. An assignment is only made to the actually selected elements.

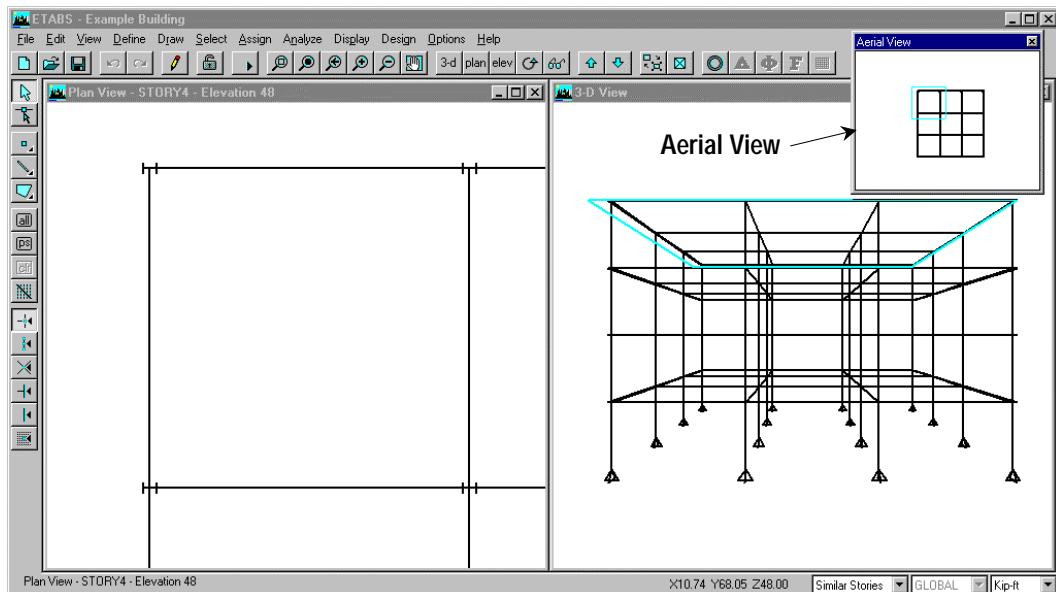
- **All Stories:** An object drawn in the plan view is applied to all story levels in the model at the same plan location. An assignment is made to the actually selected elements and to all other elements in the same plan location at all other story levels.
- **Similar Stories:** An object drawn in the plan view is applied to all similar story levels in the model at the same plan location. An assignment is made to the actually 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 dialog box which can be accessed by clicking **Edit menu > Edit Story Data > Edit** command.

The three options only apply at the time the object is drawn or the assignment is made. These options do not apply retroactively. For example if you draw an element at one story level, and then after you have drawn the element change the Plan View Drawing and Assignment option to "All Stories", the program does not retroactively change something you have previously drawn or assigned.

Current Units

The current units are displayed in a drop-down box located on the far right hand side of the status bar. You can change the current units on the fly at any time by simply clicking the drop-down box and selecting a new set of units. You can also change the current units using drop-down boxes that are located inside some of the ETABS dialog boxes. See the section titled "Units" in Chapter 20 for more information.



The ETABS Aerial View

(Above)

Figure 4-3:
*Example of the
ETABS aerial view*



Tip:

If the aerial view window is not needed simply click the "X" in the upper right hand corner of the aerial view window to close it. You can re-display it from the Options menu.

The ETABS aerial view is a small window that can float over the main ETABS 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 4-3. You can switch the aerial view on or off using the **Options menu > Show Aerial View** command as a toggle switch.

The aerial view displays a full view of the active display window's drawing in a separate window so that you can quickly zoom into any area of your model without having to restore the full view first. The aerial view can also be used to help you determine which part of the model you are zoomed into when you are working with large models. Each time the model is edited the aerial view is updated.

For example, if the active window shows a plan view of story level 4, and it is zoomed in to a small area of that story level, the aerial view will show the full view of story level 4. The portion of story level 4 that is shown in the active display window will be outlined by a bounding box in the aerial view.

You can use the aerial view to zoom into any area in the active display window. Simply draw a bounding box in the aerial view window to specify the area that you want to zoom in on. To draw the bounding box in the aerial view window put the mouse pointer at one corner of the box that you want to draw, click the left mouse button and hold it down while you drag the mouse to the diagonally opposite corner of the box that you want to draw, and release the mouse button. As you are dragging the mouse you will see the current outline of the bounding box.

**Tip:**

You can zoom quickly around a large, complex model using the aerial view.

You will notice that in some instances after you draw the bounding box its shape will change. This occurs because the aspect ratio of the bounding box is automatically adjusted to match the aspect ratio of the active display window. When changing the aspect ratio of the bounding box that you drew to match that of the active window ETABS always maintains the longer dimension of the box you drew. It only changes the shorter dimension.

If you click the right mouse button inside the bounding box in the aerial view window and hold the button down, then you can drag the bounding box to a new location in the aerial view window. The display in the active window will be updated accordingly once you release the right mouse button.

Left clicking once in the aerial view window restores the full view.

If you pan the view in the active display window then the bounding box shown in the aerial view will move also. You can pan the view in the active display menu by choosing pan from the View menu, or clicking the Pan button on the main toolbar and then left clicking in the display window and holding down the mouse button while you drag the mouse.

Using the Mouse

There are seven separate mouse actions that you can use in ETABS. They are left click, right click, hold down the Ctrl key on the keyboard and left click, hold down the Ctrl key on the keyboard and right click, hold down the Shift key on the keyboard and left click, double click and drag.

Left click means to press down the left button on your mouse and release it. In general you left click to select menu items, activate toolbar buttons, and select objects in your model. In the ETABS documentation if we say simply to click on something we always mean to left click on it.

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

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

Another use for holding down the Ctrl key on your keyboard while you left click occurs in dialog boxes where you are trying to select multiple items from a list box at the same time. Holding down the Ctrl key and left clicking on an item adds that item to the 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 make sense. One example of this can be seen after you have run an analysis by clicking **Display menu > Set Output Table Mode** and then clicking the **Select Loads** button.

If you are in a dialog box and want to select multiple adjacent items from a list box you can left click on the first item and then hold down the Shift key on the keyboard and left click on the last item. 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 make sense.



Tip:

When we tell you to click on something we mean to position the mouse pointer over that something and click the left mouse button.

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

Finally you can drag the mouse. You drag the mouse by clicking the left mouse button, holding the button down, sliding the mouse to another location and then releasing the mouse button. One example where you drag the mouse is to draw a bounding box in an aerial view window to zoom in on your model. Another time you drag the mouse is when you use the **Draw menu > Reshape Object** command.

Starting a Model

You start the ETABS graphical user interface by selecting the ETABS program from the Windows Start menu or by clicking an ETABS shortcut on your desktop. Once the graphical user interface is started you can start a new model by selecting **File menu > New Model**. Refer to Chapter 6 for more information on the ETABS modeling process. Refer to the section titled "Starting a New Model" in Chapter 8 for more information on starting a new model.

Once your model is started you may want to make use of the online help that is included in ETABS. You can access this help at any time by clicking **Help menu > Search for Help On** command. Alternatively, you can press the F1 key on your keyboard at any time to access the online help. If you are in a dialog box when you press the F1 key then you will automatically jump to context sensitive help for that dialog box. This is a very powerful way to get relevant help for ETABS.

Note:

 Refer to Chapter 6 for more information on creating a model.

Once you have started your model you should save it often. You can use the **File menu > Save** command or the **Save** button, , on the main toolbar to do this. Saving your file often is the best protection against unforeseen problems such as a power failure or a computer crash. There is no AutoSave feature in ETABS so it is up to you to save your file on a regular basis. You may also want to occasionally copy backup copies of your input file to another location for safekeeping. When you have spent a significant amount of time creating your model it is al-

ways better to err on the safe side when creating backup copies. The files you might want to copy elsewhere 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.

The Two Modes of ETABS

There are two distinct modes in ETABS. They are called **draw** mode and **select** mode. The draw mode allows you to draw objects. The select mode allows you to select objects and is used for such things as editing operations, making assignments to objects and viewing or printing results.

By default you are in select mode. You automatically enter draw mode when you select one of the submenu options of the following commands from the Draw menu or click on the corresponding button on the side toolbar:

- Draw Point Objects
- Draw Line Objects
- Draw Area Objects
- Draw Special Items

Note:

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

You remain in draw mode until you do one of the following to return you to select mode:

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

You can always tell which mode you are in by looking at your mouse pointer. In select mode the mouse pointer is the Normal Select pointer as defined by your mouse pointer properties. If you are using the default settings for your mouse properties then in select mode the mouse pointer will probably look like this, .

In draw mode the mouse pointer is the Alternate Select pointer as defined by your mouse pointer properties. If you are using the default settings for your mouse properties then in draw mode the mouse pointer will probably look like this, .

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

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

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  button on the main toolbar to quickly lock or unlock your model.

ETABS has a feature that allows you to lock or unlock your model. The toggle switch for this is available on the Options menu and also on the main toolbar. When a model is locked no changes can be made to it. You may at times want to lock your model to prevent further changes from being made to it. You can unlock your model at any time to make changes to it.

After an analysis is performed, ETABS automatically locks the model to prevent any changes that would invalidate the analysis results and subsequent design results that may be obtained. Typi-

cally if you want to make changes to your model after you have run an analysis then you must first unlock the model. When you do this you are warned that the analysis results will be deleted. If you do not want the analysis results to be deleted then you should save your model under a different name before unlocking it. Any subsequent changes will then be made to the new model.

Undo Features in ETABS

Shortcut:

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

ETABS has a Undo feature for changes to model drawing (geometry changes) that works 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 didn't want them after all you can use the **Edit menu > Undo** command to get rid of them. If you then decide you really did want them there is an **Edit menu > Redo** command that will bring them back. 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 then fifteenth and so on operation. You could not however decide that you just wanted to undo the seventh operation.

The Undo and Redo features do not work for changes made in dialog boxes. When an item is changed in a dialog box, the change is not actually implemented until the **OK** button is clicked. If the **Cancel** button is clicked the change is not made and all values in the dialog box automatically go back to their original values. If you are working in a sub-dialog box, that is, a dialog box that is called from another dialog box, the changes are not actually implemented until the **OK** button is clicked in the topmost dialog box, that is, until the last dialog box is closed by clicking the **OK** button.

Shortcut:

Use the  Redo button on the main toolbar to redo a previously undone operation.

Suppose for example that you are in a series of sub-dialog boxes that go five levels deep. In order to have the changes made at the fifth level be accepted and implemented you must click the **OK** button at the fifth, fourth, third, second and topmost level. Clicking the **Cancel** button at any one of these levels would cancel any of the changes made at that level and at any lower levels. Thus if you click the **Cancel** button in the topmost level of dialog boxes no changes will be made at any level.

Tips for Using the Graphical User Interface

Following are tips for using the ETABS graphical user interface:

- When you first start the ETABS 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 ETABS window closes the Tip of the Day window. For example, as soon as you start the ETABS graphical interface you can immediately click on the file menu and the Startup Tips window closes and the File menu appears.
- You can work in the ETABS graphical user interface most efficiently if you have at least a 17" monitor with at least 1024x768 resolution.
- In ETABS you can work in from one to four windows. Usually one window or two windows tiled vertically works best. You can set the number of windows using the **Options menu > Windows** command.
- The graphical user interface display colors can be customized to suit your individual preferences using the **Options menu > Colors** command.
- The maximum and minimum graphic font sizes used in the display windows in the ETABS graphical user interface can be modified 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. What you need to think is, "What do I want to do?" When creating or modifying a model you typically either edit the model, change the view of the model, define properties or load cases, draw something new in the model, assign something to the model such as properties or loads. Thus you can come up with a one word answer to your what to do question which is either edit, view,

define, draw, or assign. Once you have this one word answer you know which menu to go to.

- Right clicking on an object brings up a dialog box with information about location, geometry and assignments for that object. This information is for viewing only. You can not edit the information in this dialog box.
- Save your model often.
- ETABS has an Undo feature that works for multiple steps back to the last time you saved your model.
- Do not overlook the extensive online help for ETABS that is available at the press of the F1 key on your keyboard. If you are in a dialog box when you press the F1 key you will bring up context sensitive help related to that dialog box.
- Keep an eye on the text displayed on the left-hand side of the status bar at the bottom of the ETABS main window. Useful information about your model is displayed here.
- Try a few practice problems to get the hang of the ETABS graphical user interface.
- If necessary, consider attending a CSI “hands on” training session for ETABS at our offices in Berkeley, California. See the section titled “Hands on Training” in Chapter 3 for more information.
- If you have used an earlier version of ETABS you may want to import a familiar file to see how it looks in ETABS version 7.



Chapter 5

Overview of an ETABS Model

An ETABS model is different from models produced in many other structural analysis programs for two main reasons:

Note:

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

- ETABS is optimized for modeling building systems. Modeling procedures and design capabilities are all tailored toward buildings.
- An ETABS 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 these same objects to define loads.

When you run an analysis ETABS automatically converts your 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 ETABS 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 ETABS Analysis Model” later in this chapter for additional information on the analysis model.

5

The Concept of Objects

Tip:

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

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

- Draw a series of area, line and point objects that represent your building. You can use the drawing tools located on the Draw menu to do this. See Chapter 12 for documentation of the Draw menu commands.

Note that you could also create these objects directly from one of the built-in ETABS templates. See the subsection titled “Adding Structural Objects from a Template” in Chapter 8, and the section titled “Add to Model



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 ETABS.*** It is not a difficult concept, but because it is new, it may take some time for it to fully sink in and thus for you to become comfortable with it. Once 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.

To become comfortable and familiar with objects it may be helpful for you to read Chapters 23, 24 and 25 of this manual. These chapters discuss area, line and point objects, respectively, in detail. In addition, it may be helpful to work through the ETABS tutorial.

If after all of the above you have any trouble with the concept of objects, and have specific questions, please contact CSI Technical Support as described in Chapter 3. You may also want to consider attending one of the “hands on” training classes for ETABS that CSI offers at its office in Berkeley, California, also described in Chapter 3.

From Template” in Chapter 9 for more information.

- Assign structural properties and loads to these objects. You can use the assign options located on the Assign menu to do this. See Chapter 14 for documentation of the 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 drawing command. This feature is documented in Chapter 12.

- Manually mesh the area objects if they are not horizontal membrane slab or deck sections that you are letting ETABS fully automatically mesh into the analysis model. See Chapter 30 for discussion of automatic meshing and Chapter 31 for discussion of manual meshing.

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

It is extremely important that you become familiar with the ETABS objects. They are the basic building blocks for your model. If you are new to ETABS 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.



Tip:

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

Understanding area, line and point objects is probably the most important key to successful modeling in ETABS. Following is a list of chapters in this manual that discuss the ETABS objects in detail. The titles for each chapter are self-explanatory.

- Chapter 23: Area Objects
- Chapter 24: Line Objects
- Chapter 25: Point Objects

The ETABS Analysis Model

In general, it is not necessary that you concern yourself with the particulars of the ETABS analysis model. Nevertheless, it may be helpful for you to know that when you run an analysis, ETABS 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 you as 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.

ETABS 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. This is described in the section titled “Automatic Meshing of Line Objects” in Chapter 30.

 **Tip:**
If you are a SAP2000 user, you can export your object-based ETABS model to 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

Also, if your model includes horizontal area objects that are assigned deck or slab properties with membrane behavior only, then ETABS can automatically mesh the floor into the analysis model. You do not need to mesh these types of floors in your object-based model. See the section titled “Automatic Meshing of Area Objects” in Chapter 30 for more information.

All other types of floors, and all ramps and walls must be adequately meshed in your object-based model because ETABS does not automatically mesh these into the analysis model. See Chapter 31 for discussion of manual meshing techniques that are available in ETABS.

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. The process for this is described in Chapter 32. We recommend that if you want to read this chapter you first read Chapter 30 because it contains important background information (particularly the information about imaginary beams) that will help you understand Chapter 32.

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



Chapter 6

ETABS Modeling Tips

This chapter outlines a basic modeling process for creating your ETABS models and then provides you with a list of modeling tips.

Modeling Process

This section lays out a general process that you can follow to create a model in ETABS. It is not necessary, or expected, that you exactly follow the process here. The intent of this section is to help keep you headed in the right direction as you create your model and run the analysis.

Following is a process you might use to create a typical ETABS model:

1. Set the current units to those that you will want to use most often in your model. See Chapter 20 for more information.

2. Start your model by clicking the **File menu > New Model** command and choose one of the file initialization methods. See the subsection titled "Initialization of a New Model" in Chapter 8 for more information.
3. Set up your grid lines. See the subsection titled "Defining a Grid System" in Chapter 8 for more information.
4. Define your story levels. See the subsection titled "Defining Story Data" in Chapter 8 for more information.
5. If desired add structural objects from one of the built-in ETABS templates. See the subsection titled "Adding Structural Objects from Template" in Chapter 8 for more information. In general we recommend that you start your model by adding objects from a template whenever possible.
6. Use the **Options menu > Preferences** command to modify any of the default preferences if desired. See the section titled "Preferences" in Chapter 18 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 11 for more information.
8. Use the **Define menu > Static Load Cases** command to define your static load cases. See the section titled "Static Load Cases" in Chapter 11 for more information.
9. If you are using mass in your model then use the **Define menu > Mass Source** command to specify the source of mass in your model. See the section titled "Mass Source" in Chapter 11 for more information.
10. Use the commands available on the Draw menu to draw additional area, line and point objects as needed. See Chapter 12 for documentation of the drawing features in ETABS.

When you draw the objects we recommend that you assign structural properties to them as you draw them using the floating 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 for snapping (see Chapter 12), guidelines for extending or trimming other lines (see Chapter 8), or mesh lines for manual meshing (see Chapter 31).

Note that mass is required if you are doing a modal analysis to determine mode shapes (Chapter 33). It is also required for the non-iterative method of considering P-Delta (Chapter 33). 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 9 for documentation of the various Edit menu features.

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 14 for documentation of the various Assign menu features.

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

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

13. Use the **Display menu > Show Loads** command and the **Display menu > Set Input Table Mode** command to review your input. Both of these commands are documented in Chapter 16.

Another way to review your input is to right click on any object. This brings up a dialog box where you can review all assignments to the object. See Chapter 23, 24 and 25 for documentation of this feature for area, line and point objects, respectively.



Tip:

The ETABS similar stories feature is a useful and powerful tool that you can use when drawing, selecting and making assignments to objects in plan view. See the section titled "Similar Story Levels" in Chapter 22 for more information.

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

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 titled "Printing from ETABS" in Chapter 8 and see Chapter 41 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.

15. Use the **Analyze menu > Set Analysis Options** to specify various analysis parameters such as the building degrees of freedom. See the section titled "Analysis Options" in Chapter 15 for more information.
16. If your model has floors, walls or ramps that require manual meshing then use the manual meshing options available through the **Define menu > Mesh Areas** command to mesh these objects. See Chapter 31 for discussion of the manual meshing features.

Note that ETABS 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.

17. Use the **Analyze menu > Run Analysis** command to run your analysis. See the section titled "Run Analysis" in Chapter 15 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.



Note:

Note that ETABS can automatically mesh floors that have membrane properties only.



Tip:

We recommend that you run large analyses minimized.

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 16 for documentation of the Display menu features.

Note that output conventions for various objects and elements are discussed in Chapters 34 through 39.



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 titled "Printing from ETABS" in Chapter 8 and see Chapter 41 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 ETABS design postprocessors.
 21. After you have run a design, save your model before exiting ETABS. Otherwise your design is not saved.

Modeling Tips

Following is a list of modeling tips that may help you as you create your models.



Tip:

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

1. Create a Default.edb file that contains standard preferences for your company and then use it. See the subsection titled "Initialization of a New Model" in Chapter 8 for more information.
2. Save your model often. It is also useful to occasionally save a backup of your model under a different name.
3. With the powerful graphical interface of ETABS 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 ETABS by "surfing" through the various ETABS menus. Also refer to Appendix 1 which lists the complete ETABS menu structure.
5. The **Edit menu > Undo** command provides ETABS with powerful Undo capabilities. Thus you should feel free to experiment with options in ETABS 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. Keeping the auto select lists shorter, say 20 to 30 sections long, will significantly speed up the time it takes to design your model.
7. When creating your 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 available in ETABS. This provides a powerful way for you to work on multiple faces of your building in the same 2D view. See the section titled "Developed Elevations" in Chapter 12 for more information.

8. When creating or editing your model in plan view consider using the similar stories feature of ETABS. See the section titled "Similar Story Levels" in Chapter 22. If you use the similar stories feature, take care to keep track of when you have it enabled (set to Similar Stories or All Stories) or disabled (set to None). This helps avoid using the feature when you don't mean to.
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.



Tip:

In ETABS 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.

- b. Use the **File menu > Import > Overwrite Story from ETABS7 .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, suppose you have a ten-story building. Further suppose that floor levels 3 through 10 are exactly the same, and floor level 2 and the roof are unique. For floor design you can create three single story models. One single story model is for level 2, one is for levels 3 through 10 and one is 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 you can just 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.

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. These story levels can then be combined into one building using the **File menu > Import > Overwrite Story from ETABS7 .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, then you can use the **File menu > Export > Save Story as ETABS7 .edb File** command to export a story to another file as a single story model.

**Note:**

In ETABS you can easily model structures with sloping floors (ramps). For example, you can easily model parking garages.

12. The ETABS 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 titled "The ETABS Aerial View" in Chapter 4 for more information.
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 in ETABS 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 in ETABS. 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. ETABS 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 titled "Mass Source" in Chapter 11 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. ETABS automatically generates static lateral seismic and wind loads based on building code requirements. See the section titled "Static Load Cases" in Chapter 11 and see Chapters 28 and 29 for more information.
19. When drawing objects in your model we recommend that you assign structural properties to them as you draw them using the floating properties of object box. See the sections titled "Floating Properties of Object Window for Line Objects" and "Floating Properties of Object Window for Area Objects" in Chapter 12 for more information.

**Tip:**

To help draw objects in your model accurately make use of the ETABS snap options. Reference planes and reference lines can assist you when snapping. See the section titled "Reference Planes and Reference Lines" in Chapter 9 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.

20. It is important that you draw your ETABS model accurately. The ETABS snap options can help you do this. See the subsection titled "ETABS Snap Options" in Chapter 12 for more information.

If you do not draw your model accurately then ETABS may not interpret the member connectivity in the way you intend. For example, suppose 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 this case, depending on the tolerances set and how far the end of the beam is from the girder, ETABS may not interpret the beam as connecting to the girder. You can avoid this problem by using the ETABS snap options so that when the line object representing the beam is drawn it snaps on to 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 then you can use the Align features in ETABS to fix the problem. See the section titled "Aligning Points, Lines and Edges" in Chapter 9 for more information.

21. Groups can be a great benefit when creating a model in ETABS. You can select elements by groups. Suppose 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. Once the braces are selected you can make assignments to them as a group or you can 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 ETABS design postprocessor.

See the section titled "Group Name Assignments" in Chapter 14 and see Chapter 26 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.

23. 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. See Chapter 32 for more information on transformation of loads into the analysis model.
24. 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. Be sure to have a look at these.
25. 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.
26. 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.



Overview of the ETABS Menus

General

There are eleven menus provided in ETABS. They are, in order, working from left to right across the menu bar:

- File menu (see Chapter 8).
- Edit menu (see Chapter 9).
- View menu (see Chapter 10).
- Define menu (see Chapter 11).
- Draw menu (see Chapter 12).
- Select menu (see Chapter 13).
- Assign menu (see Chapter 14).
- Analyze menu (see Chapter 15).

- Display menu (see Chapter 16).
- Design menu (see Chapter 17).
- Options menu (see Chapter 18).
- Help menu (see Chapter 19).

Each of the menus is documented in a separate chapter as indicated in the above list. Most of the commands available in the menus are documented in these chapters. In addition, Appendix 1 lists the complete menu structure of ETABS including all of the commands available on each menu.

For the most part the items in the menus are arranged in a logical way. For example items associated with drawing objects are on the Draw menu, items associated with making assignments to an object are on the Assign menu and so on. Thus when you are looking for a menu command and can not remember which menu it is in think about what it is you are trying to do and in most cases you can probably guess the menu that the command is on.

For example suppose you need to create some section properties for your slab over metal deck and you can't remember where to go to do this. Thinking logically and keeping the eleven ETABS menus in mind, what you need to do is *define* the deck sections. Thus you would go to the Define menu where you would find the Define Wall/Slab/Deck Sections command.

The ETABS File Menu

General

The File menu in ETABS 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 discusses the features available on the File menu.

Starting a New Model

The **File menu > New Model** command is used to start/create a new ETABS model. Alternatively you can click the **New Model** button, , located on the main (top) toolbar. There are four distinct stages in creating a new model. They are:

- Initialize the model
- Define a grid system

- Define story data
- Add structural objects from a template

There are default values provided for each of these stages such that you can do little more than click a few **OK** buttons and you will have a complete model created with default dimensions and properties. More typically you will modify the default values provided for each of these stages to specify the particular characteristics of your model. The following four subsections describe each of these four stages of creating a new model.

Initialization of a New Model

When you execute the **File menu > New Model** command a message is displayed asking if you want to initialize your new model with definitions and preferences from an existing .edb file. You can answer either Choose .edb, Default.edb or No to this question.

If you answer Choose .edb then you specify an ETABS file that has an .edb extension. ETABS then starts your new model with the definitions and preferences from the specified .edb file. In this case ETABS 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.

If you answer Default.edb to the question then ETABS starts 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 there is no Default.edb file that is in the same directory as the ETABS.exe file, or if you answer No to the

question, then ETABS uses built-in values for all of the definitions and preferences in your new model.

Our intent is that you use this feature in the following way:

- Create .edb files to initialize your model:
 - ✓ Create a Default.edb file that is specifically tailored to the most common practices of your office. Store this file in the same directory that your ETABS.exe file is located in. Note that the Default.edb file is simply a typical ETABS .edb file that has been named Default.
 - ✓ If necessary create other ETABS .edb files that may in certain circumstances be useful for initializing your model. You may want to store these in a location where all engineers have access to them or you may want these to be more personalized and have limited access. Either way is fine as far as ETABS is concerned. It works best if these files have names different from Default.edb so that you do not confuse the files.
- When you start a new model and the message box asks if you want to initialize your new model with definitions and preferences from an existing .edb file do one of the following:

- ✓ Click the **Default.edb** button. This means that the definitions and preferences will be initialized (get their initial values) from the Default.edb file that is in the same directory as your ETABS.exe file. If the Default.edb file does not exist in this directory then the definitions and preferences are initialized using ETABS built-in defaults.

You should create your Default.edb file such that you most commonly click this button.

- ✓ In some cases you may want to click the **Choose .edb** button and specify a different file from which the definitions and preferences are to be initialized. For example, a certain client or project may require



Tip:

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

certain things in your model to be done in a certain way that is different from your typical office standards. You could have a specific .edb file set up for this client or project which could then be used to initialize all models for the client or project.

- ✓ Click the **No** button if you just want to use the built-in ETABS defaults.

8

Defining a Grid System

Once you tell ETABS your intentions for model initialization the Building Plan Grid System and Story Definition dialog box appears. In the Grid Dimensions (Plan) area of this dialog box you can define a grid line system. There are 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 direction 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. You can later edit this information using the **Edit menu > Edit Grid Data** command. Refer to the section titled "Editing Coordinate System Grid Line Data" in Chapter 9 for discussion of this command.
- **Custom Grid Spacing:** This option allows you to define nonuniformly spaced grid lines in the X and Y directions for the global coordinate system. After you choose this option you click the **Edit Grid** button and a dialog box appears where you can define the grid lines. This dialog box is the same one that appears when you execute the **Edit menu > Edit Grid Data** command, highlight the GLOBAL coordinate system and click the **Modify>Show System** button.



Tip:

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

There are several reasons why you should define a grid system for your model. They include:

- Default elevation views in the model occur at each defined primary grid line in your model.

- If you add structural objects to your model from a template then those objects are added based on the grid line definitions in your model.
- You can snap to grid lines when you draw objects in your model.
- You can mesh objects at their intersections with grid lines.
- You can define the same grid lines, with the same names, in your model as are on the building plans. This may allow for easier identification of specific locations in your model.

When you use the Custom Grid Spacing option, whatever is specified for the Uniform Grid Spacing option is used to provide the initial values in the grid definition dialog box. Note that the data for the Uniform Grid Spacing option becomes grayed out (inactive) when you select the Custom Grid Spacing option.

Regardless of which option you use to initially define grid lines you can later define additional coordinate/grid systems in your model using the **Edit menu > Edit Grid Data** command.

Defining Story Data

Note:

See the Section titled "Similar Story Levels in Chapter 22 for discussion of the ETABS similar stories feature that allows you to specify customized story similarity.

Story data is defined from the Story Dimensions area of the Building Plan Grid System and Story Definition dialog box. There are two options for defining the story data:

- **Simple Story Data:** Here you simply define the number of stories and a typical story height that is used for all story levels. Using this option ETABS 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** command. Refer to the section titled "Editing Story Data" in Chapter 9 for discussion of this command.

- **Custom Story Data:** This option allows you to define your own story names, story levels of nonuniform height and customized story similarity. After you choose this option you click the **Edit Story Data** button and the Story Data dialog box appears. This dialog box is the same one that appears when you execute the **Edit menu > Edit Story Data >Edit** command. Refer to the section titled "Editing Story Data" in Chapter 9 for discussion of this command.

Note that when you use the Custom Story Data option whatever is specified for the Simple Story Data option (which becomes grayed out, that is, inactive when you select the Custom Story Data option) is used to provide the initial values in the Story Data dialog box.

Adding Structural Objects from a Template

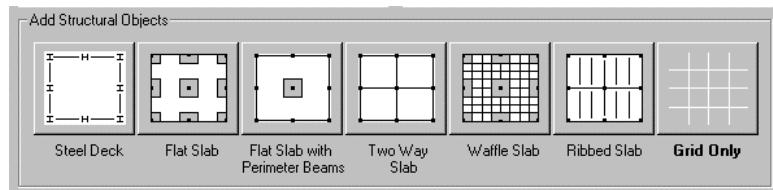
Large models can be generated automatically using ETABS built-in templates. The models can then be modified with onscreen editing to satisfy specific situations.

Note:

You can create steel and concrete building models using built-in ETABS templates. These models can then be modified with onscreen editing to satisfy specific situations.

While you are in the Building Plan Grid System and Story Definition dialog box you can also 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. 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 from Template area of the Building Plan Grid System and Story Definition dialog box is reproduced below 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) is currently selected in Add Structural Objects from Template area because its name is highlighted. When the dialog box is initially open the Grid Only selection is selected.

You can choose any of the templates by simply left clicking its associated button. When you choose one of the template buttons another dialog box appears where you can specify various types of data for the template. The data specified for each of the six templates (one steel and five concrete) are discussed in subsections below.

When you are finished specifying data for a template you click the **OK** button to return to the Building Plan Grid System and Story Definition dialog box. You may notice that the button name for the template that you just specified is highlighted. If you then decide you defined the wrong type of template you can simply click another template button and define that data for it. When ETABS creates the model it only adds structural objects based on the last button you clicked in the Add Structural Objects from Template area, that is, the highlighted button. If the last button clicked was **Grid Only** then no structural objects are added to your model from template. You will simply start out with a grid system. You can then:

- Use the commands from the Draw menu to draw objects.
- Use the **Edit menu > Add to Model from Template** command to add objects to your model. See the section titled "Add to Model from Template" in Chapter 9.
- Import stories from a SAFE .f2k file. See the section titled "Importing Files" later in this chapter.
- Import stories from another ETABS .edb file. See the section titled "Importing Files" later in this chapter.
- Copy objects (geometry only) from another ETABS .edb file. See the section titled "Cut, Copy, Paste and Delete" in Chapter 9.

- Copy objects (geometry only) from a spreadsheet. See the section titled "Cut, Copy, Paste and Delete" in Chapter 9.
- Use commands from the Edit menu to modify existing objects.

Note that when using the ETABS concrete building templates 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 in ETABS beams are modeled as line elements. Thus slabs with out-of-plane bending capability span from center of beam to center of beam in the ETABS model.

Steel Floor System Template

This template allows you to define a steel floor system. Following are discussions of each of the areas in this template:

- **Slab Edge Distances:** 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 can not be negative.
- **Secondary Beams:** Checking the secondary beams check box means to include secondary (infill) beams. If the check box is not checked then no secondary beams are included. Secondary beams are the beams that do not frame into columns. 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 ETABS 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 options for this item:
 - ✓ **No Moment Frame:** Moment is released at all beam to column connections, that is, these 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 at perimeter columns are moment resistant and at interior columns they are pinned.
 - ✓ **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 are:
 - ✓ **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 only be one) and then define additional uniformly distributed dead load for that case. 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.
- The Live Load Case drop down box lists all defined static loads that are type Live. You can choose any one of these load cases in the drop down box and then define uniformly distributed live load for that case.
- **Create Rigid Floor Diaphragm:** Checking this box applies a rigid diaphragm constraint to the area object representing the slab/deck.

Flat Slab Template

This template allows you to define a concrete flat slab floor system with drop panels. No beams are included in this floor system. Following are discussions of each of the areas in this template:

- **Slab Edge Distances:** 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 can not be negative.
- **Drop Panels:** Checking the drop panels check box means to include drop panels in the model. If the check box is not checked no drop panels are included.

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 half of the drop panel size then the drop panel is truncated at the edge of the slab.

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 dialog box.

- **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 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 are:
 - ✓ **Column:** All columns in the template model.
 - ✓ **Slab:** The floor slab not including drop panels.
 - ✓ **Drop:** 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 only be one) and then define additional uniformly distributed dead load for that case. 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.

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

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

Flat Slab with Perimeter Beams Template

This template 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 one includes beams framing 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. Following are discussions of each of the areas in this template:

- **Slab Edge Distances:** 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 can not be negative.
- **Drop Panels:** Checking the drop panels check box means to include drop panels in the model. If the check box is not checked no drop panels are included.

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 half of the drop panel size then the drop panel is truncated at the edge of the slab.

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 dialog box.

- **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 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:** All columns in the template model.
 - ✓ **Beam:** All perimeter beams in the template model.
 - ✓ **Slab:** The floor slab not including drop panels.
 - ✓ **Drop:** 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 only be one) and then define additional uniformly distributed dead load for that case. 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.

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

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

Two-Way Slab Template

This template 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. Following are discussions of each of the areas in this template:

- **Slab Edge Distances:** These are the distances from the perimeter grid line to the edge of the slab. These dis-

tances must be greater than or equal to zero. They can not be negative.

- **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 the columns and beams and you specify slab section properties to be assigned to the slab. The items in this area are:
 - ✓ **Column:** All columns in the template model.
 - ✓ **Beam X:** All beams in the template model that span in a direction parallel to the X-axis.
 - ✓ **Beam Y:** All beams in the template model that span in a direction parallel to the Y-axis.
 - ✓ **Slab:** 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 only be one) and then define additional uniformly distributed dead load for that case. 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.

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

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

Waffle Slab Template

This template 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 is modeled as fully moment resistant, as one would typically expect for a concrete structure. Following are discussions of each of the areas in this template:

Note:

In waffle slabs ETABS 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.

- **Slab Edge Distances:** 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 can not 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 are included.

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 half of the drop panel size then the drop panel is truncated at the edge of the slab.

The actual size of a drop panel included in a model may be less than that you input into the template. This happens because in waffle slabs ETABS 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 dialog box.

Checking the ribs check box means to include waffle slab ribs in the model. If the check box is not checked no slab ribs are 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 all occurs 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 this is shown in the next subsection for a one-way ribbed slab.

- **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 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:** All columns in the template model.
 - ✓ **Ribs:** All waffle slab ribs (in both the X and Y directions) in the template model.
 - ✓ **Slab:** The floor slab not including drop panels.
 - ✓ **Drop:** 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 only be one) and then define additional uniformly distributed dead load for that case. 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.

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

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

Ribbed Slab Template

This template allows you to define a one-way ribbed concrete floor slab system. No drop panels are included in this template. Note that the connections between the elements are modeled as fully moment resistant, as one would typically expect for a concrete structure. Following are discussions of each of the areas in this template:

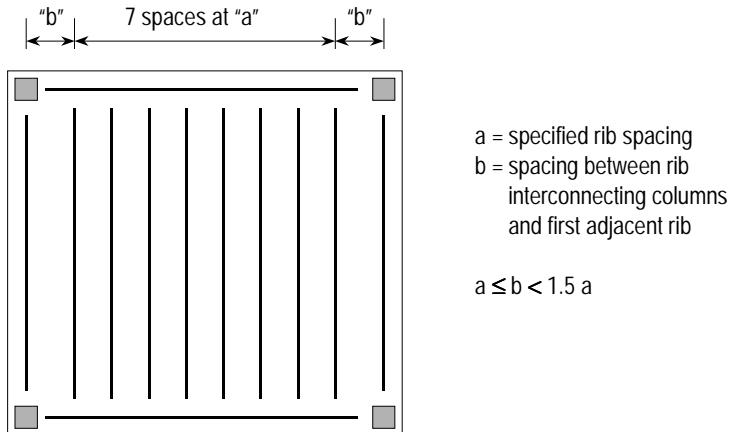
- **Slab Edge Distances:** These are the distances from the perimeter grid line to the edge of the slab. These distances must be greater than or equal to zero. They can not be negative.

Ribs: Checking the ribs check box means to include the one-way slab ribs (joists) in the model. If the check box is not checked no slab ribs are 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 all occurs 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 size of the ribs (beams) on the grid lines if necessary to maintain a constant form size for your one-way ribbed slab.

Consider the example shown in Figure 8-1. In this example the dimension “a” is the specified rib spacing. The

Figure 8-1:
Example of rib
spacing for a one-
way ribbed slab
(waffle slab similar)



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

$$a \leq b < 1.5a$$

Rib Spacing for One-Way Ribbed Slab

dimension "b" is the spacing between the ribs interconnecting columns and the first adjacent rib. Note that $a \leq b < 1.5a$.

The direction of the ribs can be specified as either X or Y. Specifying the rib direction as X means that the ribs span in a direction parallel to the X-axis. Specifying the rib direction as Y means that the ribs span in a direction parallel to the Y-axis.

- **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 the columns, beams and ribs and you specify slab section properties to be assigned to the slab. The items in this area are:
 - ✓ **Column:** All columns in the template model.
 - ✓ **Beams:** All beams (girders) in the template model. These span perpendicular to the slab ribs (joists).
 - ✓ **Ribs:** All one-way slab ribs in the template model.

- ✓ **Slab:** 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 only be one) and then define additional uniformly distributed dead load for that case. 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.

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

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

Opening an Existing Model

To open an existing ETABS version 7 or later model click the **File menu** > **Open** command or the **Open .EDB File** button, , located on the main (top) toolbar and the Open Model File dialog box appears. Find the name of the file you want to open in this dialog box and double click on it so that it appears in the File Name edit box. Alternatively you can just type the name of the file in the File Name edit box, including the path if necessary. Then click the **Open** button.

Another way to open an existing ETABS model is to select it from the list of recently opened ETABS models that is displayed near the bottom of the ETABS File menu. This list appears just below the Display Input/Output Text Files item and just above the Exit item. If the model you want to open appears in this list then simply click on its name in this list 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 then your list of recently used files disappears.

When you open an existing .edb file ETABS always creates a backup of that file, as it is when you first open it. The backup file has the same name as your .edb file but it has a .ebk extension. 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 as it was when you last opened it in the .ebk file. If you need to access the .ebk file simply change its extension to .edb and use it like a regular .edb file.

8

Saving Models



Tip:

Save your file often!

To save your ETABS model click the **File menu > Save** command or click the **Save Model** button, , located on the main (top) toolbar. If you have just created your model and this is the first time you are saving your model the Save Model File As dialog box appears where you can specify a location (directory) and name for your model file. Specify the location and name in the dialog box and click the **Save** button to save your model.

If your model previously existed or has been previously saved then 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: We recommend that you make it a habit to save your model file early and often. This helps minimize the lost work that may occur as a result of any power failures, computer malfunctions or unforeseen software behavior. Note that there is *not* an AutoSave feature in ETABS. Thus it is fully your responsibility to save your file.

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

When you save a model file ETABS 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 .Set extension is intended as a text backup file of your .edb file which is a binary file. If something happens to your .edb file of your model, such as the file becomes corrupted or is otherwise lost, you can restore your model by importing the .Set file. Use the **File menu > Import > Open ETABS7.e2k Text File** command to import a .Set file. Specify the name of your .Set file, including the .Set extension, in the resulting dialog box.

Note that a .Set file is exactly the same as a .e2k file that you can export using the **File menu > Export > Save Model as ETABS7.e2k Text File** command. The .Set file is created (and the previous .Set 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 .Set 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.

Importing Files

Note:

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

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

- **Open ETABS7.e2k Text File:** This command is used to import ETABS7 .e2k and .Set text input files. If another model is currently open this command will close that model (prompting you to save it if necessary) and open a new one for the imported file.
- **Open ETABS6 Text File:** This command is used to import an ETABS6 text input file. ETABS6 Steeler, Conker and Waller input files are not imported. If another model is currently open this command will close that model and open a new one for the imported file.

- **Overwrite Stories from SAFE.f2k Text File:** This command behaves a little differently depending on whether or not a model is open when you execute the command.

If a model is currently open when you start this import then ETABS deletes all of the objects and their assignments at the specified story level in the currently open model and replaces them with the objects and their assignments from the specified SAFE model. Grid lines are not imported from the SAFE model in this case.

If no model is currently open then a one-story ETABS model is created and the SAFE objects and their assignments are imported into this model. The SAFE grid line definitions are also imported in this case.

- **Overwrite Stories from ETABS7.edb File:** This command is only active if a model is currently open. It will delete all of the objects and their assignments at the specified story level in the currently open model and replace them with the objects and their assignments from a specified story level in the file you are importing from. The story level that you are importing to in the current model can be different from the story level that you are importing from. Grid lines are not imported using this command.

Exporting Files

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

- **Save Model as ETABS7.e2k Text File:** This command saves the model as a .e2k text input file. You can later import this file/model back into ETABS using the **File menu > Import > Open ETABS7.e2k Text File** command if you wish.

**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 5 for additional information.

- **Save Model as SAP2000.s2k Text File:** This command 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 command 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 can not 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 perform a more refined analysis and design of a concrete floor in your ETABS model.

**Note:**

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

8

**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.

- **Save Story as ETABS7 .edb File:** This command saves the specified story level as an ETABS7 .edb. If the story you are exporting is the bottom story of a structure then the restraints are included in the exported one-story model. If the story level exported is not the bottom level of a structure then ETABS fixes the base of all columns and walls in the exported one-story structure.
- **Save Story Plan as .DXF:** This command is only active if the currently active window is showing a plan view. This command only exports undeformed geometry of the structure and text. It does not export any analysis output results. All items that are exported must be showing in the currently active window when the export is done.

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 these 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 then 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.

- ✓ **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 dialog box 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 dialog box.
 - ✓ **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.
- **Save as 3D .DXF:** This command only exports undeformed geometry of the structure. It does not export any analysis output results and it does not export text.

**Note:**

*ETABS does
not export
analysis results
to .DXF files.*

The types of items listed below can be exported to a 3D .DXF file. A layer name can be specified for each of these 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 then set its layer name to None.

- ✓ **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 can not specify a beam offset for a 3D .DXF file like you can for a plan view .DXF file.
- **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. See Chapter 42 for documentation of this database file.
- **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).

Creating Videos

You can create videos in ETABS 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 media player that comes with Windows.



Tip:

You can create videos of time history response in ETABS (as .avi files) and then play back the videos using the media player that comes with Windows.

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 then the value of number of frames per second may be adjusted up or down to speed the animation up or 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 ETABS windows are showing undeformed views before attempting to create a time history video.

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.

Printing from ETABS

The majority of the printing from ETABS is done using the print commands available on the File menu. The two main printing commands on the File menu are the **File menu > Print Graphics** command and the **File menu > Print Tables** command.

Printing Graphics

Note:

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

The **File menu > Print Graphics** command prints whatever graphics are displayed in the active window to the printer that is currently specified 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 dialog box which is accessed using the **Options menu > Colors** command.

The colors (grayscale) used for black and white printers are those displayed in the Assign Display Colors dialog box when the Device Type option is set to Printer. The colors used for color printers are those displayed in the Assign Display Colors dialog box 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 dialog box when the Device Type option is set to Screen. If the object display colors are set differ-

ently for screen display and color printing then the objects will print in different colors than they display in on the screen.

Printing Text Input and Output Tables

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.

Analysis input and output data are discussed in Chapters 40 and 41 of this manual. Design input and output data are discussed in the design manuals.

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

Printing Analysis Input Data

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 dialog box where you can specify the types of input data that you want to print. See Chapter 40 for documentation of the items that can be printed from this dialog box.

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 can not be printed.

Printing Analysis Output Data

Use the **File menu > Print Tables > 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 dialog box where you can specify the types of output data that you want to print. See Chapter 41 for documentation of the items that can be printed from this dialog box.

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 can not be printed.

User Comments and Session Log

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. These 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 ETABS also occasionally adds comments to this file. You can modify or delete those comments as well.

Displaying Input/Output Text Files

The **File menu > Display Input/Output Text Files** command provides a convenient way for you to view input and output text files associated with ETABS. The input text files you might want to view include those with the .Set 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 these files have a .txt extension.

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

Exiting ETABS

8

You can use the **File menu > Exit** command to exit the ETABS program. Other ways you might also exit the ETABS program include clicking the X in the upper right hand corner of the ETABS window and right clicking the ETABS program button on your Windows taskbar and choosing Close from the resulting popup menu. If you have made changes to your model since you last saved it then ETABS prompts you to save your model when you exit using any of these methods.

Note that exiting the ETABS program in any of these ways not only closes your model but also closes the entire ETABS program. If you simply want to close one model and open another (or start a new one) then just click on the appropriate command in the File menu to open another model or start a new one. If you have made changes to your model since you last saved it then ETABS prompts you to save your model before beginning work on the next one.

The ETABS Edit Menu

General

The Edit menu in ETABS provides some basic tools for editing (modifying) the geometry of your ETABS model. This chapter discusses many of those tools. It also discusses the Reshaper tool that is available on the side toolbar and in the Draw menu as well as the ETABS nudge feature.

Some of the editing tools not discussed in this chapter are discussed in other places in this manual. The Undo and Redo commands are described in the section titled "Undo Features in ETABS" in Chapter 4. (Also see the section titled "Editing Story Data" later in this chapter for additional information on the Undo feature.) Meshing of area objects is explained in detail in Chapter 31. Automatic relabeling of objects is discussed in the sub-section titled "Relabeling Objects" in Chapter 23.

Cut, Copy and Paste

In general the cut, copy and paste commands work similarly to the standard cut, copy and paste Windows commands. However some of the behavior of these commands is specific to ETABS.

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

- The **Cut** command deletes the selected objects that are associated with the story level that is associated with the plan view in the currently 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. Also 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 besides the above-mentioned section property name are copied to the clipboard. The geometry and section property names associated with the cut objects can be pasted back into ETABS, or they can be copied to a spreadsheet, such as Microsoft Excel, in a text format. Additional discussion of the spreadsheet option is provided later in this section.
- The **Copy** command copies the geometry of the selected objects that are associated with the story level that is associated with the plan view in the currently active window to the Windows clipboard. 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 ETABS, or they can be copied to a spreadsheet, such as Microsoft Excel, in a text format. Additional discussion of the spreadsheet option is provided later in this section.



Tip:

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

- The **Paste** command copies geometry and section property names from the Windows clipboard into your ETABS model on the story level that is 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 ETABS or from a spreadsheet.

The similar stories feature of ETABS discussed in the section titled "Similar Story Levels" in Chapter 22 works for pasting objects into ETABS.

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.



Note:

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

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

You can see the text format used when ETABS geometry is copied to or from a spreadsheet by simply 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.

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 Headings in Spreadsheet

Following are the definitions for the headings for point objects in the spreadsheet data.

- **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 ETABS to the point object. DZ is 0 if the point object falls 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 then 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 titled "Reference Planes and Reference Lines" later in this chapter for discussion of reference planes.

Line Object Headings in Spreadsheet

Following are the definitions for the headings for line objects in the spreadsheet data.

- **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 then it is NONE.

If a line object has a frame section property name in a spreadsheet for which a frame section property is not already defined in the ETABS model, then ETABS sets the frame section property assignment for that object to None in ETABS when the spreadsheet data is pasted into the model. For example, if a line object has a frame section property name of COL1 in a spreadsheet, and there is no previously defined frame section property named COL1 in the ETABS model, then ETABS sets the property assignment for that line object to None in the ETABS model when the spreadsheet data is pasted into the model.

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 associated with the plan view where the line object is pasted into ETABS to the considered end point of the line object. DZ is 0 if the end point falls 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 then 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 titled "Reference Planes and Reference Lines" later in this chapter for discussion of reference planes.

- **BelowI (BelowJ):** This is a flag that indicates if the considered end point of the line object falls at the story level below the story level associated with the plan view where the line object is pasted into ETABS. This item can either be Y for Yes or N for No. Y means it does fall at the story level below where it is pasted; N means it does 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 ETABS.

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 Object Headings in Spreadsheet

Following are the definitions for the headings for area objects in the spreadsheet data.

- **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 then it is NONE.

If an area object has a property name for which a property is not already defined, then ETABS sets the property assignment for that object to None. For example if an area object has a property name of WALL1 and there is no previously defined wall, slab or deck section property named WALL1 then ETABS sets the property assignment for that 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 ETABS to the considered corner point of the area object. DZ is 0 if the corner point object falls 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 then 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 titled "Reference Planes and Reference Lines" later in this chapter for discussion of reference planes.

- **Below-n:** This is a flag that indicates if the considered corner point of the area object falls at the story level below the story level associated with the plan view where the area object is pasted into ETABS. This item can either be Y for Yes or N for No. Y means it does fall at the story level below where it is pasted; N means it does 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 ETABS.

An example where this is used is for a wall. 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

In general the delete command in ETABS works like the standard Windows delete command. This command deletes the selected object(s) and all of its assignments (loads, properties, supports etc.). Alternatively you can select the objects and press the Delete key on your keyboard to accomplish the same thing.

Add to Model From Template

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

Our intent is that you use the two-dimensional option to locate planar frames throughout a model. The three-dimensional option is intended to assist in modeling conditions where several towers rest on the same base structure.

Two-Dimensional Frame

You can add a two-dimensional frame or wall to your model using this feature. You specify the following data for this option:

- 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.

Three-Dimensional Frame

You can add a three-dimensional frame to your model using this feature. Except for the location data, the data you specify for this option is similar to that specified when you start a model from template. The following information is provided to locate the added 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) then click the **Advanced** button and fill in the starting and ending grid line ID's and the story levels at top and bottom of frame.

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

Replicating Objects

The **Edit menu > Replicate** command is a powerful way to copy objects including most of their assignments. Four types of replication are available. They are linear, radial, mirror and story. Each of these is discussed in a separate subsection below. In addition, the assignments that are and are not replicated and the available user control on which assignments you want to replicate are discussed in the subsection below titled "Assignments that are Replicated."

Note:

Replication copies objects and their assignments.

To replicate one or more objects you select the object(s) and then use the **Edit menu > Replicate** command to specify the desired replication option. The selected objects, including all of their assignments are replicated (copied) as indicated.

Important note: When using the replication feature if a replicated object falls in exactly the same location as an existing object then the replication is not done at that location. The existing object remains as it was. The replication is still done at other locations.

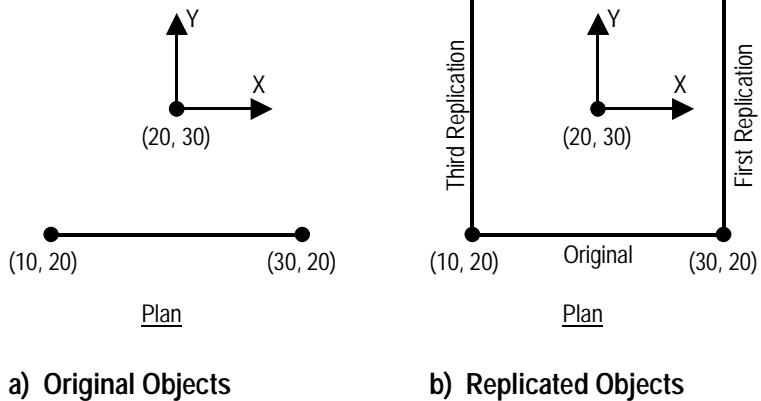
Linear Replication

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

Radial Replication

For radial replication you 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 replicated. The object and its assignments are then copied the specified number of times, each time incrementing the location of the objects by the specified rotation angle.

Figure 9-1:
Example of radial replication



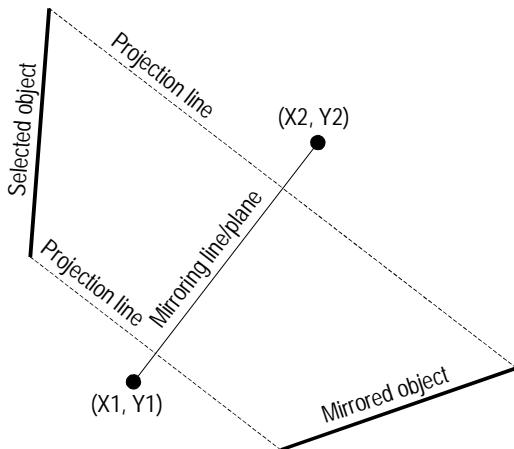
Two options are available for specifying the point to rotate about. You can either tell ETABS to rotate about the center of the selected objects or you can give it the global X and Y coordinates of a specific point to rotate about.

When the rotation occurs about the center of the selected objects ETABS 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 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 9-1 shows an example of radial replication. Figure 9-1a 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 9-1b shows the result of the replication.

Figure 9-2:
Example of mirror
replication



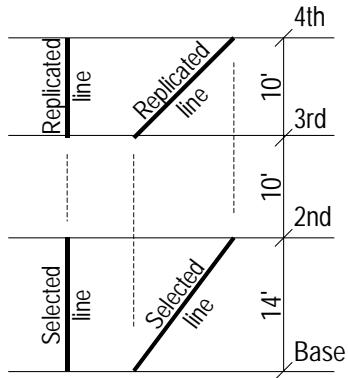
Mirror Replication

For mirror replication you specify a line in the global X-Y plane to mirror about or if you prefer you can 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.

You specify the line in the X-Y plane by specifying two points (X_1, Y_1) and (X_2, Y_2) in global coordinates. ETABS replicates the selected objects by mirroring the objects and their assignments about the specified line/plane. Figure 9-2 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.

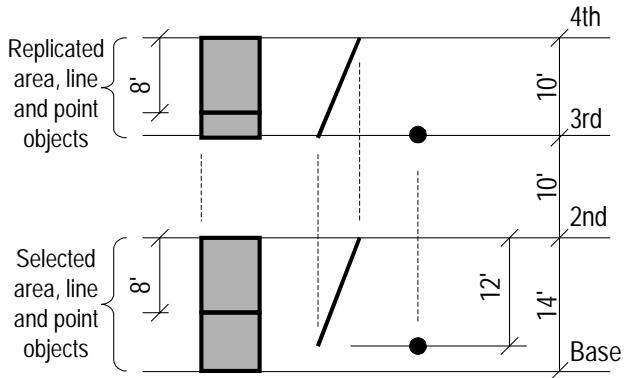
Story Replication

For story replication you specify 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 then you should be aware of the following:



a) Elevation

(Above)
Figure 9-3:
Examples of story replication



b) Elevation

- 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 9-3a 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 falls below the bottom of story level X, then 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 9-3b shows some examples. Note that the height of the lower replicated area object on the left side of Figure 9-3b is reduced from 6 feet to 2 feet when it is replicated in order to fit into the 4th story level. Also note that the line object is taken to the bottom of the story level since 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.

Assignments that are Replicated

All assignments to area, line and point objects are replicated except for:

- 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. Those assignments are listed in Table 9-1.

Table 9-1:

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	

(Below)

Table 9-2:

Object assignments whose replication you can control

The area, line and point object assignments whose replication you can control if you wish are listed in Table 9-2.

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
Link properties	Link properties	Temperature Loads
Forces	Nonlinear hinges (pushover)	
Ground displacements	Property modifiers	
Temperatures	Point loads	
	Distributed loads	
	Temperature loads	

You can control these object assignments by clicking the **Options** button when you are in the Replicate dialog box. This brings up the Replicating Object Assignments dialog box where you can check or uncheck boxes to indicate which assignments you want replicated. By default all assignments are replicated except for the diaphragm, pier and spandrel label assignments discussed at the beginning of the subsection which are never replicated.

9

Editing Coordinate System Grid Line Data

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. See Chapter 21 for additional information on coordinate systems and grid lines.

Grid lines may be edited in one direction at a time. When editing grid line data for a particular grid line the following information can be provided:



Note:

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

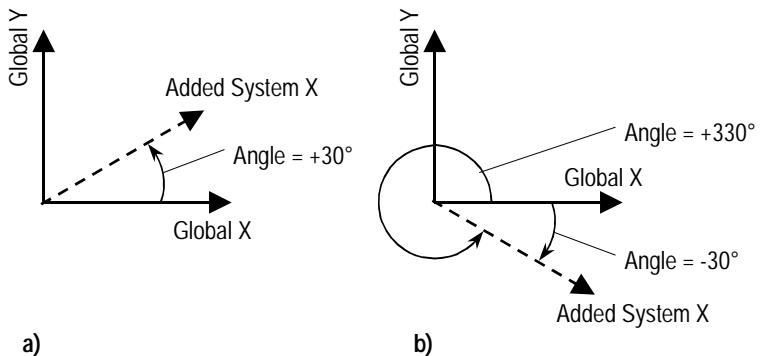
- **Grid ID:** This is an identifier for the grid line. It can be left blank if desired.
- **Coordinate/Spacing:** This is the location of the grid line in the specified coordinate system. Grid line locations can be specified by their coordinate or by their spacing.
- **Primary/Secondary Grid Lines:** 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.

Note: After changing the color of a grid line using this color box if you subsequently change the default color of grid lines using the **Options menu > Colors** command all grid lines will be set to the new default color *except* for those that you have changed using this color box.

- **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 then 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 then 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 dialog box 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

Figure 9-4:
Example of rotation angle used to specify orientation of added coordinate/grid systems relative to the global coordinate/grid system



9

Locate System Origin dialog box 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 9-4 shows some examples. The figure shows the global X-axis and the orientation of the X-axis for the added system. Figure 9-4a shows an example where the rotation angle is specified as 30 degrees. Figure 9-4b shows an example where the rotation angle could be specified as either 330 degrees or -30 degrees.

Editing Story Data

You can use the **Edit menu > Edit Story Data** command to edit story information and to insert new story levels or delete existing story levels. When you execute the **Edit menu > Edit Story Data** command a submenu appears with three options. They are edit, insert story and delete story.

Clicking the **Edit menu > Edit Story Data > Edit** command brings up the Story Data dialog box. The items in this dialog box are described in the sections titled "Editing Story Level Data" and "Similar Story Levels" in Chapter 22.

Cautionary Note: Once you change something in the story data and close the dialog box by clicking the **OK** button you can not undo the change.

Inserting a New Story Level

Clicking the **Edit menu > Edit Story Data > Insert Story** command brings up the Insert New Story dialog box. You specify the following items in this dialog box:

- **Story ID:** This is the name of the new story level.
- **Story Height:** This is the interstory height (not elevation above the base level) of the new story level.
- **Insert Above Level:** This is the story level that the new story level is to be inserted above. It can be any story level that currently exists in the model.
- **Replicate New Story:** You can specify an existing story level that the new story level is to be replicated from. If you specify an existing story level instead of None then all of the framing and all of the assignments in the specified existing story level will be copied to the new story level.

Note that specifying an existing story level (say Story Level X) for this option instead of None is equivalent to the following:

- ✓ Create the new story level (call it Story Level Y) using the None option in the Replicate New Story area of the Insert New Story dialog box.
- ✓ Select all of the objects on Story Level X and click the **Edit menu > Replicate** command to open the Replicate dialog box.
- ✓ Select the Story tab and highlight Story Level Y in the Replicate on Stories area of the dialog box.
- ✓ Click the **OK** button.

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

Deleting a Story Level

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

Reference Planes and Reference Lines

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.

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:

- If you draw vertical line objects or vertical area objects (columns or walls) in plan on a reference plane level then ETABS 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 ETABS does *not* break the vertical object at the reference plane. A single object is drawn from the story level to the story level below.



Note:

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

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

Merging Points

Under the **Options menu > Preferences > Dimensions/Tolerances** command one of the options is the Auto Merge Tolerance. When points are created (drawn, moved, copied, replicated, etc.) such that they fall closer together than the Auto Merge Tolerance the new point is automatically merged into the existing point by ETABS.

Any points in the model may be merged at any time using the **Edit menu > Merge Points** command. To use this command you first select the points that you want to merge. Then you execute the command and specify a merge tolerance. The merge then takes place. ETABS uses the following logic to merge the points:

1. ETABS orders the selected points based first on the number of grid lines that pass through them and based second on the order in which they were drawn.
2. ETABS 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. They are:

- Suppose one point falls exactly at a story level (call it Story Level X) and a second point falls a very short distance above the first point. Thus the second point is associated with Story Level (X+1). In this 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, then those two points will *never* merge. This is true regardless of the specified merge tolerance.

Aligning Points, Lines and Edges

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

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 then 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 then 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

this case since the column-type line object is no longer vertical, ETABS automatically changes it to a brace-type line object.



Tip:

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

- 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 this 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, then ETABS deletes them after the line object is moved.
- When ETABS 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 this 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 then ETABS deletes them after the edge of the area object is moved.

Coordinate System

The coordinate system specified in the **Edit menu > Align Points/Lines/Edges** command indicates which coordinate/grid system is to be considered for the following align options:

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

Align Options

There are eight separate align options available for the **Edit menu > Align Points/Lines/Edges** command in ETABS. They are:

- Align to X-coordinate of.
- Align to Y-coordinate of.
- Align to Z-coordinate of.
- Align to X grid lines.
- Align to Y grid lines.
- Trim selected lines at.
- Extend selected lines to.
- Align selected points to.

These options are discussed in the subsections below.

Align to X, Y or Z-Coordinate

For these options you 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 suppose that you choose to align to an X-coordinate of 4 and suppose that your maximum move allowed is 0.2. Then:

- 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 suppose that you choose to align to an X grid line at a coordinate of 4 and suppose that your maximum move allowed is 0.2. Then:

- 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 ETABS to insure that a beam is connected to a girder the end of the beam should fall 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.

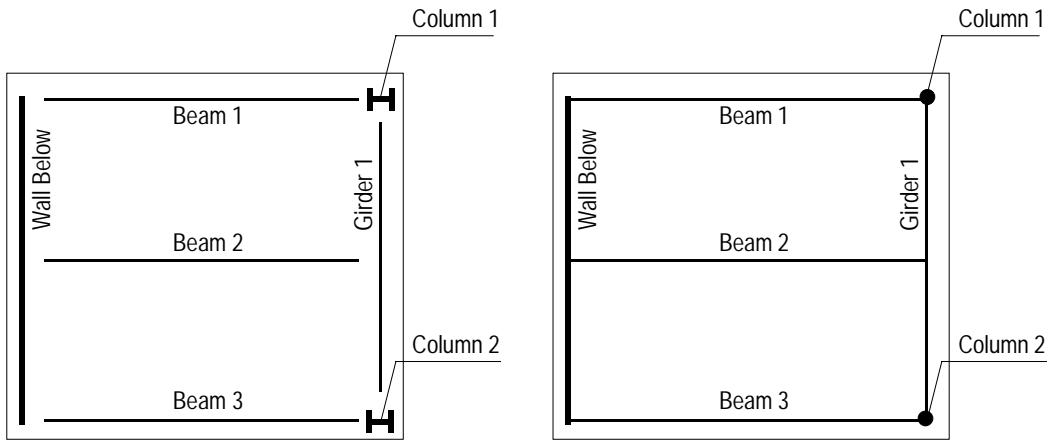
For these options you specify what you want the selected lines to be trimmed at. Then if the end of the selected line object is within the maximum move allowed of the specified trim (extend) item, the line option is trimmed (extended) to that item.

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 fall 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 then that girder can be extended to the line object representing the column even though the girder falls in a horizontal plane and the column falls in a vertical plane.

Consider the example shown in Figure 9-5. Figure 9-5a 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.



a) Floor Plan as Drawn in Building Plans

b) Floor Plan in ETABS Model

(Above)

Figure 9-5:
Example of extending selected lines

You could select Beams 1, 2 and 3 and Girder 1 and execute the **Edit menu > Align Points/Lines/Edges** command using the "Extend selected lines to" align option and specifying that the lines are to be extended to "Line or Edge" to obtain the model shown in Figure 9-5b 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" then only 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. No other line extensions are done in this 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" then all of the extensions shown in Figure 9-5b are done except that the left side of Beams 1, 2 and 3 are not 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 fall within the maximum allowed move distance to the end of the line then the trim (extend) is done to the item that is closer to the end of the line.

For example, suppose that in Figure 9-5 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" then the right end of Beam 2 is 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 Selected Points

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.

9

When checking for the closest specified item to a selected point ETABS 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 is a distance that is specified in the current units. If the selected object is within the align tolerance distance of whatever it is specified to be aligned with, then the selected object is moved. If it is not within the align tolerance distance then 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 then 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 since 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 then it is possible to have just one of the point objects move (align).

Moving Point, Line and Area Objects



Tip:

When you move objects in the Z direction they can not cross a story level.

You can use the **Edit menu > Move Points/Lines/Areas** command to move selected objects in any direction. You can specify distances in the global X, Y and Z directions that the object is to be moved. One restriction on the movement is that when you move objects in the Z direction they can not cross a story level. See the subsection below titled "Moving Objects in the Z Direction" for more information.

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 then the column will become sloped. (Note that ETABS 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 then ETABS deletes them after the line object is 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 these point objects then ETABS deletes them after the area object is 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 midheight 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 the -5 feet then ETABS moves the point to the second story level elevation. If you specify a Z coordinate greater than +5 feet then ETABS moves the point to the third story level elevation. 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.

Expanding and Shrinking Areas

Note:

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

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

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 then the command is ignored and the object is not shrunk. For example suppose you have a rectangular area object whose dimensions are 40 inches by 60 inches. If you specify an offset value of -20 inches or less (e.g., -25 inches) then 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 then the two sides would have to cross (overlap) each other. This is not allowed.

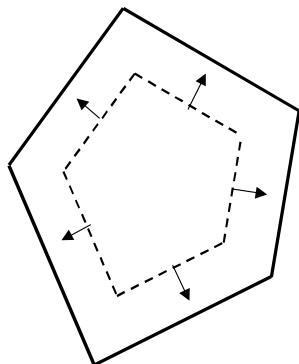
Useful Feature: Typically when you select an area object and use the **Edit menu > Expand/Shrink Areas** command all sides of the area object are moved. It is possible to select one or more sides of an area object individually and have the **Edit menu > Expand/Shrink Areas** command only apply to the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

- Press and release the E key on your keyboard to enter the edge select mode. In this mode you can click on the edge of an area object and that edge is selected. You are not able to select entire area objects by clicking inside them in this 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 are 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 do this.

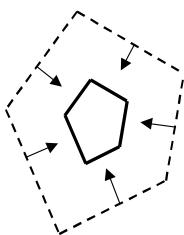


Tip:

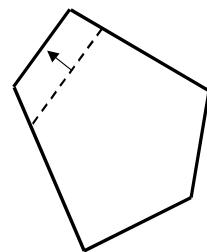
You can expand or shrink selected edges of area objects.



a) Expanded Area Object



b) Shrunk Area Object



c) Area Object with One Side Expanded

(Above)

Figure 9-6:
Example of expanded and shrunk area objects

Figure 9-6 shows some examples of expanded and shrunk area objects. In the figure the dashed line represents the original area object and the solid line represents the final area object after it is expanded or shrunk. Note that in Figure 9-6c since only one edge of the area object is moved the solid line lies on top of the dashed line for all other edges.

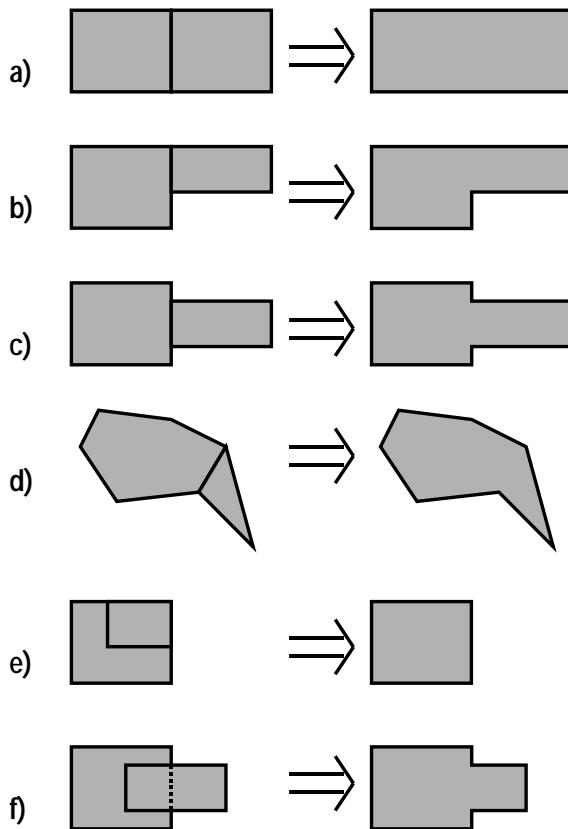
Merging Areas

You can select two area objects that have a common edge or overlap and then use the **Edit menu > Merge Areas** command to merge the area object into one area object. You can not 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 then the property and assignments come from the first drawn area object. Since you may not remember which area object was drawn first you should carefully check the assignments to the new combined area object in this case.

Figure 9-7 shows some examples of merged area objects.

Figure 9-7:
Examples of merged
area objects



Joining Lines

You can select two or more collinear line objects with common end points and the same type of property (frame section, link or none) and then use the **Edit menu > Join Lines** command 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.

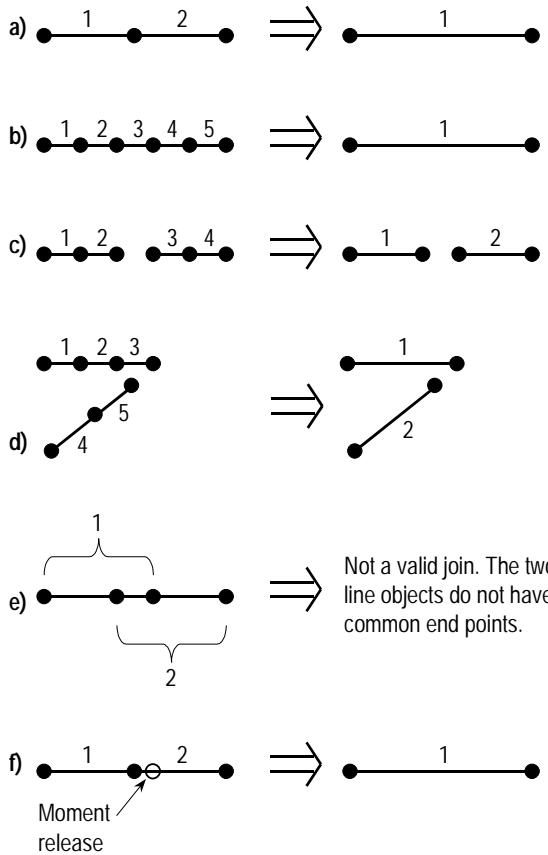
- When line objects with frame section properties are joined the section property assigned to the combined line object is the one that had the largest area. If two of the combined frame sections have the same area then the property of the first drawn object is used.
- Load and mass assignments from the unjoined line objects are combined on the joined object.
- Assignments to the unjoined line objects that would be illegal in 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 9-8 shows some examples. Item a in the figure shows that two collinear line objects with a common end point (and the same property type assignment) are joined into one line object. Item b shows that five collinear line objects can be joined at the same time. Items c and d 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.

Item e in Figure 9-8 illustrates that the collinear line objects must have a common end point otherwise they are not joined. If you want to join the example shown in Figure 9-8e you should move one of the center joints so that it is coincident with the other center joint and then perform the join.

Figure 9-8f illustrates that assignments to the unjoined line objects that would be illegal in middle of the joined line object are ignored. In this case a moment release that is in the center of the combined beam is ignored. If you want this moment release to remain then you should not join the line objects.

Figure 9-8:
Examples of joined
line objects



Dividing Lines

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

- **Divide into 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.
- **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 9-9 shows some examples.

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

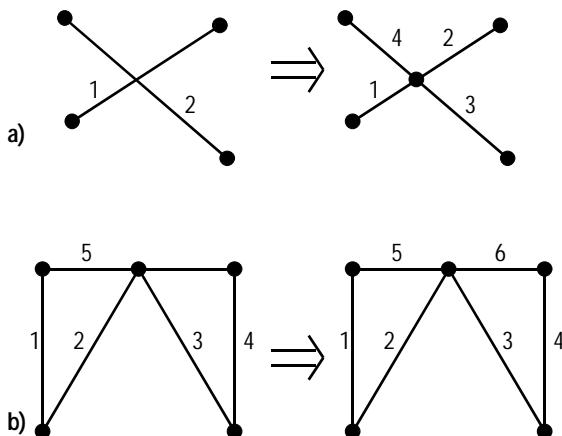


Figure 9-9a shows two crossing line objects. Initially the line objects are not connected at their intersection. When the two line objects are selected and the Break at Intersections with Selected Lines and Points option is used to divide the lines, each of the lines is broken into two objects at the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

Figure 9-9b 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 you could 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 you could select the point at the top of the braces and the top beam (line object labeled 5) to achieve the same result.

In either case shown in Figure 9-9 ETABS would provide connectivity at the intersection points regardless of whether or not the line objects are divided unless the intersecting line objects are indicated not to be meshed using the **Assign menu > Frame/Line > Automatic Frame Mesh/No Mesh > Don't Mesh It** command. In most cases manual breaking up of intersecting line objects as shown in Figure 9-9 is not necessary unless different properties are to be assigned or a different label is required. The example in Figure 9-9 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.

Reshaper Tool

You can activate the reshaper tool by either clicking on **the Reshaper** button, , on the side toolbar or by clicking **Draw menu > Reshape Object**. When you activate the reshaper tool you enter reshape mode. Once you are in reshape mode you remain in that mode until you do one of the following:

- Click the Pointer button, , on the side toolbar.
- Press the Esc key on your keyboard.
- Choose one of the drawing options from the Draw menu or the side toolbar.
- Click on one of the select items in the Select menu.
- Run an analysis.

**Tip:**

When you reshape objects you can not move them in the Z direction such that they cross a story level.

9

Once you are in reshape mode you can 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 below.

Whether you are reshaping area, line or point objects be sure to read the subsection below titled "Moving/Reshaping Objects in the Z Direction."

Note that the drawing constraints discussed in the subsection titled "Drawing Constraints in ETABS" in Chapter 12 are available when you use the reshapertool.

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.

**Note:**

The drawing constraints discussed in the subsection titled "Drawing Constraints in ETABS" in Chapter 12 are available when you use the reshapertool.

- Right click on one of the corner points of the area object. This brings up a dialog box 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, not any surrounding elements that it may be connected to.

See the subsection below titled " Moving/Reshaping Objects in the Z Direction."

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, not any 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, not any surrounding elements that it may be connected to.

- Right click on one of the end points of the line object. This brings up a dialog box 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, not any surrounding elements that it may be connected to.

See the subsection below titled " Moving/Reshaping Objects in the Z Direction."

Reshaping Dimension Lines

Dimension lines are discussed in the subsection titled "Dimension Lines" in Chapter 12. You can use the reshaper tool to move a dimension line to a new location. You can not lengthen or shorten a dimension line, even using the reshaper tool.

To relocate the 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 disconnect from the objects it is attached to when it is reshaped/moved.

- Right click on the point object. This brings up a dialog box 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.

See the subsection below titled " 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 can not 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 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 relocating an area object corner point that occurs at the midheight 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 then ETABS moves the point to the second story level elevation. If you specify a Z coordinate greater than the third story level elevation then ETABS moves the point to the third story level elevation. If you specify a Z coordinate between the second and third story elevations, inclusive, then ETABS moves the point to the specified elevation.

The ETABS Nudge Feature

ETABS includes a nudge feature that allows you to modify the geometry of your model in a plan view. To use the nudge feature you simply select the item(s) that you want to nudge and then 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) are nudged (moved) when you press the Ctrl and arrow keys is specified in the ETABS Dimension/Tolerance Preferences. You can see this item by clicking Option menu > Preferences > Dimensions/Tolerances. The name of the item that controls the movement is Plan Nudge Value.
- You can not 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



Tip:

You can nudge dimension lines.

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 re-oriented 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.



The ETABS View Menu

General

The View menu in ETABS provides basic options and tools for viewing your ETABS model. This chapter discusses those options and tools.

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 discussed in Chapter 16.

Types of Views

The following types of views are available in ETABS:

- Three-dimensional (3D)
- Plan

- Elevation
- Perspective
- Custom

Each of these types of views are briefly discussed in separate subsections below.

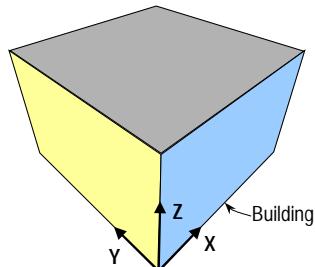
Three Dimensional (3D) Views

You can set a window to a three-dimensional (3D) view either by using the **View menu > Set 3D View** command or by clicking the **3D View** button, , on the main (top) toolbar. If you use the menu command the Set 3D View dialog box is opened where you can define the view direction by specifying a plan angle, elevation angle and an aperture angle. All angles are specified in degrees. The view direction defines the location where you are standing as you view the building from the outside.

Figure 10-1a 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 10-1b, c and d illustrate how the plan, elevation and aperture angles are defined. Following are explanations of the terms used in Figure 10-1.

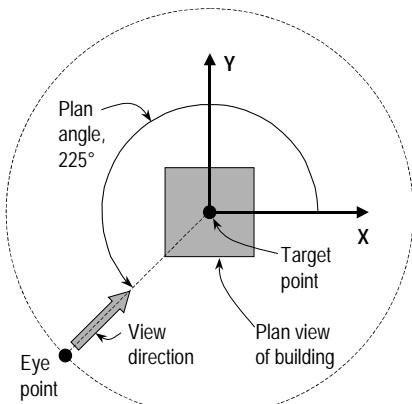
- **Eye point:** This is the location from which you are viewing the building.
- **Target point:** This is the geometric center of the building.
- **View direction:** This is defined by a line drawn from the eye point to the target point.
- **Plan angle:** This is the angle (in degrees) from the positive 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.

Figure 10-1:
Illustration of plan,
elevation and aperture
angles used to
define a 3D view

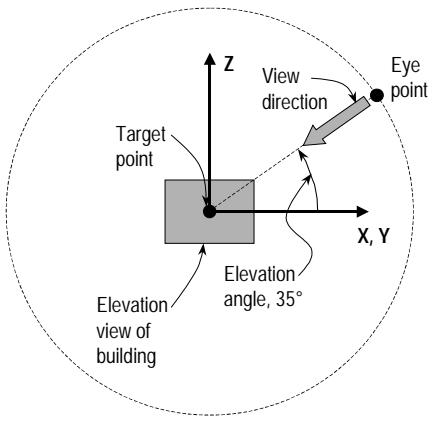


Plan angle = 225°
Elevation angle = 35°
Aperture angle = 60°

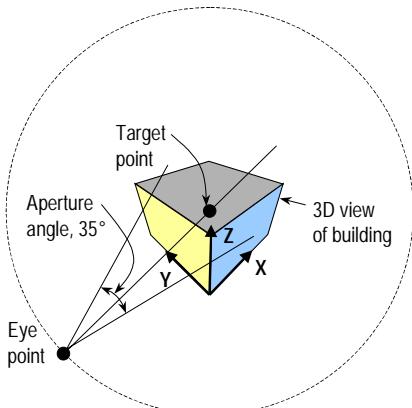
a) Default 3D View



b) Plan Angle



c) Elevation Angle



d) Aperture Angle

**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.

10

- **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.
- **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:
 - ✓ ETABS constructs the view direction line from the eye point to the target point.
 - ✓ ETABS constructs a 3D bounding rectangular box that just encloses the model.
 - ✓ ETABS passes a plane through the target point and perpendicular to the view direction line.
 - ✓ ETABS projects the eight corner points of the bounding box onto the plane. Call these the projected corner points.
 - ✓ ETABS constructs lines from the eye point to the projected corner points of the bounding box. Eight such lines are constructed since there are eight corner points. Call these eight lines corner point lines.
 - ✓ ETABS 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 ETABS does not allow the eye point to be located inside the structure.

The 3D View dialog box 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 10-1. The other fast views give you 3D perspective views of the specified planes.

Clicking the **3D View** button, , on the toolbar gives you the default 3D view with the plan, elevation and aperture angle as specified in Figure 10-1.

When the active window is showing a 3D view you can click the **Rotate 3D View** button, , on the main toolbar and use the mouse to adjust the view direction. Once you have clicked the button simply left click the mouse in the window with the 3D view and while holding down the mouse left button drag the mouse to adjust the view direction. Note that as soon as you left click the mouse in the window with the 3D view a bounding box (dashed lines enclosing the model) appears and that 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 below titled "Perspective Views" for additional information on three-dimensional views.

Plan Views

Note:

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



You can set a window to a plan view either by using the **View menu > Set Plan View** command or by clicking the **Plan View** button, , on the main (top) toolbar. When you click the menu item or the toolbar button the Select Plan Level dialog box appears. From this dialog box you can select the story level or reference plane for which you want to show a plan view. Refer to the section titled "Reference Planes and Reference Lines" in Chapter 9 for additional information on reference planes.

When the active window is a plan view you can use the **Move Up in List** button, , and the **Move Down in List** button, , to quickly change to other plan views. The list referred to is

the one you see in the Select Plan Level dialog box when you initially set the plan view.

In plan view the following objects are visible assuming, of course, that they are specified as visible in the Set Building View Options dialog box that is accessed through the **View menu > 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 either have an end point in the considered plan view or pass through the considered plan view. Note that column-type line objects can not pass through plan views at story levels. They can only pass through plan views at reference plane levels.
- All wall-type area objects that either have corner points in the considered plan view or pass through the considered plan view. Note that wall-type area objects can not 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 below titled "Perspective Views" for additional information on plan views. See Chapter 13 for special selection rules when windowing in plan view.

Elevation Views

By default ETABS defines elevation views along each of the defined primary grid lines in your model. You can use the **View menu > Set Elevation View** command or by clicking the **Elevation View** button, **elev**, on the main (top) toolbar to display the Set Elevation View dialog box which allows you to add addi-

tional elevation views, modify or delete existing elevation views and indicate which elevation view you want to display.

Developed elevation views can be displayed but not defined from the Set Elevation View dialog box. Refer to the section titled "Developed Elevations" in Chapter 12 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 dialog box includes a list of the defined elevation names (including the names of defined developed elevations, if any) and several command buttons. To indicate the elevation view that you want to display simply highlight the appropriate elevation name in the Set Elevation View dialog box and click the **OK** button. To delete a defined elevation view highlight the elevation in the Set Elevation View dialog box and click the **Delete Elevation Name** button.

To define a new elevation view click the **Add New Elevation** button in the Set Elevation View dialog box. (Note: The **Add New Elevation** button is not used to define developed elevations. Use the **Draw menu > Draw Special Items > Draw Developed Elevation Definition** command to define a developed elevation.) To modify an existing elevation view highlight the elevation in the Set Elevation View dialog box and click the **Modify>Show Elevation** button. Note that you can not modify the default elevations along the grid lines that are created by ETABS, however, you can delete them. Also you can not 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 dialog box. Here you 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 then 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 then the elevation is a view of the XZ plane in the specified coordinate system at the specified Y ordinate.

When the active window is an elevation view you can use the **Move Up in List** button,  , and the **Move Down in List** but-

ton, , to quickly change to other elevation views. The list referred to is the one you see in the Set Elevation View dialog box when you initially set the elevation view.

Refer to the subsection below titled "Perspective Views" for additional information on elevation views.

Perspective Views

The **Perspective Toggle** button, , located on the main (top) toolbar is a useful tool. It has slightly different behavior depending on whether it is used in a plan, elevation or three-dimensional view.

Perspective Toggle in a Plan View

When you click the **Perspective Toggle** button in a plan view the view switches to a 3D perspective view. If the plan view is of a story level then 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 then the perspective view only shows the objects that are associated with the story level that the reference plane is associated with.

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 the **Rotate 3D View** button, , on the main toolbar is active and you can use it to adjust the view direction.

As the name indicates, the **Perspective Toggle** button is a toggle switch. Thus you can use it once in a plan view to switch to a perspective view of a story level. If you later click the **Perspective Toggle** button again in this same view it will switch you back to your original plan view.

Perspective Toggle in an Elevation View

When you click the **Perspective Toggle** button in an elevation view the view switches 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 ele-

vation angle set to 0 degrees and the aperture angle set to 60 degrees. If the elevation displayed is a developed elevation then the initial direction of the 3D perspective view is looking directly at the first segment of the developed elevation.

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

As the name indicates, the **Perspective Toggle** button is a toggle switch. Thus you can use it once in an elevation view to switch to a perspective view of structure. If you later click the **Perspective Toggle** button again in this same view it will switch you back to your original elevation view.

Perspective Toggle in a Three-Dimensional View

When you click the **Perspective Toggle** button in a three dimensional window the view switched from a perspective view to an isometric view. In other words, the aperture angle is toggled to 0 degrees.

As the name indicates, the **Perspective Toggle** button is a toggle switch. Thus you can use it once in a 3D view to switch to an isometric view of structure. If you later click the **Perspective Toggle** button again in this same view it will switch you back to your original perspective view.

Custom Views

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/or selected objects only displayed, and/or a special zoom set and/or 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.

Viewing Tools Available in ETABS

The following viewing tools are available in ETABS:

- Specify view limits
- Show selection only and show all
- Zoom features
- Pan feature
- Refresh views and windows
- Change axes locations
- Make measurements in your ETABS model

Each of these viewing tools are briefly discussed in separate subsections below.

View Limits

You can use the **View menu > Set Building View Limits** command to display the Set Building View Limits dialog box and set the limits for a view. When you set the view limits only objects that fall entirely inside the view limits are displayed. The view limits only affect objects; they have no effect on the coordinate/grid systems which still show in their entirety.

The Set Limits dialog box 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.

There are two different methods available for specifying the X and Y-axis limits. Method one is to type in minimum and maximum X and Y coordinates. Method two is to graphically set the limits in the small plan view located in the Plan Limits area of the dialog box.

In the Plan Limits area there is a dashed box with selection handles on the four sides superimposed over a plan view of the structure. The dashed box defines the view limits. You can 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.

Show Selection Only and Show All

Sometimes you may find that there are too many objects in a view for you to clearly see whatever it may be that you want to see. In such cases you may want to select just a few of the objects in the view and then 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.

Zoom Features

There are five zoom features available in ETABS. These features allow you to zoom in or out on a view. Zooming in shows you a closer view of the model and zooming out shows you a farther away view of the model. All five zoom features are available both on the View menu and on the main (top) toolbar. The zoom features and their associated toolbar buttons are:

- **Rubber Band Zoom,**  : This command allows you to zoom in on the model by windowing. To use the command you 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 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 this case the **View menu > Restore Full View** command returns you to the view you were in right before you executed the **View menu > Pan** command.

When you use both zoom commands and pan commands together ETABS has one of the following behaviors depending on the order 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 were in right 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 then the effect is to toggle between two zoom settings. You can not 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 dialog box that you reach using the **Options menu > Preferences > Dimensions/Tolerances** command.

The ETABS default value for the Auto Zoom Step is 10 percent. What this means is that when you use the **View menu > Zoom In One Step** command ETABS 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 dialog box that you reach using the **Options menu > Preferences > Dimensions/Tolerances** command.

The ETABS default value for the Auto Zoom Step is 10 percent. What this means is that when you use the **View menu > Zoom Out One Step** command ETABS 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 titled "Dimensions and Tolerances" under the

section titled "Preferences" in Chapter 18 for more information on the Pan Margin item.

Click the **View menu** > **Pan** command or the **Pan** button, , located on the main (top) toolbar to pan a view. Once you have clicked the menu command or toolbar 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 re-click the menu command or toolbar button every time you pan. If you have panned a view one or more times then clicking the **View menu** > **Restore Full View** command, or its associated toolbar button on the main (top) toolbar returns you to the view you were in right before you executed the first **View menu** > **Pan** command.

10 Refresh Views and Windows

The **View menu** > **Refresh Window** command and the corresponding **Refresh Window** button, , on the main (top) toolbar 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.

Change Axes Location

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 dialog box 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**, , on the main (top) toolbar.

Make Measurements in your ETABS Model

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.



Tip:

*You can use the **Draw menu > Draw Dimension Line** command to draw dimension lines that include dimension text (measurements).*

- **Lines:** When you execute the **View menu > Measure > Line** command you simply left click on two points to define the line you want to measure and ETABS reports the length of the line in the status bar at the bottom of the ETABS window.
- **Areas:** When you execute the **View menu > Measure > Area** command you simply left click on the corner points of an area you want to measure and ETABS reports the area and perimeter of the area in the status bar at the bottom of the ETABS window.

When defining the last point for the area you should either double left click or single left click and then press the Enter key (or Esc key) on your keyboard.

- **Angles:** When you execute the **View menu > Measure > Angle** command you simply left click on three points to define two lines that have one common endpoint. ETABS reports the angle between these lines in the status bar at the bottom of the ETABS window. The angle is always reported in degrees and it is always less than or equal to 180 degrees.

10

Note the following about using the **View menu > Measure** command.

- After you have drawn the line, area or angle and reviewed the measurement you can click anywhere and the drawn line, area or angle is deleted.
- The measurements are always reported in the current units.

Building View Options

You can control the building view options through the Set Building View Options dialog box. You can access this dialog box by clicking **View menu > Set Building View Options** or by clicking the **Set Building View Options** button, , on the main (top) toolbar. *Note that the settings you make in this dialog box only affect the currently active window.*

Note:

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

The Set Building View Options dialog box allows you to specify viewing options in the following categories:

- View by colors
- Special effects
- Object visibility
- Object viewing options

- Piers and spandrels
- Other visibility options
- Special frame items
- Other special items.

Each of these building display option categories are briefly discussed in separate subsections below.

View by Colors

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 dialog box 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.

You also have the option of displaying the model in black and white where all objects and text are displayed in black and the background is white. 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 dialog box 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 18 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 dialog box 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 dialog box 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 dialog box.



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 Visibility

The Object Visibility area of the Set Building View Options dialog box 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 can not select the object. Contrast this with the information in the second paragraph describing the Object Edge feature in the subsection above titled "Special Effects."

**Note:**

If an object does not have its corresponding check box checked in the Object Visibility area of the Set Building View Options dialog box, then you can not see or select the object.

10

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 para-

graphs 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 are 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:
 - ✓ Point Objects are visible when the Point Objects check box is checked. They are not visible when the box is unchecked.
 - ✓ When point objects are visible you can select them by clicking on them or by windowing them.
 - ✓ You only see a dot representing a point object when the point objects are visible and the Object Shrink feature is on. The Object Shrink feature is controlled either in the Special Effects area of the Set Building

View Options dialog box or using the **Object Shrink Toggle** button on the main (top) toolbar.

- ✓ When point objects are not visible (the Point Objects check box is unchecked) you can not select them and you will never see dots representing them.
- ✓ 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.

Object View Options

The Object View Options area of the Set Building View Options dialog box 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 dialog box:

- **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.

**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.

- **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.

Note that no control is provided for the point local axes. This is done because in ETABS, by default, 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 dialog box 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 dialog box:

- **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.

Other Visibility Options

The Other Visibility Options area of the Set Building View Options dialog box allows you to toggle the display of other miscellaneous items on and off. Following is a list of the specific items you control in this area of the dialog box:

- **Story Labels:** This item toggles the labels (names) for story levels on and off in elevation views. Note that ETABS 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 "Reference Planes and Reference Lines" in Chapter 9 for additional information.

- **Reference Planes:** This item toggles the display of reference planes on and off. See the section titled "Reference Planes and Reference Lines" in Chapter 9 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 dialog box must be checked for the supports to be visible.* Four basic graphic symbols used for displaying supports in the ETABS graphical interface are:

Roller: 

Fixed: 

Pinned: 

Other: 

- **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.

Special Frame Items

The Special Frame Items area of the Set Building View Options dialog box 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 Object Visibility area of the dialog box then 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 dialog box:

- 10
- **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 which 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) then 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.
- **End 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 > Frame/Line > Frame Rigid Offsets** command. This command brings up the Assign Frame End Offsets dialog box. In this dialog box you can specify both an end offset along the length of the beam and frame joint offsets. This item only controls the display of frames with end offsets along the length of the beam assigned to them. It does not control the display of frames with joint offsets assigned to them. This is controlled by the Joint Offsets item below.

- **Joint Offsets:** This item toggles the display of text saying "OFF*" without the quotes for any line object with frame section 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 > Frame Rigid Offsets** command. This command brings up the Assign Frame End Offsets dialog box. In this dialog box you can specify both frame joint offsets and end offsets along the length of the beam. This item only controls the display of frames with frame joint offsets assigned to them. It does not control the display of frames with end offsets along the length of the beam assigned to them. This is controlled by the End Offsets item above.
- **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 then the value represents the minimum number of output stations. If it is not reported in parenthesis then 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 dialog box alert you that a particular type of assignment is made to a line object with frame section properties but it 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 dialog box 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 dialog box:

- **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 done by ETABS. See Chapter 30 for information on automatic meshing. 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 dialog box 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.



The ETABS Define Menu

General

The main purpose of the ETABS Define menu is to provide a means of defining section properties and load case definitions. This chapter discusses the features available on the Define menu.

Items related to ETABS nonlinear analysis are mentioned in passing but are not elaborated on; they are beyond the scope of this manual. Separate documentation is provided for these items in technical notes on our web site.

Material Properties

The material properties in ETABS are always linear elastic. Use the **Define menu > Material Properties** command to define material properties. This command brings up the Define Materials dialog box where the names of all defined material properties are listed. In this dialog box you can do the following:

- Click the **Add New Material** button to display the Material Property Data dialog box 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 dialog box 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 can not 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 can not delete a material property if it is currently in use.

The Material Property Data dialog box consists of six different areas. They are:

- **Material name:** Here you can specify or modify the name of a material property. Note that you can not change the name of the built-in STEEL and CONC material properties.
- **Type of material:** You specify the material to either be isotropic or orthotropic. The option chosen here affects what is shown in the Analysis Property Data area of the dialog box.

Note:

You can specify isotropic or orthotropic material properties in ETABS.

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.

The isotropic mechanical and thermal properties relate strain to stress and temperature change as shown in Figure 11-1a. In the figure **e1** is Young's modulus of elas-

$$\begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{bmatrix} = \begin{bmatrix} \frac{1}{e1} & -\frac{u12}{e1} & -\frac{u12}{e1} & 0 & 0 & 0 \\ \frac{1}{e1} & \frac{-u12}{e1} & \frac{-u12}{e1} & 0 & 0 & 0 \\ & \frac{1}{e1} & 0 & 0 & 0 & 0 \\ & & \frac{1}{g12} & 0 & 0 & 0 \\ & & & \frac{1}{g12} & 0 & 0 \\ & & & & \frac{1}{g12} & 0 \end{bmatrix} \text{Symmetrical}$$

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{bmatrix} + \begin{bmatrix} a1 \\ a1 \\ a1 \\ 0 \\ 0 \\ 0 \end{bmatrix} \Delta T$$

$$\begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{bmatrix} = \begin{bmatrix} \frac{1}{e1} & -\frac{u12}{e2} & -\frac{u13}{e3} & 0 & 0 & 0 \\ \frac{1}{e2} & \frac{-u23}{e3} & 0 & 0 & 0 & 0 \\ & \frac{1}{e3} & 0 & 0 & 0 & 0 \\ & & \frac{1}{g12} & 0 & 0 & 0 \\ & & & \frac{1}{g13} & 0 & 0 \\ & & & & \frac{1}{g23} & 0 \end{bmatrix} \text{Symmetrical}$$

a) Isotropic Material

b) Orthotropic Material

(Above)

Figure 11-1:
Illustration of how mechanical and thermal properties relate strain to stress and temperature change for isotropic and orthotropic materials

11

ticity, $u12$ 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 ETABS derives it from the specified Young's modulus and Poisson's ratio as shown in Equation 11-1.

$$g12 = \frac{e1}{2(1+u12)} \quad \text{Eqn. 11-1}$$

Note that in ETABS 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 11-1b. 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 ETABS 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:** In this area you specify 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 you 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 11-1.

For orthotropic materials you define three values (one for each local axis direction) for the modulus of elasticity, Poisson's ratio, coefficient of thermal expansion and shear modulus.

- **Display Color:** Here you assign a color to the material property. If you use the **View menu > Set Building View Options** command to display the Set Building View Options dialog box, then you can then choose an option to view the model based on the colors associated with the material properties. In this case each object appears in a color associated with its assigned material property. See the section titled "Building View Options" in Chapter 10 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 dialog box.

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 dialog box. In general the design property data specified in the material property 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, ETABS calculates the hinge force-deformation properties based on these properties.

If the type of design is **Steel** then 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 where 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.

**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.

11

If the type of design is **Concrete** then the following items are specified:

- ✓ **Specified concrete compressive strength, f'_c :** This item is used in all calculations.
- ✓ **Bending reinf. yield stress, f_y :** This is the reinforcing steel yield stress used in the calculations for bending and axial load calculations.
- ✓ **Shear reinf. yield stress, f_{ys} :** This is the reinforcing steel yield stress used in the calculations for shear calculations.
- ✓ **Lightweight concrete check box:** 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** then nothing is specified in the Design Property Data area.

Frame Section Properties

You can use the **Define menu > Frame Sections** command to define frame section properties. This command brings up the Define Frame Properties dialog box. The Properties area of this dialog box lists the names of all the currently defined frame section properties. The Click To area of the dialog box 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

The drop-down box that initially says Import I/Wide Flange in the Click To area of the Frame Properties dialog box allows you to import many different types of frame section properties from one of several section databases that are included with ETABS. The types of section properties you can import are:

- I-shaped members including wide flange sections
- Channels
- Double channels
- Structural tees
- Single angles
- Double angles
- Structural tubes
- Pipe sections
- Rectangular sections
- Circular sections
- General sections

11

Note:

Several steel section databases are included with ETABS. You can also create your own section databases.

Note that the last three items in the above list are not available in the section databases provided with ETABS but could be in section databases that you create yourself. General sections have section properties (area, moment of inertia, etc.) associated with them but no dimensions specified for them. All other types of sections in the above list have dimensions specified.

The section databases that are included with ETABS are:

- **Aisc.pro:** American Institute of Steel Construction shapes.
- **Cisc.pro:** Canadian Institute of Steel Construction shapes.

- **Euro.pro:** European steel shapes.
- **Sections.pro:** American Institute of Steel Construction shapes.

In addition you can create your own section database files using the utility program called Proper.

The default section database that ETABS will open is called Sections.pro. When ETABS is shipped Sections.pro is the same as Aisc.pro. You can overwrite this version of Sections.pro with any other section database if desired.

11 To import a section from a database you click the **Define menu > Frame Sections** command, click on one of the section types in the Import drop-down box and specify a database file to choose the section from, if necessary.

ETABS 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 then continue to hold down the left 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.

**Shortcut:**

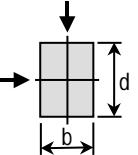
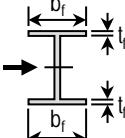
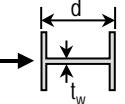
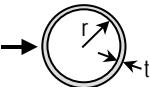
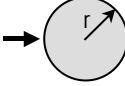
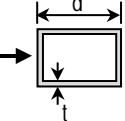
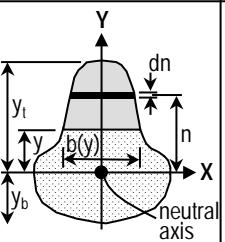
You can use the Assign menu > Frame/Line > Frame Section command to simultaneously define frame sections and assign them to selected line objects.

Adding User-Defined Frame Section Properties

The drop-down box that initially says Add I/Wide Flange in the Click To area of the Frame Properties dialog box allows you to easily define section properties for many different types of frame sections. The types of section properties you can define are:

- I-shaped members
- Channels
- Double channels
- Structural tees
- Single angles
- Double angles
- Structural tubes
- Pipe sections
- Rectangular sections
- Circular sections
- General sections
- Auto select section list
- Sections defined in the Section Designer utility
- Nonprismatic sections

For all but the last four items in the above list you simply specify dimensions for the section and ETABS automatically calculates the section properties. For general sections you simply specify the section properties (area, moment of inertia, shear area, etc.). No dimensions are input for general sections. Figure 11-2 is provided to help you determine the shear area for general sections of various shapes.

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	$\pi r t$
	Solid circular section: Shear forces from any direction	$0.9 \pi r^2$
	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 $Q(y) = \int_{y_b}^{y_t} n b(n) dn$	$\frac{I_X^2}{\int_{y_b}^{y_t} \frac{Q^2(y)}{b(y)} dy}$

(Above)

Figure 11-2:
Shear areas for various sections

**Tip:**

You can use the Section Designer utility to graphically define frame sections. Select the Add SD Section option in the Define Frame Properties dialog box.

Auto select section lists are simply lists of previously defined steel sections. These are useful for Steel Frame Design and Composite Beam Design where ETABS 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.

11

You can use the Section Designer utility to graphically define unusual sections. ETABS then calculates the section properties for that section. See the section titled "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 titled "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 titled "Reinforcing for Concrete Frame Section Properties" later in this chapter for more information.

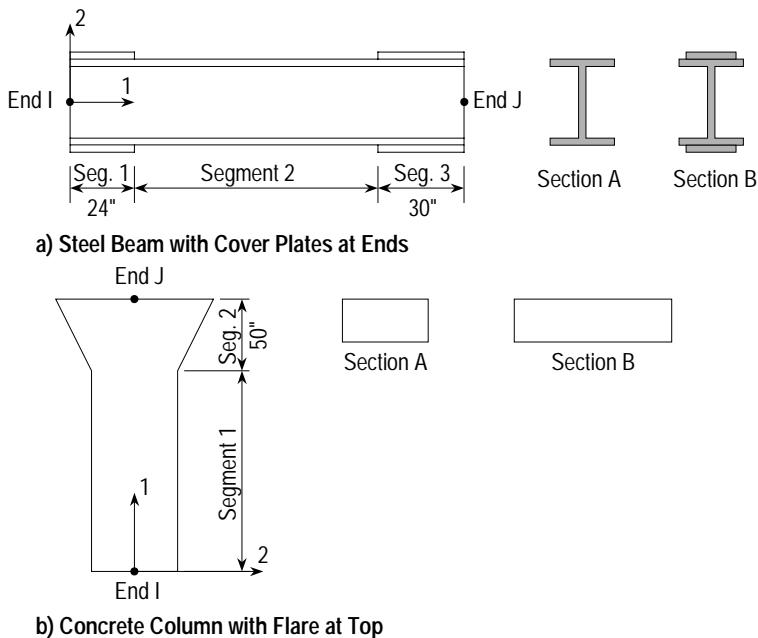
Adding Frame Section Properties using Section Designer

Select the Add SD Section option in the Define Frame Properties dialog box to define a frame section property using the Section Designer feature. This brings up the SD Section Data dialog box. The following areas are in this dialog box:

- **Section name:** Here you can specify or modify the name of the frame section.
- **Material:** Here you can specify or modify a defined material property. Depending on the design type of the material property selected various options in the Section Type area may or may not be available.
 - ✓ If the design type of the specified material property is Steel, then the General Steel Section and Other (not designed) options are available.

- ✓ If the design type of the specified material property is Concrete, then the Concrete Column and Other (not designed) options are available.
- ✓ If the design type of the specified material property is None, then only the Other (not designed) option is available.
- **Design Type:** Here you can specify the section type. If the section type is General Steel Section then any frame section assigned this property is designed by the Steel Frame Design postprocessor as a general section. If the section type is Concrete Column then any frame section assigned this property is designed by the Concrete Frame Design postprocessor. If the section type is No Check/Design (not designed) then any frame section assigned this property is not designed by any 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:** Once you have appropriately specified items in the rest of the dialog box, 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 dialog box. You can then click the **OK** button to complete the definition of the frame section property.

Figure 11-3:
Nonprismatic frame
section examples



Nonprismatic Sections

Note:

ETABS analyzes nonprismatic sections. The Steel

Frame Design postprocessor can design nonprismatic sections. The Concrete

Frame Design and Composite Beam Design postprocessors do not currently design non-prismatic 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; these do not need to be of equal length. Most common situations can be modeled using from one to five segments.

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 11-3 for examples of nonprismatic frame sections. Figure 11-3a 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.

(Below)**Table 11-1:**

Input for nonprismatic frame section example in Figure 11-3a

11

Once both section A and section B are 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 dialog box. Table 11-1 shows the assignments that would be entered in the dialog box. Note that the variation items grayed out in the table are not used by ETABS because the start section and the end section are the same.

In Table 11-1 note that segment 2 has a length type of variable and a segment length of 1. See the subsection below titled "Segment Lengths" for an explanation of this.

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		

(Below)**Table 11-2:**

Input for nonprismatic frame section example in Figure 11-3b

Figure 11-3b 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, selecting "Add Rectangular" from the Add drop-down box.

Once both section A and section B are 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 dialog box. Table 11-2 shows the assignments that would be entered in the dialog box. Note that the variation items grayed out in the table are not used by ETABS because the start section and the end section are the same.

Segment	Start Section	End Section	Length	Length Type	EI33 Variation	EI22 Variation
1	A	A	1	Variable		
2	A	B	50	Absolute	Cubic	Linear

Segment Lengths

The length of a nonprismatic segment may be specified as either a variable length or an absolute length.

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, L_c , is first calculated as the total length minus the end offsets: $L_c = L - (i\text{off} + j\text{off})$. In this equation L is the full length of the frame element and $i\text{off}$ and $j\text{off}$ 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, then 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.

Starting and Ending Sections

The properties for a segment of a nonprismatic section are defined by specifying:

- 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.

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.

Specifically, the linear, parabolic or cubic variation for EI_{33} is calculated by ETABS 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, then 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 dialog box and you specify the material property as one with a Design Type of Concrete, a **Reinforcement** button appears in the Frame Properties dialog box. Clicking the **Reinforcement** button brings up the Reinforcement Data dialog box.

There are two tabs in the Reinforcement Data dialog box. They are labeled column and beam. Pick the tab that corresponds to the frame section you are defining.

Reinforcing Information for Beams

For concrete beams there are two types of reinforcing information that you specify. They are rebar cover and reinforcement overrides. 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:

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.

**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.

11

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 ETABS 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) ETABS 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 ETABS 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 ETABS 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 ETABS 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.

Reinforcing Information for Columns

For columns the following areas are provided in the Reinforcement Data dialog box:

- **Configuration of Reinforcement:** Here you can specify rectangular or circular reinforcement. You can if desired put circular reinforcement in a rectangular beam or put rectangular reinforcement in a circular beam.
- **Lateral Reinforcement:** If you have specified a rectangular configuration of reinforcement then the only choice available to you here is ties. If you have specified a circular configuration of reinforcement then you have an option of either ties or spiral for the lateral (transverse) reinforcement.
- **Rectangular Reinforcement:** This area is visible if you have chosen a rectangular 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 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 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.



Note:

Cover is typically measured from the nearest edge of the concrete section to the centroid of the reinforcing steel.

- **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 that when a member with this frame section property is run through the Concrete Frame Design postprocessor the reinforcement is either to be checked or to be designed. If the reinforcement is to be checked then all information in the Reinforcement Data dialog box is used. If the reinforcement is to be *designed* then all information in the Reinforcement Data dialog box is used except the bar size is ignored and the total required steel area is calculated. For design the configuration of reinforcement, lateral reinforcement and cover is used.

If you specify reinforcing in a concrete column frame section property that is specified using the section designer utility, then 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.

Wall/Slab/Deck Section Properties



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.

You use the **Define menu > Wall/Slab/Deck Sections** command to define wall, slab or deck section properties. This command brings up the Define Wall/Slab/Deck Sections dialog box. The Sections area of this dialog box lists the names of all the currently defined wall, slab and deck section properties. The Click To area of the dialog box allows you to define new wall, slab and deck sections, modify existing wall, slab and deck section definitions and delete existing wall, slab and deck sections. Note that you can only delete wall, slab and deck sections if they are not currently assigned to any area objects in your model.

The drop-down box that initially says Add New Wall in the Click To area of the Define Wall/Slab/Deck Sections dialog box allows you to define new wall, slab and deck sections.

Defining Wall and Slab Sections

When you define a new wall or slab section, or modify an existing one, the Wall/Slab Section dialog box appears. Following is a discussion of each of the areas in this dialog box.

- **Section name:** Here you can specify or modify the name of a wall or slab section.
- **Material:** Here you can choose the material property for the slab or wall from a list of all defined material properties.
- **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 either 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 out-of-plane shearing deformations are considered in the analysis. When thick plate behavior is *not* included these shearing deformations are not considered in the analysis.

We recommend that in ETABS you typically do not use the thick plate option except maybe if 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 dialog box you can then choose an option to view the model based on the colors associated with the section properties. In this case each object appears in a color associated with its assigned section property. See the section titled "Building View Options" in Chapter 10 for more information. You can change the color associated with the material by clicking in the color box.

Defining Deck Sections



Tip:

If you want to use the Composite Beam Design post-processor then you must define the slab using Deck properties, not slab properties, even if you are using a solid slab over the composite beams.



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.

When you define a new deck section, or modify an existing one, the Deck Section dialog box appears. Following is a discussion of each of the areas in this dialog box.

- **Section name:** Here you can specify or modify the name of a deck section.
- **Type:** There are three options for the deck type. They are: filled deck, unfilled deck and solid slab. The type of deck section controls which features are active in the rest of the dialog box. Following is a discussion of the three deck type options.
 - ✓ **Filled deck:** If you select the filled deck option then 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 then the slab cover item in the Geometry area 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 then 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.
- **Geometry:** You specify the geometry of the slab and deck in this area. The following items are specified:
 - ✓ **Slab depth:** Depth of the slab not including the height of the metal deck.
 - ✓ **Deck depth:** Depth (height) of the metal deck.
 - ✓ **Rib Width:** Average width of the metal deck ribs.

**Note:**

Deck section properties have membrane behavior only. No plate bending behavior is modeled for deck sections.

11

**Note:**

When you assign deck section properties ETABS assumes that the deck spans in the same direction as the local 1-axis of the area object to which the deck is assigned

- ✓ **Rib Spacing:** 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:** Height of the shear studs after welding.
 - ✓ **Tensile Strength, Fu:** Fu value for the shear studs.
- **Material:** You specify 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 you specify a slab material property. If the deck is unfilled you 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.
 - ✓ **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/length² 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 dialog box.

A special case does exist for this mass, as follows. If the deck is a filled deck and the mass per unit volume of the designated Slab Material is zero then ETABS 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 then ETABS assumes the mass of the metal deck is also zero.

- **Display Color:** Here you assign a color to the deck section. If you use the **View menu > Set Building View Options** command to display the Set Building View Options dialog box you can then choose an option to view the model based on the colors associated with the section properties. In this case each object appears in a color associated with its assigned section property. See the section titled "Building View Options" in Chapter 10 for more information. You can change the color associated with the material by clicking in the color box.

ETABS has three built-in default area object properties. They are DECK1, SLAB1 and WALL1. These are, as the names indicate 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, a restriction on deleting area object properties is that ETABS 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

The different types of link properties available in ETABS are:

- Linear
- Damper
- Gap
- Hook
- Plastic1
- Isolator1
- Isolator2

Note:

 In a linear analysis ETABS 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 three different sets of properties assigned to them. They are linear properties, nonlinear dynamic properties that are used for nonlinear dynamic (time history) analysis and nonlinear static properties that are used for nonlinear static (pushover) analysis. Linear link elements can only have linear properties assigned to them. Note that you must have the nonlinear version of ETABS to utilize the nonlinear static and dynamic link properties. Discussion of the nonlinear dynamic and nonlinear static properties is beyond the scope of this manual.

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 a linear analysis ETABS 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. ETABS 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.

See Chapter 37 for further discussion of link elements.

Frame Nonlinear Hinge Properties

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. Discussion of the frame nonlinear hinge properties is beyond the scope of this manual.

Section Cuts

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. They are discussed in detail in the section titled "Section Cuts" in Chapter 26. Please refer to that discussion for a detailed explanation of section cuts in ETABS.

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. (See Chapter 31 for discussion of manual meshing). If you define these groups before manual meshing then 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 ETABS. However, before you use this command you first should define the group that is 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.

When you execute the **Define menu > Section Cuts** command the Section Cuts dialog box appears. The Section Cuts area of this dialog box lists the names of all the currently defined section cuts. The Click To area of the dialog box allows you to define new section cuts, modify existing section cut definitions and delete existing section cuts.

11

Defining Section Cuts

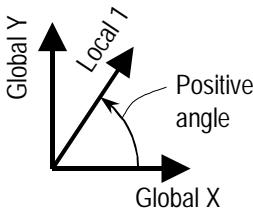
When you click the **Add Section Cut** button in the Section Cuts dialog box or when you highlight an existing section cut name and click the **Modify>Show Section Cut** button the Section Cut Data dialog box appears. This dialog box is broken into four areas that are described below.

- **Section cut name:** Here you can specify or modify the name of a section cut.
- **Group:** Here you specify or modify the name of the group that defines the section cut. See the section titled "Section Cuts" in Chapter 26 for more information.
- **Summation about this location:** Here you define 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 dialog box. Any value between -360 degrees and +360 degrees, inclusive can be input.

Response Spectrum Functions

A response spectrum function is simply a list of period versus spectral acceleration values. In ETABS 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 titled "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.

11

Response Spectrum Functions from a File

You can add a response spectrum definition to ETABS 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 ETABS should ignore. These header lines do not need \$ symbols at the beginning of them. ETABS quits reading the file either when it reaches the end of the file or when it reaches a blank line. Note that ETABS 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 ETABS.

Click the **Add Spectrum from File** button in the Click To area of the Define Response Spectrum Functions dialog box to add a new response spectrum function definition from an existing text file. This brings up the Response Spectrum Function Definition dialog box. The following areas exist in this dialog box:

- **Function name:** Here you can specify or modify the name of the response spectrum function.
- **Function file:** Click on the **Specify File** button in this area to bring up a dialog box where you indicate the name of the text file that includes your response spectrum data.

Typically ETABS 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 ETABS may suddenly be unable to locate the response spectrum file. If you click the **Convert to User-Defined** button then ETABS 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 ETABS skips the number of lines at the top of the file indicated in the Header Lines to Skip item.

- **Define Function:** This area displays the period and spectral acceleration values for the function. You can only view the values in this area. You can not edit these values unless you convert the function to a user-defined function. 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 dialog box. Then click the **Display Graph** button in the Function Graph area of the dialog box to display the graph of the function. This also fills in the values in the Define Function 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.

ETABS 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 then it skips to the next line.

- If there is not a \$ symbol as the first character on the line then ETABS reads the information on the line.
- If the line is blank or if the end of the file is reached then ETABS stops reading and closes the file.

User-Defined Response Spectrum Functions

Click the drop down box just below the **Add Spectrum from File** button in the Click To area of the Define Response Spectrum Functions dialog box and click on Add User Spectrum to add a new user-defined response spectrum. This brings up the Response Spectrum Function Definition dialog box. The following areas exist in this dialog box:

- **Function name:** Here you can specify or modify the name of the response spectrum function.
- **Define Function:** You 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 are 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 then simply click the **Refresh Graph** button located just below the graph.

Note:

Response spectra in ETABS are always defined as period versus spectral acceleration.

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

ETABS 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 dialog box 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 these code-specific items brings up a code-specific Response Spectrum Function Definition dialog box. The following areas exist in this dialog box:

- **Function name:** Here you can specify or modify the name of the response spectrum function.
- **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 ETABS includes are discussed in separate subsections below.
- **Define Function:** This area displays the period and spectral acceleration values for the function. You can only view the values in this area. You can not 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 then simply click the **Refresh 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 11-3.

$$C_{sm} = \frac{1.2A_vS}{RT_m^{2/3}} \leq \frac{2.5A_a}{R} \quad \text{Eqn. 11-3}$$

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_{DI} = 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_{DI} = 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 dialog box highlight an existing response spectrum name and then click on the **Modify>Show Spectrum** button to modify the spectrum. The same dialog box 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 dialog box and click the **Delete Spectrum** button.

Time History Functions



Tip:

You can define time history functions using one of several built-in time history function templates.

A time history function may be either 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.

Click the **Define menu > Time Functions** command to define time history functions. When you execute this command the Define Time History Functions dialog box appears. The Functions area of this dialog box lists the names of all the currently defined time history functions. The Click To area of the dialog box allows you to add a new function from a text file, add a new user-defined time history function, add a new time history function based on one of several ETABS built-in function templates, modify an existing time history function definitions and delete existing time history function definitions.

Time History Functions from a File

You can add a time history definition to ETABS 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 ETABS should ignore. These header lines do not need \$ symbols at the beginning

of them. ETABS quits reading the file either when it reaches the end of the file or when it reaches a blank line. Note that ETABS 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 dialog box to add a new time history function definition from an existing text file. This brings up the Time History Function Definition dialog box. The following areas exist in this dialog box:

- **Function name:** Here you can specify or modify the name of the time history function.
- **Function file:** Click on the **Specify File** button in this area to bring up a dialog box where you indicate the name of the text file that includes your time history function data.

Typically ETABS 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 ETABS may suddenly be unable to locate the function file. If you click the **Convert to User-Defined** button then ETABS 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 ETABS 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 ETABS how many function values or sets of time and function values are specified on each line.

- **Values are:** Here you specify 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 then you 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 then you also specify the number of characters per item. Each item on a line is assigned the same number of character spaces. ETABS 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 dialog box 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.

ETABS reads the 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 then it skips to the next line.
- If there is not a \$ symbol as the first character on the line then ETABS 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 then ETABS stops reading and closes the file.

User-Defined Time History Functions

Click the drop down box just below the **Add Function from File** button in the Click To area of the Define Time History Functions dialog box and click on Add User Function to add a new user-defined time history function. This brings up the Time History Function Definition dialog box. The following areas exist in this dialog box:

- **Function name:** Here you can specify or modify the name of the time history function.
- **Define Function:** You 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 are 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 then simply click the **Refresh 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.

Note that you can also define a user-defined periodic function by clicking the drop down box just below the **Add Function from**

File button in the Click To area of the Define Time History Functions dialog box and then clicking on Add User Periodic Function. This brings up the Time History User Periodic Function Definition dialog box. This dialog box is the same as that just described for the User Function except that it has one additional item that you can specify. That item is the number of cycles.

For a user periodic function you specify the time and function values for one cycle of the function and you specify the number of cycles. When ETABS uses the function it assumes it continues for the specified number of cycles even though you only specified values for the first cycle.

If you convert a user periodic function to a user function then values are shown for all of the specified cycles.

ETABS Template Time History Functions

ETABS allows you to easily define sine, cosine, ramp, sawtooth and triangular time history functions using built-in ETABS 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 dialog box 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 specialized Time History Function Definition dialog box. The following areas exist in this dialog box:

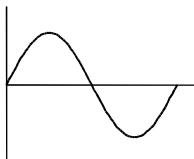
- **Function name:** Here you can specify or modify the name of the time history function.
- **Parameters:** You specify the parameters that define the template time history function in this area. These parameters vary for the different templates. The parameters specified for each of the templates included are discussed in separate subsections below.

- **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 can not edit these 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. 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 template parameters. If your computer has any problem with the automatic update then simply click the **Refresh Graph** button located just below the graph.

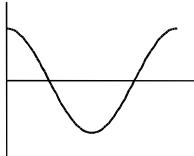
You can run your mouse pointer over the function graph and a dot appears along the line representing the time history function. The coordinates of the dot are reported in the box just below the graph.



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:** This is the period of the sine function. It is the time in seconds that it takes for the function to complete one cycle.
- **Number of Steps per Cycle:** This is the number of steps, that is, function value points, provided for each cycle of the function.



- **Number of Cycles:** This is the number of cycles in the function.
- **Amplitude:** This is 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 its positive maximum value again. The following parameters are specified for the cosine time history function template.

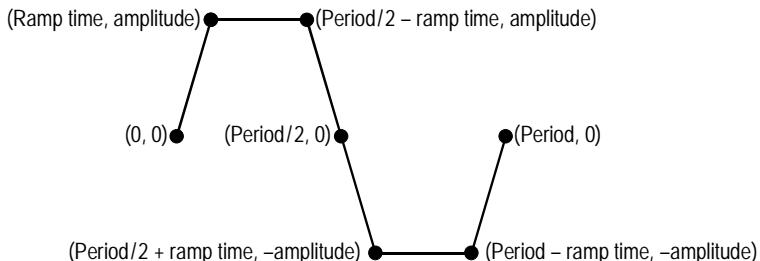
- **Period:** This is the period of the cosine function. It is the time in seconds that it takes for the function to complete one cycle.
- **Number of Steps per Cycle:** This is the number of steps, that is, function value points, provided for each cycle of the function.
- **Number of Cycles:** This is the number of cycles in the function.
- **Amplitude:** This is 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 below.

- **Ramp Time:** This is the time that it takes for the ramp function to initially reach its maximum value. It is usually set to one second.

Figure 11-4:
Points that define a cycle of a sawtooth time history function



- **Amplitude:** This is the maximum value of the ramp function. It is usually set to 1.
- **Maximum Time:** This is 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 11-4 illustrates a single cycle of a sawtooth time history function and the seven above-described points.

The following parameters are specified in the sawtooth time history function template.

- **Period:** This is the period of the sawtooth function. It is the time in seconds that it takes for the function to complete one cycle.
- **Ramp Time:** This is time that it takes for the sawtooth function to ramp up from a function value of 0 to its maximum amplitude.
- **Number of Cycles:** This is the number of cycles in the function.
- **Amplitude:** This is 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 sawtooth time history function template.

- **Period:** This is the period of the triangular function. It is the time in seconds that it takes for the function to complete one cycle.
- **Number of Cycles:** This is the number of cycles in the function.
- **Amplitude:** This is the maximum function value of the triangular function.

11 Static Load Cases

Static load cases are discussed in Chapter 27. In ETABS 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. As you will see in the ensuing discussion, 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.



Note:

See Chapter 27 for additional information on static load cases.

Click the **Define** menu > **Static Load Cases** command to define static load cases. This brings up the Define Static Load Case Names dialog box. The static load case names are specified in the Loads area of the dialog box. There is no limit on the number of static load cases that you can define.

Four separate items are specified in the Loads area of the dialog box. They are:

- **Load:** This is the name of the static load case.

- **Type:** This is the type of the static load case. ETABS 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 ETABS 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 titled "Live Load Reduction" under the section titled "Preferences" in Chapter 18 for more information.
 - ✓ **Quake:** Earthquake load.
 - ✓ **Wind:** Wind load.
 - ✓ **Snow:** Snow load.
 - ✓ **Other:** Other load that either does not fall into one of the above categories or you do not want it included in the design load combinations that are automatically created by ETABS.
- **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 of your 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, then 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 either 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 either choose None from it or you can choose one of several different code-specified loads which is then automatically created for the specified load case.

Note:

See Chapters 28 and 29 for detailed discussion of automatic seismic and wind loads.

If you do not want to use the automatic lateral loads and instead plan to assign your own loads using the commands available on the Assign menu then 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 then click the **Show Lateral Load** button. The parameters for the automatic seismic and wind loads are discussed in detail in Chapters 28 and 29, respectively.

Use the following procedure to add a new static load case in the Define Static Load Case Names dialog box:

- Type the name of the load case in the Load edit box.

- Select a load type from the Type drop-down box.
- Type a self-weight multiplier in the Self-Weight Multiplier edit box.
- If the load type specified is Quake or Wind then select an option from the Auto Lateral Load drop-down box.
- Click the **Add New Load** button.
- If an automatic lateral load is selected in the Auto Lateral Load drop-down box then click the **Show Lateral Loads** button and review or modify the parameters for the automatic lateral load in the resulting dialog box. Then click the **OK** button to return to the Define Static Load Case Names dialog box.

Use the following procedure to modify an existing static load case in the Define Static Load Case Names dialog box:

- Highlight the the existing load case in the Loads area of the dialog box. 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.

Use the following procedure to delete an existing static load case in the Define Static Load Case Names dialog box. 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 dialog box. 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

Click the **Define** menu > **Response Spectrum Cases** command to define a response spectrum case. This command brings up the Define Response Spectra dialog box. *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 dialog box lists the names of all the currently defined response spectrum cases. The Click To area of the dialog box allows you to define new response spectrum cases, modify existing response spectrum cases and delete existing response spectrum cases.

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 dialog box. The following subsections describe each of the areas in this dialog box:

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, ETABS 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 then you are specifying that there is two percent modal damping in all modes for the response spectrum analysis **and you are also telling ETABS that the response spectrum functions specified for this response spectrum case are for two percent damping.**

If you have link elements defined in your model and damping is specified in the linear properties of the link element then the actual damping for a mode may be larger than that specified in the structural and function damping term because ETABS 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 titled "Link Properties" for additional information on this.

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) ETABS 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 ETABS 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). ETABS then modifies the specified 4% damped spectrum by the factor determined in Equation 11-7.

$$\frac{2.31 - 0.41 \ln 6.3}{2.31 - 0.41 \ln 4} = \frac{1.555}{1.742} = 0.89 \quad \text{Eqn. 11-7}$$

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 can not override the modal damping specified for all modes on a mode-by-mode basis.

Modal Combination

In this area you specify the method ETABS uses to combine modal responses in the response spectrum analysis and you also define a damping value.

The following options are available for modal combinations:

Note:

ETABS defaults to the CQC method of modal combination.

- **CQC:** This is the Complete Quadratic Combination method described by E. L. Wilson, A. D. Kiureghian and E. Bayo (1981b). 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, then 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 11-4.

$$f_1 = \frac{S_{A\max}}{2\pi S_{V\max}} \quad \text{Eqn. 11-4}$$

where $S_{A\max}$ is the maximum spectral acceleration and $S_{V\max}$ 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 11-5.

$$f_2 = \frac{1}{3} f_1 + \frac{2}{3} f_r \quad \text{Eqn. 11-5}$$

where f_r is the rigid frequency of the seismic input, that is, that 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 11-6.

$$f_2 = f_r \quad \text{Eqn. 11-6}$$

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:** Combine 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:

$$R = \max(\bar{R}_1, \bar{R}_2, \bar{R}_3)$$

where

$$\bar{R}_1 = R_1 + 0.3(R_2 + R_3)$$

$$\bar{R}_2 = R_2 + 0.3(R_1 + R_3)$$

$$\bar{R}_3 = R_3 + 0.3(R_1 + R_2)$$

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% over-conservative 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/seconds² and that its value will change as you change the units in your model. Essentially ETABS 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 convert 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 by using the right hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

Time History Cases

Click the **Define menu > Time History Cases** command to define a time history case. This command brings up the Define Time History Cases dialog box. *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 dialog box lists the names of all the currently defined time history cases. The Click To area of the dialog box allows you to define new time history cases, modify existing time history cases and delete existing time history cases.

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 dialog box. The following subsections describe each of the areas in this dialog box:

History Case Name

Here you can specify or modify the name of the time history case.

Options

You specify various options for the time history analysis in the options area of the Time History Case Data dialog box.

- **Analysis type:** This option 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 you specify a single cycle of the periodic function and then ETABS assumes that the specified cycle continues indefinitely.

ETABS shows time history results for a single cycle that occur 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 (pushover) 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 dialog box where you can specify or modify the modal damping. In this dialog box you can specify a damping that applies to all modes and then, if desired, overwrite the damping for any mode(s). Following are three bullet items discussing the three areas in the Modal Damping dialog box.
- ✓ **Damping for All Modes:** Enter the damping for all modes in this dialog box. This is a percent critical damping. A damping that is 5% Of critical damping is entered as 0.05.

**Note:**

In ETABS 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 then 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 then 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 is 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 dialog box. 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 dialog box. 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 dialog box. Make sure that the Dynamic

Analysis check box is checked and click the **Set Dynamic Parameters** button to bring up the Dynamic Analysis parameters dialog box where you can specify the number of modes that is 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 then that damping override is simply ignored when the analysis is run.

- **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 times 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 times 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



Note:

If you start a time history from the final conditions of a previous history then both histories must be of the same type. That is, they must both be linear or they must both be nonlinear.

only have to run the other (gravity) time history once rather than multiple times.

Often times 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.

11

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 dialog box. 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 dialog box. 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 may either be a defined static load case, acc dir 1, acc dir 2 or acc dir 3. The three accelerations (acc dir 1, acc dir 2 and acc dir 3) are ground accelerations in the local axes directions of the time history. Positive acc dir 3 corresponds to the positive global Z direction always. See the discussion of the Angle item in this area for information about acc dir 1 and acc dir 2. When you



Note:

You can perform multiple support excitation time history analysis in ETABS using displacement time histories.

specify one of these three 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 this 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 ETABS 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 between two points displacement varies as a cubic between those same two points. Thus between these 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 3) then this scale factor has units of Length/seconds². If the load is a static load case then 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 that 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 this 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 then 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. Then the first five 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.

Static Nonlinear/Pushover Cases

11

Click the **Define menu > Static Nonlinear/Pushover Cases** command to define static nonlinear load cases. Discussion of the parameters used to define these load cases is beyond the scope of this manual.

Load Combinations

Load combinations are discussed in detail in Chapter 27. Click the **Define menu > Load Combinations** command to define load combinations. This command brings up the Define Load Combinations dialog box. The Combinations area of this dialog box lists the names of all the currently defined load combinations. The Click To area of the dialog box 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 dialog box. The following bullet items include brief discussions of each of the areas in this dialog box.

- **Load combination name:** Here you can specify or modify the name of the load combination. Refer to Chapter 27 for limitations on load combination names (labels).

**Note:**

*See Chapter 27
for discussion
of load combi-
nations.*

- **Load combination type:** Here you specify the type of load combination as ADD (Additive), ENVE (Envelope), ABS (Absolute) or SRSS. The meaning of each of these types of combinations is discussed in Chapter 27. 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

Refer to the section titled "Mass" in Chapter 27 for discussion of the mass source. Click the **Define menu > Mass Source** command to the mass source for ETABS. Following are bullet items discussing each of the areas in this dialog box.

- **Mass Definition:** Here you specify whether ETABS 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 ETABS determines the mass from element/object masses and additional masses. See Chapter 27 for more information.

**Note:**

See Chapter 27
for discussion
of the mass
source.

- **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 ETABS 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 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 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. Type in the revised scale factor in the Scale Factor 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 only lateral mass:** If this check box is checked then only assigned translational mass in the global X and Y axes directions and assigned rotational mass moments of inertia about the global Z-axis are considered in the analysis. All other assigned masses are 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 then leave this box unchecked.



Chapter 12

12

The ETABS Draw Menu

General

This chapter discusses the drawing tools and controls that are available on the ETABS Draw menu.

The **Draw menu > Select Object** command is simply used to switch you 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 **Pointer** button, on the side toolbar and executing one of the Select menu commands.

The **Draw menu > Reshape Object** command activates the reshaper tool. This tool is discussed in the section titled "Reshaper Tool" in Chapter 9.



Tip:

Most of the tools available on the Draw menu are also available on the side toolbar.

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 you can use the **Options menu > Lock Model** command or you can click the **Lock/Unlock Model** button, , on the main (top) toolbar. 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 ETABS Similar Stories Feature

Note:

The similar stories feature is only active in plan view. It works for drawing, assigning and selecting.

Do not overlook the Similar Stories feature in ETABS when drawing objects in plan view in ETABS. Similar story assignments are made in the story level data (**Edit menu > Edit Story Data > Edit** command). The similar stories feature drop-down box located on the right hand side of the status bar (just to the left of the drop-down coordinate system box) at the bottom of the ETABS window controls what happens when an object is drawn in ETABS.

The similar stories drop down box can be set to One Story, All Stories or Similar Stories. Each of these is discussed below. Note that the similar stories feature is only active in plan view. It works for drawing, assigning and selecting.

- **One Story:** This option means that the drawn object only occurs at the story level that it is drawn on.
- **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 to the story level at which the object is drawn. 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.

Drawing Point Objects

You can only draw point objects in plan view. You can not 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, , on the side toolbar. (You can also click on the **Create Points (plan, elev, 3D)** button, , after clicking on the Draw Point Objects button, but this is not currently necessary since there is only one flyout button). Once you have activated the Draw Point Objects command 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 of two grid lines *in the same coordinate/grid system* included in the "rubber band" window.

Drawing Line Objects

To draw line objects use the **Draw menu > Draw Line Objects** command or click the **Draw Line Objects** button, , on the side toolbar. If you use the menu option a submenu appears with five line object drawing options. Similarly, if you use the toolbar button option a new toolbar pops out with the same five line object drawing options. Those line object drawing options and their

associated toolbar buttons are discussed in the bullet items below. Note that the items in parenthesis, such as (plan, elev, 3D) indicate the types of views for which the line object drawing option is active. (See the subsection below titled "Floating Properties of Object Window for Line Objects" for additional information.)

- **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 then ETABS 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 ETABS 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 also think of the single left click as finishing the line but not picking up your pencil from the paper.

12

Tip:



To finish drawing a line object you either double left click on the end point or you single left click and then press the Enter key (or the Esc key) on your keyboard.

If you double left click on the end point of the line then ETABS does not assume that you 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 you still remain in the same line drawing mode. 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 you can 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 titled "The Two Modes of ETABS" in Chapter 4 for more information.

When using this command in an elevation view or 3D view if you draw a line object that crosses story levels then ETABS immediately breaks the object up at the story levels. For example if you draw a line object that has its top at the 4th story level and it bottom at the 2nd story level ETABS 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), **: Once you have activated the Create Columns in Region or at Clicks (plan) command there are two ways you can draw the columns. They are:
 - ✓ 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, of course, also to other story levels if you have the similar stories feature in the ETABS status bar activated. See the section titled "Similar Story Levels" in Chapter 22 for additional information on the similar stories feature.

- **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 then the spacing and extent (length) of the secondary beams is based on the existing beams rather than the grid lines. Figure 12-1 shows an example of a grid line space and secondary beams. Note that secondary beams are not included on the grid lines.

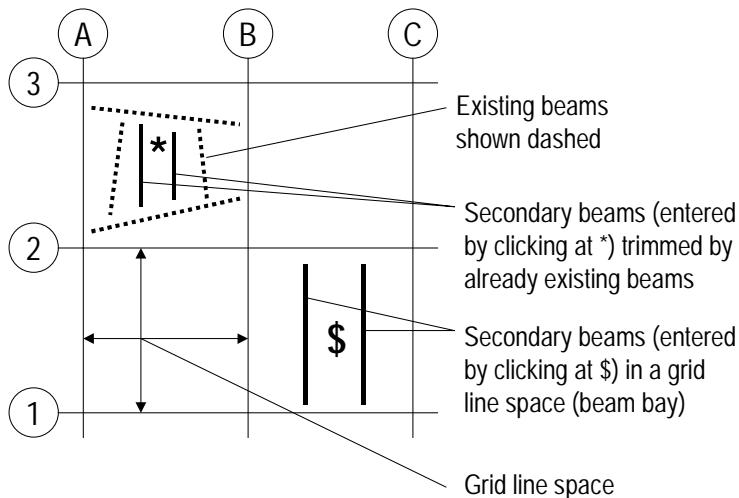
This command works in two different ways. You can click inside the space created by adjacent intersecting grid lines (from the same coordinate/grid system) and secondary beams are drawn in that space.



Note:

The spacing and orientation of secondary beams are controlled in the floating Properties of Object window.

Figure 12-1:
Example of secondary beams



12

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 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).

See the subsection titled "Floating Properties of Object Window for Line Objects" for additional information about secondary beams.

- **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.



Note:

The type of brace created (X, V, chevron, eccentric, etc.) is controlled in the floating Properties of Object window.

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.

See the subsection titled "Floating Properties of Object Window for Line Objects" for additional information about drawing braces beams.

Floating Properties of Object Window for Line Objects

There is a Show Floating Property Window toggle switch available on the Options menu. When this feature is activated a floating Properties of Object window pops up as soon as you click the menu item or toolbar button to draw a line object. This floating window includes a drop-down box with an alphabetical listing of all currently defined frame section properties. Whatever frame section property is showing in the pop up window is then assigned to the line object when you draw it. If you do not want to make a frame section assignment to the line object then you can choose None in the drop-down box.

Note that when you are drawing secondary beams you can also specify the spacing, span direction of the beams and whether the M3 moment at the beam ends is released (pinned) or not released (continuous) in the Properties of Object window. There are two spacing options available in the pop up box. They are:

- Specify a maximum spacing. In this case ETABS will draw an appropriate number of secondary beams such that they do not exceed the specified maximum spacing.
- Specify the number of equally spaced secondary beams.

When you are drawing braces in elevation you can specify in the Properties of Object window whether the braces are X-braces, V-braces, inverted V-braces (chevron), or single eccentric braces sloped forward or backward. The V-braces can be specified either eccentric or concentric in the window. Also you can specify whether the M3 moment at the brace ends is released (pinned) or not released (continuous).

Drawing Area Objects

To draw area objects use the **Draw menu > Draw Area Objects** command or click the **Draw Area Objects** button, , on the side toolbar. If you use the menu option a submenu appears with five area object drawing options. Similarly, if you use the toolbar button option a new toolbar pops out with the same five area object drawing options. Those area drawing options and their associated toolbar buttons are discussed in the bullet items below. 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. (See the subsection below titled "Floating Properties of Object Window for Area Objects" for additional information.)



Tip:

To finish drawing an area object you either double left click on the last point or you single left click and then press the Enter key on your keyboard.

- **Draw Areas (plan, 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 you can either double left click to finish the object or you can single left click and then press the Enter key on the keyboard.

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 ETABS does not allow the area object drawn to cross a story level. For example you can not 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. The bottom can not 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 then ETABS 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 ETABS 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 12-1 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 you 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 then ETABS 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 ETABS is expecting you to draw the end point of an area object (wall below) because you will see the dashed "rubber 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) then ETABS does not assume that you 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 you can press the Enter key on the keyboard. This will terminate the drawing of the next wall and you will 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.

Floating Properties of Object Window for Area Objects

There is a Show Floating Property Window toggle switch available on the Options menu. When this feature is activated a floating Properties of Object window pops up as soon as you click the menu item or toolbar button to draw an area object. This floating window includes a drop-down box with an alphabetical listing of currently defined wall/slab/deck section properties. The properties available in this drop down box depend on the type of view you are in. Whatever wall/slab/deck section property is showing in the pop up window is then assigned to the area object when you draw it. The Opening assignment in this drop-down box assigns an *unloaded* opening. If you do not want to make a wall/slab/deck section assignment to the area object then you can choose None in the drop-down box.

Developed Elevations

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 dialog box appears. In this dialog box you can define names for developed elevations that you plan to draw, modify the names of existing developed elevations (or ones you plan to draw), and delete names of existing developed elevations (or ones you plan to draw). 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 ETABS windows then all switch to special plan views used for creating and modifying developed elevation views. Note the following about these special plan views:

**Note:**

A line defining a developed elevation can not intersect (cross) itself and it also cannot close.

- ✓ In these 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 ETABS 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 dialog box you enter the special plan views to work on the developed elevation that was highlighted in the dialog box. This developed elevation may or may not currently be defined. If it is not defined you will see a blank plan view otherwise 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 either 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.

After you draw a developed elevation or if the developed elevation you highlighted in the Elevation Views dialog box 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 Reshape button, , on the side toolbar.
- Click on the line defining the developed elevation. Selection handle 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 while holding 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 and while holding down the mouse left button you can drag the selection handle (point) to a new location thus changing the shape of the line defining the developed elevation.
- When you are through reshaping a developed elevation definition do one of the following:
 - ✓ Right click in the ETABS 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.

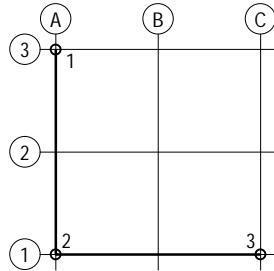
- ✓ If instead you want to work with another developed elevation then select a developed elevation view name from the drop down box in the status bar (at the bottom of the ETABS window) that ordinarily displays the similar stories option. When you click on a developed elevation view in this drop down 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 then click on the **Draw menu > Reshape Object** command or the Reshape button, , on the side toolbar again.

Note that when you finish drawing a line that defines a developed elevation you can then immediately either 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).

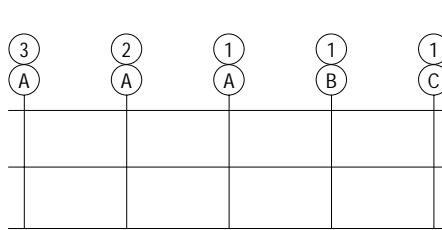
Figure 12-2 shows three example developed elevations together with their associated plan views. The elevations associated with a particular plan are shown next to the plan. The heavy line in the plan views shows the extent of each developed elevation. The open circles on the heavy lines show the locations 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 12-2c the final mouse click (point 5) can not occur at grid intersection A-3 (the same location as point 1) because this would close the line and developed elevations are not allowed to be closed in this manner.

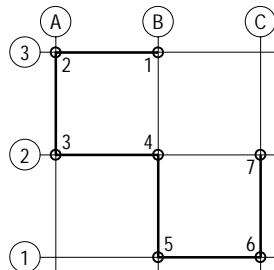
Also note that the three examples shown in Figure 12-2 all have the mouse clicks occurring on grid intersections. It is not necessary to click on grid intersections to define developed elevations. You can click anywhere in plan when defining a developed elevation and the elevation is created for the point that you clicked on.



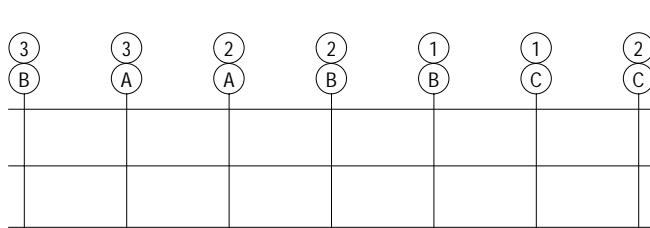
a) Plan



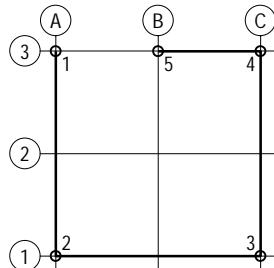
Developed Elevation



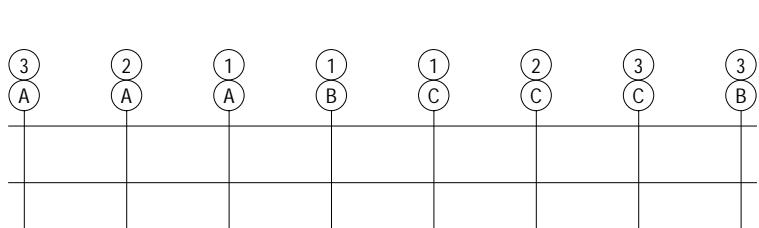
b) Plan



Developed Elevation



c) Plan



Developed Elevation

*(Above)***Figure 12-2:***Example definitions
of developed eleva-
tions*

Dimension Lines

12

You can use the **Draw menu > Draw Dimension Line** command to draw dimension lines at any location in a plan or elevation view. You can not draw dimension lines in a three-dimensional view. Once 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 done creating it. Now you can 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 completed.

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 18 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 that is in the ETABS status bar. The ETABS similar stories feature works for dimension lines. See the section titled "Similar Story Levels" in Chapter 22 for additional information on the similar stories feature.

You can use the **View menu > Set Building View Options** command or the **Set Building View Options** button,  located on the main (top) toolbar to toggle the visibility of the dimension lines on or off.

Once you have drawn a dimension line you can select it any time you are in the select mode by clicking directly on it. You can not select a dimension line by windowing it. The reason you might want to select a grid line is 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 titled "The ETABS Nudge Feature" in Chapter 9. This feature works as described there for dimension lines.

The reshaper tool is described in Chapter 9. The section titled "Reshaping Dimension Lines" in that chapter provides information on using the tool to relocate dimension lines.

Special Drawing Controls

ETABS provides drawing controls that allow you to snap to various items and drawing controls that allow you to constrain the direction in which you can draw a line. These types of drawing controls are discussed in the subsections below.

ETABS Snap Options

The ETABS snap features allow you to snap to various items when you are drawing or editing. There are six separate snap features available in ETABS. You can have these six features toggled on or off in any combination. You can toggle the snap features on and off using the **Draw menu > Snap to** command or by clicking one or more of the six snap feature toolbar buttons on the side toolbar. The six snap features and their associated toolbar buttons are:

- **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 is 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 available under 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 then select it from the side toolbar or using the **Draw menu > Snap to** command.
- 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 the distance that 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 available under the **Options menu > Preferences > Dimensions/Tolerances** command.

- When the desired snap location is 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 then 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 active. 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.

Drawing Constraints in ETABS

Drawing constraints provide the capability to constrain one of the axes when you are drawing an object in plan view or reshaping an object in plan view using the reshaper tool. See the section titled "Reshaper Tool" in Chapter 9 for information on reshaping objects. Using the drawing constraints you can quickly draw a line object parallel to one of the global axes or at any arbitrary angle. The drawing constraint tools can be activated using the **Draw menu > Constrain Drawn Line to** command or they can be activated by pressing either the X, Y, Z or A key on your keyboard.

The drawing constraints include:

- **Constant X:** Locks the X component of the next point so that it is the same as the previous point.
- **Constant Y:** Locks the Y component of the next point so that it is the same as the previous point.
- **Constant Z:** Locks the Z component of the next point so that it is the same as the previous point.

- **Constant Angle:** Allows you to specify an angle in degrees in the status bar at the bottom of the ETABS window. Drawing is then constrained along this angle. The angles are measured from the global X-axis. Positive angles appear counterclockwise as you look down on the plan.

Note that when you use the keyboard command to specify a constant angle constraint, every time you press the A key on the keyboard the constant angle changes to be equal to the angle of the line that is within the screen selection tolerance (**Options menu > Preferences > Dimensions/Tolerances**) distance from the mouse pointer. You can, of course, always change this angle by editing it in the ETABS status bar.

- **None:** Removes the current drawing constraint. Pressing the space bar on your keyboard also removes the current drawing constraint.

There are three steps to using the constraint tools:

- Locate the first point.
- Press one of the constraint keys on the keyboard (X, Y, Z or A) or use the **Draw menu > Constrain Drawn Line to** command.
- Locate the next point. ETABS only picks up the unconstrained component of the next point.

Drawing constraints are always removed as soon as you draw the next point.

Note that snaps can be used in conjunction with constraints. In this case only the unconstrained component of the selected snap point is used when a constraint is selected.



The ETABS Select Menu

General

The Select menu in ETABS provides basic options and tools for selecting objects in your ETABS model. This chapter discusses those options and tools. Do not overlook the important note concerning window selections in plan view at the end of the bullet item titled "Window."

The similar stories feature of ETABS discussed in the Section titled "Similar Story Levels" in Chapter 22 works for selections. Note that the similar stories feature is only active in plan view.

Basic Methods of Selecting Objects

There are three basic methods of selecting objects in ETABS. They are:

- **Left click:** Here you simply left click on an object to select it. If there are multiple objects one on top of the

**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.

13

**Note:**

An entire object must lie within the rubber band window for the object to be selected.

other then you can hold down the Ctrl key on your keyboard as you left click on the objects. A dialog box will appear that allows you to specify which object you want to select.

- **Window:** Here you draw a window around one or more objects to select them. To draw a window around an object first position your mouse pointer 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. Finally release the left mouse button. 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.
 - ✓ You do not necessarily have to start the window above and to the left of the object(s) you are selecting. You could alternatively start the window 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.

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 are selected.

- **Intersecting Line:** Here you draw a line through one or more objects to select them. To use this selection method you first tell ETABS that you want to use intersecting line selection either by clicking the **Select menu > Select Using Intersecting Line** command or by clicking on the **Set Intersecting Line Select Mode** button, .

**Tip:**

Use the intersecting line selection method in a perspective view to select all columns at a story level.

Once you have told ETABS to use the intersecting line selection method you then draw the intersecting line as follows. First position your mouse pointer to one side of the object(s) you want to select. Then 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. Finally release the left mouse button. 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.
- ✓ When you make a selection using the intersecting line method you do not then remain in an intersecting line mode such that you could immediately make another intersecting line selection. Instead you default back to a window selection mode. You must tell ETABS that you want to use the intersecting line selection method (either by clicking the **Select menu > Select Using Intersecting Line** command or by clicking on the **Set Intersecting Line Select Mode** button, ) every single time you use the selection method even if you are doing several intersecting line selections in a row.

Other methods of selecting objects are discussed in the next section.

Other Methods of Selecting Objects

You can click on the select menu to gain access to other methods of selecting objects. These methods include:

- **Select on XY plane:** Using this command you simply 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:** Using this command you simply 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:** Using this command you simply 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:** Using this command you can select any collection of objects that has been defined as a group. See Chapter 26 for discussion of groups.
- **Select by frame sections:** Using this command you can specify a frame section property name and all line objects that are assigned that frame section property are selected.
- **Select by wall/slab/deck sections:** Using this command you can specify a wall/slab/deck section property name and all area objects that are assigned that wall/slab/deck section property are selected.
- **Select by link properties:** Using this command you can specify a link property name and all line objects that are assigned that link property are selected.



Note:

These selection methods also work for deselection.

- **Select by line object type:** Using this command you can specify that certain types of line objects are to be selected. The choices for the types of line objects are column, beam, brace, null or dimen lines (short for dimension lines). See the section titled "Line Object Labeling and Frame Type" in Chapter 24 for more information.
- **Select by area object type:** Using this command you can specify that certain types of area objects are to be selected. The choices for the types of area objects are floor, wall, ramp or null. See the section titled "Area Object Labeling and Area Type" in Chapter 23 for more information. Note that openings are a subset of null area objects.
- **Select by story level:** Using this command you can specify a story level and all objects (point, line and area) associated with that story level are selected.
- **Select all:** This selection method 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, , on the side toolbar to execute this command.
- **Select invert:** This command changes the selection such that the currently selected objects are no longer selected and all objects that are not currently selected are selected.
- **Get previous selection:** This selection method selects whatever objects were last previously selected. For example suppose you select some line objects by clicking on them and then assign them some frame section properties. You can then use this command to select the line objects again and assign something else to them such as member end releases. You can also use the **Restore Previous Selection** button, , on the side toolbar to execute this command.

- **Clear selection:** Using this command clears the selection of all currently selected objects. It is an all or nothing command. You can not selectively clear a portion of a selection using this command. If you want to selectively clear a selection you can either left click on the selected objects one at a time or you can use the deselect tool documented below. You can also use the **Clear Selection** button, , to clear the entire selection.

Deselecting Objects

You can deselect objects one at a time by left clicking on the selected objects. A more powerful way to deselect items is to use the **Select menu > Deselect** command. This gives you access to most of the above described selection methods except that now they are used to deselect rather than to select. Suppose that you wanted to select all of the objects in your model except for those in a particular XZ plane. You could do this by first using the **Select menu > Select All** command and then using the **Select menu > Deselect > XZ Plane** command.



Chapter 14

14

The ETABS Assign Menu

General

The Assign menu in ETABS provides basic options and tools for assigning section properties, loads, and more to area, line and point objects in your ETABS model. This chapter discusses those options and tools. Note that before you make an assignment to an object using the Assign menu you must first **select** the object.

The similar stories feature of ETABS discussed in the Section titled "Similar Story Levels" in Chapter 22 works for assignments. Note that the similar stories feature is only active in plan view.

Assignments to Point Objects

Use either the **Assign menu > Joint/Point** command or the **Assign menu > Joint/Point Load** command to make assignments to point objects. The following subsections discuss the assignments that you can make to point objects.

Rigid Diaphragm Assignments to Point Objects



Tip:

You can also assign rigid diaphragms to area objects using the **Assign menu > Shell/Area > Rigid Diaphragm command.**

14



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.

Use the **Assign menu > Joint/Point > Rigid Diaphragm** command to designate a rigid diaphragm. This command provides 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.

When you select one or more point objects and click the **Assign menu > Joint/Point > Rigid Diaphragm** command the Assign Diaphragms dialog box appears. The Diaphragms area of the Assign Diaphragms dialog box lists the names of all the currently defined rigid diaphragms. The Click To area of the dialog box allows you to define new rigid diaphragms, change an existing diaphragm name and delete an existing diaphragm.

Typically to assign a new diaphragm you select the point objects, enter the Assign Diaphragm dialog box, type a new diaphragm name in the edit box in the Diaphragms area, click the **Add New Diaphragm** button and then click the **OK** button.

If you want to add additional point objects to an existing diaphragm definition you select all of the point objects that you want to add to the diaphragm, enter the Assign Diaphragm dialog box, highlight the name of the diaphragm that you are adding the point objects to in the edit box in the Diaphragms area and click the **OK** button. Note that this adds to the existing diaphragm definition, it does not replace it.

You can click the **Assign menu > Joint/Point > Rigid Diaphragm** command and enter the Assign Diaphragms dialog box without first making a selection if you wish (as long as the model is unlocked). This is useful if you want to change a diaphragm name or delete a diaphragm. In this case you enter the Assign Diaphragms dialog box without first making a selection, make the desired name changes or deletions and then click the **OK** button. Since you entered the dialog box without a selection ETABS knows not to make any assignment to the highlighted diaphragm when you click the **OK** button. In this special case where you enter the dialog box without a selection, whatever diaphragm is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the dialog box.

**Tip:**

To delete specific point objects from a diaphragm definition, select those point objects, enter the Assign Diaphragm dialog box, highlight None in the Diaphragms area and click the **OK** button.

If you want to delete some point objects from a diaphragm definition, then select all of the point objects that you want to delete from the diaphragm, enter the Assign Diaphragm dialog box, highlight None in the Diaphragms area and click the **OK** button.

In ETABS 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.

Note that you can also apply a rigid diaphragm to an area object. See the subsection titled "Rigid Diaphragm Assignments to Area Objects" in the section titled "Assignments to Area Objects" later in this chapter for more information. In most instances it is better to assign the rigid diaphragm to an area object.

Panel Zone Assignments to Point Objects

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 dialog box.

When specifying a panel zone assignment to a point object you indicate 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 these items is discussed in the subsections below. The headings used for the subsections correspond to the areas in the Assign Panel Zone dialog box.

You can not 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 subsection below titled "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. These rotational properties are assigned to the panel zone spring that connects the two ETABS-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 then the panel zone assignment is ignored by ETABS. When you run the analysis a warning message reports panel zone assignments that are ignored because of this (if any).

- **Elastic properties from column and doubler plate:** When using this option you specify a doubler plate thickness. ETABS 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. These rotational properties are assigned to the panel zone spring that connects the two ETABS-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 then the panel zone assignment is ignored by ETABS. When you run the analysis a warning message reports panel zone assignments that are ignored because of this (if any).

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 then you must specify the panel zone properties as a link property.*

- **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). These two rotational spring properties are assigned to the panel zone spring that connects the two ETABS-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 ETABS-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 then ETABS assumes that degree of freedom is rigidly connected. Therefore, because of this, if for some reason you want one of the degrees of freedom of the panel zone to have essentially zero stiffness you should specify a small stiffness for that degree of freedom in the link properties.

**Note:**

A panel zone assignment allows relative movement, typically rotation, between beam and column, beam and brace, or brace and column members.

If you have nonlinear static properties defined for the link property then those properties are considered when you run a static nonlinear (pushover) analysis. Similarly, If you have nonlinear dynamic properties defined for the link property then those properties are considered when you run a nonlinear time history analysis. Thus when you indicate that the panel zone properties 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 2-Axis area.

14***Connectivity***

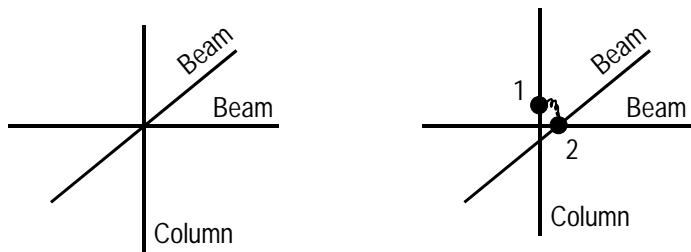
The choices for panel zone connectivity are beam-column, beam-brace and brace-column. Unless you choose the Specified Link Property option in the Properties area, the only active panel zone connectivity option is beam-column.

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 then the panel zone assignment is ignored by ETABS. For example, if you specify brace-column connectivity at a point object and there are no braces at the point object, then the panel zone assignment is ignored. When you run the analysis a warning message reports panel zone assignments that are ignored because of this (if any).

Following are descriptions of the three types of panel zone connectivity:

- **Beam-column:** For beam-column connectivity two separate joints are internally created by ETABS 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.

Figure 14-1:
Panel zone connectivity



a) Beam-Column Connection

b) Panel Zone Representation

Consider Figure 14-1a which shows a beam-column joint. Figure 14-1b shows the effect of assigning a panel zone with beam-column connectivity to this joint. Joints 1 and 2 are created internally by ETABS. 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.

Joints 1 and 2 are connected by zero-length springs whose properties are 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 ETABS 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 discussion of beam-column connectivity for additional information.
- **Brace-column:** For brace-column connectivity two separate joints are internally created by ETABS 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

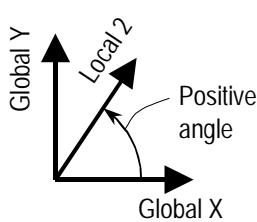
the discussion of beam-column connectivity for additional information.

Local 2-Axis

The local axes of the panel zone element are determined as follows:

- **Local 1-axis:** The positive local 1-axis is in the same direction as the positive global Z-axis (upward), always.
- **Local 2-axis:** You specify the positive direction of the local 2-axis as one of the following:
 - ✓ **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, then 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, then 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.
- **Local 3-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. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

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 then 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). When this option is selected, the items in the Properties, Connectivity and Local Axis areas of the dialog box 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 dialog box is opened.

Restraint (Support) Assignments to Point Objects

You use the **Assign menu > Joint/Point > Restraints (Supports)** command to bring up the Assign Restraints dialog box 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 dialog box. 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 titled "Analysis Options" in Chapter 15 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.

14

The Fast Supports area of the Assign Restraints dialog box 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.

Assigning a support to a point object is only meaningful if structural objects are connected to the point object. Otherwise the point support will support air, so to speak; that is, it will not support anything. Either the point object must be connected directly to a structural object or it must be on top of a floor-type area object that ETABS can automatically mesh. See Chapter 30 for discussion of the automatic meshing capability of ETABS.

Point Spring Assignments to Point Objects

Use the **Assign Menu > Joint/Point > Point Springs** command to open the Assign Springs dialog box 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 dialog box:



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:** Here you specify the spring stiffness for anywhere from one to all 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 here.
- **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 then this option increases the existing spring stiffness assuming, of course, 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 then 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). When this option is selected the items in the Spring Stiffness in Global Directions area of the dialog box 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 dialog box is opened.

Assigning a spring to a point object is only meaningful if structural objects are connected to the point object. Otherwise the spring will support air, so to speak; that is, it will not support anything. Either the point object must be connected directly to a structural object or it must be on top of a floor-type area object that ETABS can automatically mesh. See Chapter 30 for discuss-

sion of the automatic meshing of floor-type area objects that is done by ETABS.

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 zero). If you decide to assign some negative spring stiffness to a point object, you should do it with great care because it can get you into trouble. Negative spring stiffness at a point during the analysis causes your structure to be unstable and thus ETABS terminates the analysis and provides an error message that there is an instability. ETABS does not check for negative spring stiffness prior to 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 dialog box. 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. You specify this 6x6 matrix by clicking the **Assign Menu > Joint/Point > Point Springs** command and then clicking the **Advanced** button to bring up the Coupled 6x6 Spring dialog box. In this dialog box you define the 6x6 matrix for the coupled springs.

Figure 14-2 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 since the off-diagonal terms are all zero. The diagonal terms are what you are specifying 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 dialog box rather than clicking on the **Advanced** button.

Figure 14-2:
6x6 symmetric matrix of spring stiffness coefficients

$$\begin{Bmatrix} F_x \\ F_y \\ F_z \\ M_x \\ M_y \\ M_z \end{Bmatrix} = - \begin{bmatrix} ux & uxuy & uxuz & uxr_x & uxry & uxrz \\ uy & uyuz & uyr_x & uyr_y & uyrz \\ uz & uzrx & uzry & uzrz \\ rx & rxry & rxrz \\ ry & ryzr \\ rz \end{bmatrix}_{\text{symmetric}} \begin{Bmatrix} ux \\ uy \\ uz \\ rx \\ ry \\ rz \end{Bmatrix}$$

Link Property Assignments to Point Objects

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 can not modify the local axes directions for grounded, zero-length link elements.



Tip:

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 dialog box. Simply highlight the name of a defined link property in the dialog box and then click the **OK** button to assign a link property to the selected point object(s).

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 dialog box. Simply highlight the name of a defined link property in the dialog box and then click the **OK** button to assign a link property to the selected point object(s).

If you want to remove a link property assignment from a point object then 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 dialog box and click the **OK** button.

Note you can not 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 "Panel Zone Assignments to Point Objects" for more information.

Additional Point Mass Assignments to Point Objects

Note:

You can assign multiple link properties (elements) to the same point object.

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 ETABS 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 titled "Mass Source" in Chapter 11 and the section titled "Mass" in Chapter 27 for more information.

Clicking the **Assign menu > Joint/Point > Additional Point Mass** command brings up the Assign Masses dialog box. Following are descriptions of the three areas in this dialog box.

- **Masses in Global Directions:** Specify the translational masses in the global coordinate system direction in this area. The masses are entered in Force-Second²/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-Second² units. Figure 14-3 is provided to assist you in calculating rotational mass moments of inertia for various shaped 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 then this option increases the existing mass assuming, of course, you are specifying a positive mass.

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, $MMI_0 = 0$	$MMI_{CM} = MMI_0 + MD^2$

(Above)

Figure 14-3:*Mass moment of inertia for various areas*

- ✓ **Replace existing masses:** Replaces the currently specified mass, if any, with the new spring stiffness assignment. If there is not an existing assignment then 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). When this option is selected the items in the Masses in Global Directions and Mom. of Inertia in Global Directions areas of the dialog box 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 dialog box is opened.

14 Note that if you select the Include only Lateral Mass option when defining the mass source then 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 zero). If you decide to assign some negative mass to a point object, you should do it with great care because it can get you into trouble. If ETABS detects negative mass at a point during the analysis it will terminate the analysis and provide an error message about negative mass. ETABS does not check for negative mass prior to running the analysis.

Force Loads to Point Objects

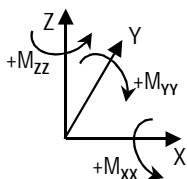
Use the **Assign menu > Joint/Point Loads > Force** command to bring up the Point Forces dialog box and assign point loads to selected point objects. Note that the point loads are specified in global coordinate system directions.

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 you should use the **Assign menu > Frame/Line Loads > Point** command to apply the point loads.

The following bullet items discuss the three areas in the Point Forces dialog box:

- **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. See the section titled “The Right Hand Rule” in Chapter 23 for more information.



- **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 then this option increases the total point load on the point object assuming, of course, 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 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 point load assignments made to the selected point object(s). When this option is selected the items in the Loads area of the dialog box 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 dialog box is opened.

Ground Displacement Assignments to Point Objects

Use the **Assign menu > Joint/Point Loads > Ground Displacement** command to bring up the Ground Displacements dialog box 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 box below carefully.

Assigning a displacement to a point object is only meaningful if structural objects are connected to the point object. Otherwise the point object displacement will have no affect on the structure. Either the point object must be connected directly to a structural object or it must be on top of a floor-type area object that ETABS can automatically mesh. See Chapter 30 for discussion of the automatic meshing of floor-type area objects that is done by ETABS.

The following bullet items discuss the three areas in the Ground Displacements dialog box:

- **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.



Important Information about Ground Displacement Assignments

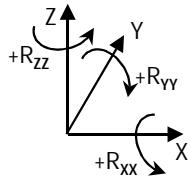
Point object ground displacements are only meaningful when they are applied to point objects that are connected to the ground in the direction that the displacement is applied. Point objects are connected to the ground through one of the following:

- Restraints
- Springs
- Grounded link elements assigned to a single point object

When ground displacements are assigned to a point object that is restrained then 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 then that displacement is ignored by ETABS when the analysis is run.



- **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. See the section titled “The Right Hand Rule” in Chapter 23 for more information.
- **Options:** The following three assignment options are available:
 - ✓ **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 dialog box 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 dialog box is opened.

Temperature Loads Assignments to Point Objects

Use the **Assign menu > Joint/Point Loads > Temperature** command to bring up the Point Temperatures dialog box 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 discuss the three areas in the Point Temperatures dialog box:

- **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

**Shortcut:**

You can use the Assign menu > Frame/Line > Frame Section command to simultaneously define frame sections and assign them to selected line objects.

temperature change assignments have already been made then this option increases the total temperature change on the point object assuming, of course, you are specifying a positive temperature change.

- ✓ **Replace existing load:** Replaces the currently specified temperature change, if any, with the new temperature change 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 temperature change assignments made to the selected point object(s). When this option is selected the items in the Temperature area of the dialog box 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 dialog box is opened.

Assignments to Line Objects

Use either the **Assign menu > Frame/Line** command or the **Assign menu > Frame/Line Load** command to make assignments to line objects. The following subsections discuss the assignments that you can make to line objects.

Frame Section Assignments to Line Objects

Use the **Assign menu > Frame/Line > Frame Section** command to open the Assign Frame Properties dialog box and assign frame section properties to line objects. To use this command select some line objects, then click the menu command to open the dialog box, highlight a frame section in the Properties area of the dialog box and click the **OK** button to make the assignment.

The frame section property that you highlight in the Properties area can either be a previously defined property or you can define it on the fly while you are in the Assign Frame Properties dialog box. For defining frame section properties, the Assign Frame Properties dialog box has all of the functionality that the Define Frame Properties dialog box has. See the section titled "Frame Section Properties" in Chapter 11 for more information.

Frame Releases and Partial Fixity Assignments to Line Objects

You can release any of the three translational and three rotational degrees of freedom at either end of a line object. However, these releases only are meaningful 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 done by specifying a spring stiffness when you assign the member end release.

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.

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 to open the Assign Frame Releases dialog box and assign frame releases to line objects. To use this command select some line objects, then click the menu command to open the dialog box 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 then check the No Releases check box at the bottom of the dialog box 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 assures 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. ETABS checks for these conditions when you click the OK button in the Assign Frame Releases dialog box and if unstable releases are specified provides a message telling you this.

- Releasing U1 (axial) at both ends.
- Releasing U2 (shear force 2, major) at both ends.
- Releasing U3 (shear force 3, minor) at both ends.
- Releasing R1 (torsion) at both ends.
- Releasing R2 (moment 22, minor) at both ends *and* U3 (shear force 3, minor) at either end.
- Releasing R3 (moment 33, major) at both ends *and* U2 (shear force 3, major) at either end.

Frame Rigid Offset Assignments to Line Objects

Note:

Do not confuse frame member end joint offsets with end offsets along the length of the member. They are two entirely separate things.

In ETABS 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 members are large and the length of the overlap can be a significant fraction of the total length of the frame element. ETABS provides the capability of defining rigid end offsets along the length of frame members to account for these finite dimensions of structural elements. See the subsection below titled "Rigid End Offsets Along the Length of Frame Elements" for more information.

**Note:**

Rigid end offsets along the length of a frame element account for the finite size of beam and column intersections.

**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.

When a line object is used to model a frame section the line object is assumed to be located at the centroid of the frame section. Thus when line objects (frame sections) intersect in a model it means that the centroids of the associated frame objects intersect. In a real structure this is 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. ETABS provides the capability of defining rigid frame end joint offsets to account for these eccentric connections. See the subsection below titled "Rigid Frame End Joint Offsets" for more information.

Use the **Assign menu > Frame/Line > Frame Rigid Offsets** command to open the Assign Frame End Offsets dialog box where you can define both rigid end offsets along the length of frame elements and rigid frame end joint offsets. Be careful that you do not get these two types of rigid offsets confused.

Any end offset assigned to a line object is ignored unless the line object also has a frame section assigned to it.

Rigid End Offsets Along the Length of Frame Elements

Rigid end offsets along the length of frame members are defined in the End Offset Along Length area of the Assign Frame End Offsets dialog box. Use the **Assign menu > Frame/Line > Frame Rigid Offsets** command to open this dialog box.

In the End Offset Along Length area you have the choice of having ETABS determine the end offset lengths automatically or specifying them yourself. You also can specify the rigid-zone factor. These items are described below.

Automatically Calculated End Offset Lengths

ETABS automatically calculates offset lengths for beam and column-type frame elements. 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:**

ETABS outputs forces at the inside face of end offsets along the length of the member.

14

When ETABS 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 ETABS 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 ETABS automatically calculated end offsets along the length of frame members:

- When more than one beam frames into a column ETABS 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.

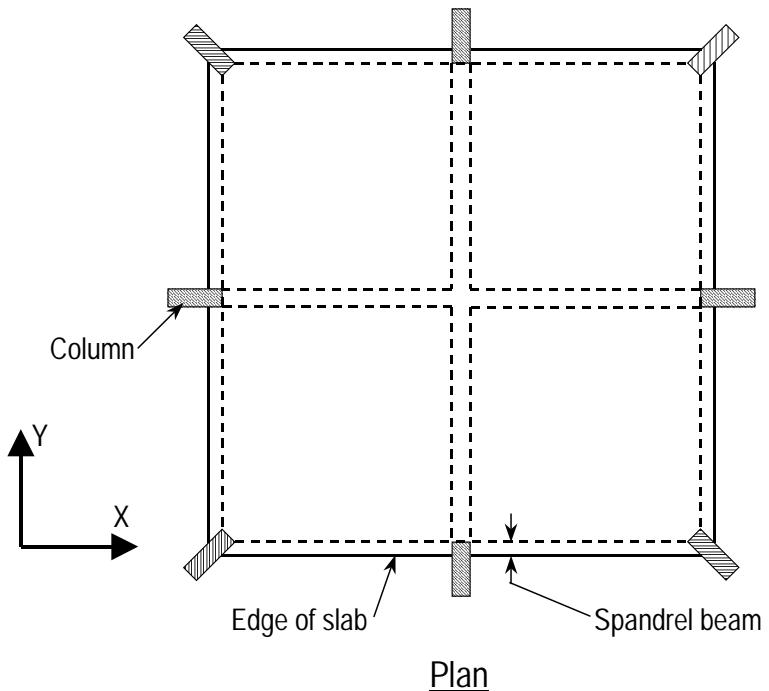
End Offset Properties and the Rigid-Zone Factor

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 rigid the outside portion of the end offset is assumed rigid, that is, the portion at the end of the frame member. By default ETABS 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 is 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 these 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.

Figure 14-4:
Example rigid end
joint offsets



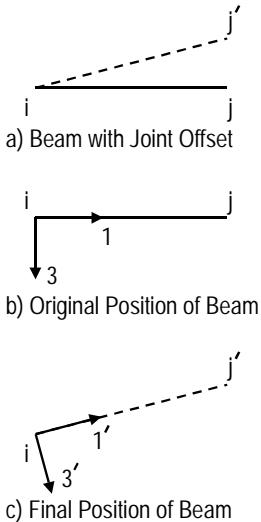
Rigid Frame Joint Offsets

Note:
When you specify member end joint offsets the local axes of the member are always based on the final position of the member after the end joint offsets are applied.

Rigid frame joint offsets are defined in the Frame Joint Offset area of the Assign Frame End Offsets dialog box. Use the **Assign menu > Frame/Line > Frame Rigid Offsets** command to open this dialog box. In the Frame Joint Offset area you specify the global X, Y and Z joint offsets at each end point of the frame element.

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.

The floor plan shown in Figure 14-4 illustrates a concrete beam and slab system with such a condition. Note that all of the spandrel beams frame into the edge of the column, not the column center line. This circumstance can be modeled in ETABS by providing a joint offset to the top (j-end) and bottom (i-end) of each column in either the global X direction, global Y direction, or both directions depending on how the column is oriented.



14

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 are 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 are applied.

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 both sketches b and c the local 2-axis points upward and thus does not show in the plan view sketches. ETABS bases the local axes of the beam on those shown in sketch c.

Frame Output Station Assignments to Line Objects

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 then it is the minimum number of output stations. If it is not reported in parenthesis then it is the maximum spacing between output stations.

Frame output stations are designated locations along a frame element. They are used as locations to report output forces, perform design and plotting points used for graphic display of force diagrams. When force diagrams are plotted, exact forces are plotted at each output station and then these 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 to designate the output stations for a frame element. Two options are available for defining output stations for a beam:

- **Specify the minimum number of output stations along the beam:** In this case ETABS 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.

**Note:**

Use the View menu > Set Building View Options command to toggle the display of frame element output stations on and off.

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 face of the end offset, not at the center of the support.

- **Specify the maximum spacing between stations:** In this case ETABS 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.

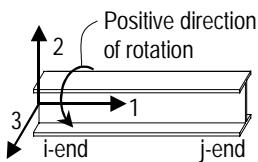
Note in the above that ETABS determines the location of output stations in a different order depending on whether you specify a minimum number of stations or a maximum spacing of stations.

By default, for beams output stations are provided at a maximum spacing of 2 feet for English units and 0.5 meters for metric units, and of course, at all point load locations. By default a minimum of three output stations are specified (the two ends and the middle) for columns and braces.

Local Axes Assignments to Line Objects

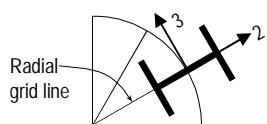
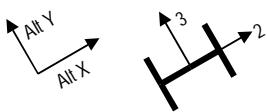
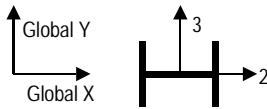
By default the local 1-axis of a line object extends from the i-end of the element to the j-end. The default orientation of the local 2 and 3 axes depends on the frame-type (column, beam or brace) and in some instances the orientation of the frame element itself. See the section titled "Line Object Local Axes" in Chapter 24 for more information.

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 to bring up the Axis Orientation dialog box. There are four options in this box:



- **Angle:** Rotate 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 counter-clockwise, that is, the right hand rule applies.
- **Rotate by Angle:** Rotate 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 counter-clockwise, that is, the right hand rule applies.
- **Column Major Direction (local 2-axis) 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:

14

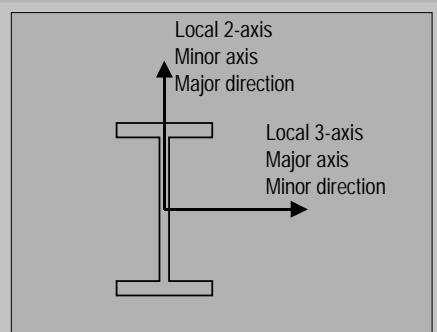


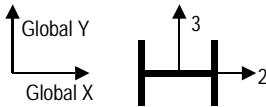
- ✓ If the column falls at the intersection of two global coordinate system grid lines then the major direction (local 2-axis) is the same as the positive global X-axis.
- ✓ If the column falls at the intersection of two grid lines from an additional rectangular coordinate system then the major direction (local 2-axis) is the same as the positive X-axis of that additional coordinate system.
- ✓ If the column falls at the intersection of two grid lines from an additional cylindrical coordinate system then the major direction (local 2-axis) is in the outward radial direction of that additional coordinate system.



Column Major Direction

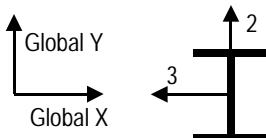
The **column major direction** is the same as the local 2-axis direction (which is also the same as the minor axis). Loads acting in the major direction cause M₃ bending and V₂ shear. In a wide flange member this corresponds to bending resisted by the flanges and shear resisted by the web.



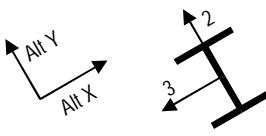


- ✓ If the column does not fall at the intersection of two grid lines from the same coordinate system then the major direction (local 2-axis) is the same as the positive global X-axis.

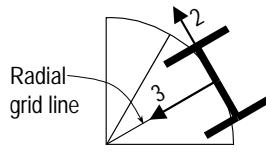
- **Column Major Direction (local 2-axis) is Y (or Tangential):** This option has no effect unless the selected element is a column. It sets the column major direction (the local 2-axis) as follows:



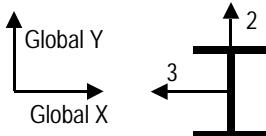
- ✓ If the column falls at the intersection of two global coordinate system grid lines then the major direction (local 2-axis) is the same as the positive global Y-axis.



- ✓ If the column falls at the intersection of two grid lines from an additional rectangular coordinate system then the major direction (local 2-axis) is the same as the positive Y-axis of that additional coordinate system.



- ✓ If the column falls at the intersection of two grid lines from an additional cylindrical coordinate system then the major direction (local 2-axis) is in the tangential direction of that additional coordinate system pointing counterclockwise.



- ✓ If the column does not fall at the intersection of two grid lines from the same coordinate system then the major direction (local 2-axis) is the same as the positive global Y-axis.

Frame Property Modifier Assignments to Line Objects

Use the **Assign menu > Frame/Line > Frame Property Modifiers** command to bring up the Analysis Property Modification Factors dialog box. Here you can specify modification factors for the following frame analysis section properties in your model.

- Cross sectional (axial) area

**Note:**

The frame property modifiers only affect the analysis properties. They do not affect the design properties.

- Shear area in 2 direction
- Shear area in 3 direction
- Torsional constant
- Moment of inertia about the 3-axis
- Moment of inertia about the 2-axis

The modification factors are multiplied times 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.

14

Link Property Assignments to Line Objects

Use the **Assign menu > Frame/Line > Link Properties** command to assign link properties to a line object. This command brings up the Assign Link Properties dialog box. Simply highlight the name of a defined link property in the dialog box and then click the **OK** button to assign a link property to the selected line object(s).

**Note:**

You can assign multiple link properties (elements) to the same line object.

If you want to remove a link property assignment from a line object then 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 dialog box and click the **OK** button.

Frame Nonlinear Hinge Assignments to Line Objects

Use the **Assign menu > Frame/Line > Frame Nonlinear Hinges** command to brings up the Assign Frame Hinges (Push-over) dialog box where you can assign nonlinear frame hinges (pushover) 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.

If you have selected a single element before implementing this command then the dialog box shows you the currently assigned hinges, if any. If you have selected multiple elements before implementing this command then:

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 then the dialog box shows you those assignments.
- If the elements do not all have the same hinge assignments then the dialog box 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 is filled in the drop-down box and edit box at the top of the dialog box. 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 is filled in the drop-down box and edit box at the top of the dialog box. Click the **Delete** button.

When you have finished specifying the hinge property assignments click the **OK** button to exit the dialog box.

Pier Label Assignments to Line Objects

A wall pier 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 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 then you must assign the pier label to the line and area objects separately. For example, suppose 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 you would 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 Design Manual 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 dialog box. There you can either 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 then select the objects, enter the Pier Names dialog box, highlight None and click the **OK** button.

You can click the **Assign menu > Frame/Line > Pier Label** command and enter the Pier Names dialog box 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 this case you enter the Pier Names dialog box without first making a selection, make the desired name changes or deletions and then click the **OK** button. Since you entered the dialog box without a selection ETABS knows not to make any assignment to the highlighted

pier when you click the **OK** button. In this special case where you enter the dialog box without a selection, whatever pier is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the dialog box.

Spandrel Label Assignments to Line Objects

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 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 then you must assign the spandrel label to the line and area objects separately. For example, suppose a wall spandrel that is to be labeled S23 is made up of both line and area objects. You would 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 you would 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 Design Manual for more information on wall spandrel labeling.

Typically to assign a new spandrel label to a line object you select the line object and click the **Assign menu > Frame/Line > Spandrel Label** command to enter the Spandrel Names dialog box. There you can either 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 then select the objects, enter the Spandrel Names dialog box, highlight None and click the **OK** button.

**Tip:**

ETABS distributes the springs associated with the line object to all of the nodes associated with the internal-to-

ETABS (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 assure that internally in ETABS a sufficient number of springs are used in the analysis model.

You can click the **Assign menu > Frame/Line > Spandrel Label** command and enter the Spandrel Names dialog box 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 this case you enter the Spandrel Names dialog box without first making a selection, make the desired name changes or deletions and then click the **OK** button. Since you entered the dialog box without a selection ETABS knows not to make any assignment to the highlighted spandrel when you click the **OK** button. In this special case where you enter the dialog box without a selection whatever spandrel is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the dialog box.

14

Line Spring Assignments to Line Objects

Use the **Assign menu > Frame/Line > Line Springs** command to bring up the Assign Spring dialog box 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 can not define tension-only or compression-only line springs.

ETABS distributes the springs associated with the line object to all of the nodes associated with the internal-to-ETABS (analysis model) representation of the line object. Note that internally ETABS may mesh (break up) a line object into several elements with associated points between each element. See the section titled "Overlapping Line Objects" in Chapter 24 for more information.

If you are modeling a beam on elastic foundation with a line spring you may want to mesh the line object yourself to assure that internally in ETABS a sufficient number of springs are used in the analysis model. ETABS 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 dialog box. They are:

- **Line spring:** Here you specify the direction of the springs as one of the three local axes of the line object and you specify a stiffness for the spring. The units for the stiffness are Force/Length².
- **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 then this option increases the existing spring stiffness assuming, of course, 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 then 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 dialog box 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 dialog box 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, you should do it with great care because it can get you into trouble. If negative spring stiffness occurs at any point in your model during the analysis then ETABS terminates the analysis and provides an error message that there is an instability. ETABS does not check for negative spring stiffness prior to running the analysis.

Additional Line Mass Assignments to Line Objects



Tip:

Additional line mass is only considered by ETABS if the mass is specified to be determined from material property masses.

14
Additional line mass is ignored if the mass is determined from a load combination.

Use the **Assign menu > Frame/Line > Additional Line Mass** command to assign additional line mass to a line object. Note that the additional line mass is only considered by ETABS 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 titled "Mass Source" in Chapter 11 and the section titled "Mass" in Chapter 27 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 is to be considered (see the section titled "Mass Source" in Chapter 11) then the additional line mass is only active in the global X and Y directions.

Clicking the **Assign menu > Frame/Line > Additional Line Mass** command brings up the Assign Mass dialog box. Following are descriptions of the two areas in this dialog box.

- **Line mass:** Specify the translational mass per unit length in this area. The masses are entered in Force-Second²/Length² 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 then this option increases the existing line mass assuming, of course, 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 then 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). When this option is selected the items in the Line Mass area of the dialog box 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 dialog box is opened.

Automatic Frame Mesh/No Mesh Assignments to Line Objects

ETABS automatically meshes frame elements as necessary in the analysis. See Chapter 30 for details of automatic meshing. In some cases you may not want ETABS to automatically mesh a frame element.



Note:

See Chapter 30
for more information on the
automatic
meshing performed by
ETABS.

14

For example, by default where you have intersecting X-braces ETABS would connect these 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 Mesh/No Mesh** command to tell ETABS not to automatically mesh them. You can also use this command again if you later decide that you want ETABS to automatically mesh the braces. When you execute this command you have three options:

- **Mesh it:** This tags the frame element to be automatically meshed, as required, by ETABS. By default all line objects have this tag when they are drawn.
- **Don't mesh it:** This tags the frame element to *not* be automatically meshed by ETABS.
- **Cancel:** This gives you a way to get out of the command without having to assign a mesh it or a don't mesh it tag to the frame element.

**Note:**

The positive directions for point moments are determined using the right hand rule. See the section titled "The Right Hand Rule" in Chapter 23 for more information. 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.

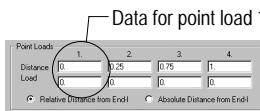
14

Point Load Assignments to Line Objects

Use the **Assign menu > Frame/Line Loads > Point** command to bring up the Frame Point Loads dialog box and assign point loads to selected line objects. The following bullet items discuss the four areas in the Frame Point Loads dialog box:

- **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.
- **Load Type and Direction:** Here you specify whether the loads are point loads or point moments. You also specify 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 either an absolute distance or a relative distance. An absolute distance is the actual distance from the left end of the line object to the point where the load intensity is specified. 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.

If you want to specify more than four point loads you 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 to again bring up the Frame Point Loads dialog box 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 then this option increases the total point load on the line object at that location assuming, of course, 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 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 point load assignments made to the selected line object(s). When this option is selected the items in the Load Type and Direction and the Point Loads areas of the dialog box 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 dialog box is opened.

Distributed Load Assignments to Line Objects

Use the **Assign menu > Frame/Line Loads > Distributed** command to bring up the Frame Distributed Loads dialog box 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 discuss the four areas in the Frame Distributed Loads dialog box:

- **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.
- **Load Type and Direction:** Here you specify whether the loads are forces (line loads) or moments (line moments). You also specify 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:

Distributed loads can be uniform or non-uniform (trapezoidal) and they can be full length or partial length.

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:

The positive directions for distributed moments are determined using the right hand rule. See the section titled "The Right Hand Rule" in Chapter 23 for more information. 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.

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.

Also note that only forces can be specified as projected loads, not moments. Figure 14-5 shows an example of how ETABS considers projected loads on line objects. Figure 14-5a illustrates a projected uniform distributed load of intensity w . The direction of the load is the ETABS gravity projection direction. Note that this is equivalent to a force of $w(\cos\theta)$ acting along the entire length of the line object in the gravity direction as shown in Figure 14-5b.

- **Trapezoidal Loads:** Here you specify non-uniform distributed loads acting on a line object. The distributed loads can be specified either 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.

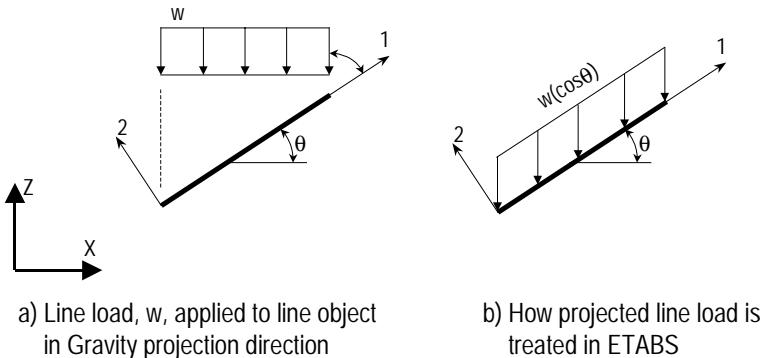
The loaded length for a trapezoidal load may be specified using either relative or absolute distances. An absolute distance is the actual distance from the left end of the line object to the point where the load intensity is specified. A relative distance is equal to the distance from the left end of 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.

Note:

Input partial length uniform loads as trapezoidal loads.

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 14-6 shows some examples. All of the distances shown in Figure 14-6 are relative distances.

Figure 14-5:
Uniform load, w ,
acting on a line ob-
ject in the Gravity
projection direction



You should always 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. ETABS ignores any boxes where the distance is smaller than the distance in the previous box.

Note:

In the Frame Distributed Loads dialog box trapezoidal and uniform load assignments are additive.

- **Uniform Load:** In this area you can 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 then this option increases the total point load on the point object at that location assuming, of course, 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 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.

Figure 14-6:
Examples of trapezoidal loads

	<table border="1"> <thead> <tr> <th>Set #</th><th>1</th><th>2</th><th>3</th><th>4</th></tr> </thead> <tbody> <tr> <td>Distance</td><td>0.25</td><td>0.5</td><td>0</td><td>0</td></tr> <tr> <td>Load</td><td>1</td><td>1</td><td>0</td><td>0</td></tr> </tbody> </table>	Set #	1	2	3	4	Distance	0.25	0.5	0	0	Load	1	1	0	0
Set #	1	2	3	4												
Distance	0.25	0.5	0	0												
Load	1	1	0	0												
	<table border="1"> <thead> <tr> <th>Set #</th><th>1</th><th>2</th><th>3</th><th>4</th></tr> </thead> <tbody> <tr> <td>Distance</td><td>0</td><td>1</td><td>0</td><td>0</td></tr> <tr> <td>Load</td><td>0</td><td>1</td><td>0</td><td>0</td></tr> </tbody> </table>	Set #	1	2	3	4	Distance	0	1	0	0	Load	0	1	0	0
Set #	1	2	3	4												
Distance	0	1	0	0												
Load	0	1	0	0												
	<table border="1"> <thead> <tr> <th>Set #</th><th>1</th><th>2</th><th>3</th><th>4</th></tr> </thead> <tbody> <tr> <td>Distance</td><td>0</td><td>0.5</td><td>1</td><td>0</td></tr> <tr> <td>Load</td><td>0</td><td>1</td><td>1</td><td>0</td></tr> </tbody> </table>	Set #	1	2	3	4	Distance	0	0.5	1	0	Load	0	1	1	0
Set #	1	2	3	4												
Distance	0	0.5	1	0												
Load	0	1	1	0												
	<table border="1"> <thead> <tr> <th>Set #</th><th>1</th><th>2</th><th>3</th><th>4</th></tr> </thead> <tbody> <tr> <td>Distance</td><td>0</td><td>0.33</td><td>0.67</td><td>1</td></tr> <tr> <td>Load</td><td>0</td><td>1</td><td>1.25</td><td>0</td></tr> </tbody> </table>	Set #	1	2	3	4	Distance	0	0.33	0.67	1	Load	0	1	1.25	0
Set #	1	2	3	4												
Distance	0	0.33	0.67	1												
Load	0	1	1.25	0												
	<table border="1"> <thead> <tr> <th>Set #</th><th>1</th><th>2</th><th>3</th><th>4</th></tr> </thead> <tbody> <tr> <td>Distance</td><td>0</td><td>0.5</td><td>0.5</td><td>1</td></tr> <tr> <td>Load</td><td>1</td><td>1</td><td>2</td><td>2</td></tr> </tbody> </table>	Set #	1	2	3	4	Distance	0	0.5	0.5	1	Load	1	1	2	2
Set #	1	2	3	4												
Distance	0	0.5	0.5	1												
Load	1	1	2	2												

- ✓ **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 Load Type and Direction and the Point Loads areas of the dialog box 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 dialog box is opened.

Temperature Load Assignments to Line Objects

Use the **Assign menu > Frame/Line Loads > Temperature** command to bring up the Line Object Temperatures dialog box 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 discuss the four areas in the Line Object Temperatures dialog box:

- **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.
- **Object Temperature:** Here you specify 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 then enter 0 for the uniform temperature change. A positive temperature change corresponds to an increase in the temperature of an object.
- **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 dialog box*. The following three assignment options are available:
 - ✓ **Add to existing object temperatures:** Adds the specified uniform temperature change to the line object. If one or more uniform temperature change assignments have already been made then this option increases the total uniform temperature change on the line object assuming, of course, 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 below for more information.

Note:

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.

- ✓ **Replace existing object temperature:** Replaces the currently specified uniform temperature change, if any, with the new uniform temperature change assignment. If there is not an existing assignment then the new assignment is still made. This is the default option. Note that only the temperature changes in the load case that is specified above are replaced.

This option has no affect on the end point temperature option. See the End Point Temperature Option bullet item below for more information.

- ✓ **Delete existing loads:** Deletes any and all uniform temperature change assignments made to the selected line object(s). When this option is selected any value input in the Object Temperature area of the dialog box 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 above are deleted.

This option has no affect on the end point temperature option. See the End Point Temperature Option bullet item below for more information.

Note that the default option is Replace and that the program defaults to this every time the dialog box is opened.

- **End Point Temperature Option:** If you check the Include Effect of Point Temperatures check box in this area then ETABS 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. ETABS assumes that the temperature change varies linearly along the length of the line object based on the specified changes at the end points.

Note:

When end point temperatures are specified to be included in the line object temperature load ETABS 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 don't want to include the effect of point temperatures then simply leave the box unchecked.

Note that the effect of the end point temperatures is not additive to itself. You either consider the end point temperatures or you do not. You control this by either checking or unchecking the box. Thus the options in the Object Temperature Options area have no meaning and consequently no affect on the option of including the effect of the point temperatures.

Assignments to Area Objects

Use either the **Assign menu > Frame/Line** command or the **Assign menu > Frame/Line Load** command to make assignments to line objects. The following subsections discuss the assignments that you can make to line objects.

Wall, Slab and Deck Section Assignments to Area Objects

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 to open the Assign Wall/Slab/Deck Sections dialog box and assign section properties to area objects. To use this command select some area objects, then click the menu command to open the dialog box, highlight a wall, slab or deck section in the Sections area of the dialog box 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 or you can define it on the fly while you are in the Assign Wall/Slab/Deck Sections dialog box. For defining wall, slab and deck section properties, the Wall/Slab/Deck Sections dialog box has all of the functionality that the Define Wall/Slab/Deck Sec-

tions dialog box has. See the section titled "Wall/Slab/Deck Section Properties" in Chapter 11 for more information.

Important note concerning decks: When you assign deck section properties ETABS assumes that the deck spans in the same direction as the local 1-axis of the area object to which the deck is assigned. See the section titled "Default Area Object Local Axes" in Chapter 23 for definition of the local axes for area objects.

Opening Assignments to Area Objects



Tip:

You can assign **unloaded** openings to area objects as you draw them by selecting the **Openings** option in the floating Properties of Object dialog box.

Use the **Assign menu > Shell/Area > Openings** command to bring up the Assign Openings dialog box and designate a selected area object as an opening. In the Assign openings dialog box you can indicate that the area object is one of the following:

- **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 ETABS.
- **Loaded opening:** ETABS 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 done by ETABS (see Chapter 30) and some of the manual meshing that you can do (see Chapter 31) 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.

Rigid Diaphragm Assignments to Area Objects

Use the **Assign menu > Shell/Area > Rigid Diaphragm** command 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 ap-

**Tip:**

You can also assign rigid diaphragms to point objects using the Assign menu > Joint/Area > Rigid Point command.

plicable to floor type area objects and to null-type area objects that happen to be in a horizontal plane.

In ETABS 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.

Internally in ETABS 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 ETABS automatically meshing the area object.

When you select one or more area objects and click the **Assign menu > Shell/Area > Rigid Diaphragm** command the Assign Diaphragms dialog box appears. Refer to the subsection titled "Rigid Diaphragm Assignments to Point Objects" in the section titled "Assignments to Point Objects" earlier in this chapter for a full description of this dialog box.

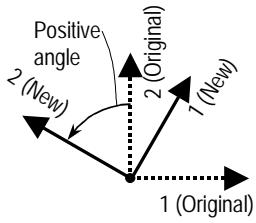
Note that you can also apply a rigid diaphragm constraint directly to point objects. See the subsection titled "Rigid Diaphragm Assignments to Point Objects" in the section titled "Assignments to Point Objects" earlier in this chapter for more information. In most instances it is better to assign the rigid diaphragm to an area object.

Local Axes Assignments to Area Objects

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. See the section titled "Default Area Object Local Axes" in Chapter 23 for more information on the local axes orientation of area objects.

**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.



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 to bring up the Assign Local Axis dialog box. Here you specify an angle in degrees. The angle specified is 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 counter-clockwise 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 can not rotate the local axes of pier and spandrel elements. See Chapter 38 for discussion of pier and spandrel local axes.

Shell Stiffness Modifiers Assignments to Area Objects

14

Use the **Assign menu > Shell/Area > Shell Stiffness Modifiers** command to bring up the Analysis Stiffness Modification Factors dialog box. Here you can specify modification factors 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 times 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 thickness³ of the shell element.

See the section titled "Shell Element Internal Forces and Stresses" in Chapter 36 for additional information.

Pier Label Assignments to Area Objects

A wall pier 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 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 then you must assign the pier label to the line and area objects separately. For example, suppose 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 you would 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 Design Manual for more information on wall pier labeling.

Typically to assign a new pier label to an area object you select the area object and click the **Assign menu > Shell/Area > Pier Label** command to enter the Pier Names dialog box. There you can either 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 then select the objects, enter the Pier Names dialog box, highlight None and click the **OK** button.

You can click the **Assign menu > Shell/Area > Pier Label** command and enter the Pier Names dialog box 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 this case you enter the Pier Names dialog box without first making a selection, make the desired name changes or deletions and then click the **OK** button. Since you entered the dialog box without a selection ETABS knows not to make any assignment to the highlighted pier when you click the **OK** button. In this special case where you enter the dialog box without a selection whatever pier is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the dialog box.

Spandrel Label Assignments to Area Objects

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 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 then you must assign the spandrel label to the line and area objects separately. For example, suppose a wall spandrel that is to be labeled S23 is made up of both line and area objects. You would 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 you would 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 Design Manual for more information on wall spandrel labeling.

Typically to assign a new spandrel label to an area object you select the area object and click the **Assign menu > Shell/Area > Spandrel Label** command to enter the Spandrel Names dialog box. There you can either 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 then select the objects, enter the Spandrel Names dialog box, highlight None and click the **OK** button.

You can click the **Assign menu > Shell/Area > Spandrel Label** command and enter the Spandrel Names dialog box 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 this case you enter the Spandrel Names dialog box without first making a selection, make the desired name changes or deletions and then click the **OK** button. Since you entered the dialog box without a selection ETABS knows not to make any assignment to the highlighted spandrel when you click the **OK** button. In this special case where you enter the dialog box without a selection, whatever spandrel is highlighted when you click the **OK** button retains exactly the same definition it had before you entered the dialog box.

Area Spring Assignments to Area Objects

Use the **Assign menu > Shell/Area > Area Springs** command to brings up the Assign Spring dialog box 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:**

ETABS 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.

ETABS 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. Note that in some cases internally ETABS may mesh (break up) an area object into several elements with associated points between each element. See Chapter 30 for more information on automatic meshing. In other cases you must manually mesh the area object. See Chapter 31 for more information on manual meshing.

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 assure that internally in ETABS a sufficient number of springs are used in the analysis model. ETABS 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 dialog box. They are:

- **Area spring:** Here you specify the direction of the springs as one of the three local axes of the area object and you specify a stiffness for the spring. The units for the stiffness are Force/Length³.
- **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 then this option increases the existing spring stiffness assuming, of course, 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 then 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 items in the Area Spring area of the dialog box 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 dialog box 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, you should do it with great care because it can get you into trouble. If negative spring stiffness occurs at any point in your model during the analysis then ETABS terminates the analysis and provides an error message that there is an instability. ETABS does not check for negative spring stiffness prior to running the analysis.

Additional Area Mass Assignments to Area Objects

Use the **Assign menu > Shell/Area > Additional Area Mass** command to assign additional area mass to an area object. Note that the additional area mass is only considered by ETABS 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 titled "Mass Source" in Chapter 11 and the section titled "Mass" in Chapter 27 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 titled "Mass Source" in Chapter 11) then the additional area mass is only active in the global X and Y directions.

Clicking the **Assign menu > Shell/Area > Additional Area Mass** command brings up the Assign Mass dialog box. Following are descriptions of the two areas in this dialog box.

- **Area mass:** Specify the translational mass per unit area in this region. The masses are entered in Force-Second²/Length³ units.



Tip:

Additional area mass is only considered by ETABS 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.

- **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 then this option increases the existing area mass assuming, of course, 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 then 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 this option is selected the items in the Area Mass area of the dialog box 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 dialog box is opened.

Automatic Membrane Floor Mesh/No Mesh Assignments to Area Objects

ETABS automatically meshes area objects that are assigned deck properties or slab properties with membrane behavior only into the analysis model as necessary. See Chapter 30 for details of automatic meshing. In some cases you may not want ETABS to automatically mesh an area object into the analysis model.

For example, by default when you have a composite beam floor system ETABS automatically meshes the deck at every beam and girder. This allows ETABS to automatically distribute the loading on the deck to each beam or girder in an appropriate manner. Suppose you have already assigned loads to the beams as frame loads and have not assigned any load (including self-weight) to the deck. In this case you might decide that you want to mesh the deck yourself, using a coarser mesh than the ETABS automatic mesh. In this case you would tag the deck not to be

Note:

See Chapter 30 for more information on the automatic meshing performed by ETABS.

meshed. We do not see this example as a very likely scenario, but it does illustrate when you might use this feature.

To tag the deck not to be meshed you select it and click the **Assign menu > Shell/Area > Automatic Membrane Floor Mesh/No mesh** command. You can also use this command again if you later decide that you want ETABS to automatically mesh the area objects. When you execute this command you have three options:

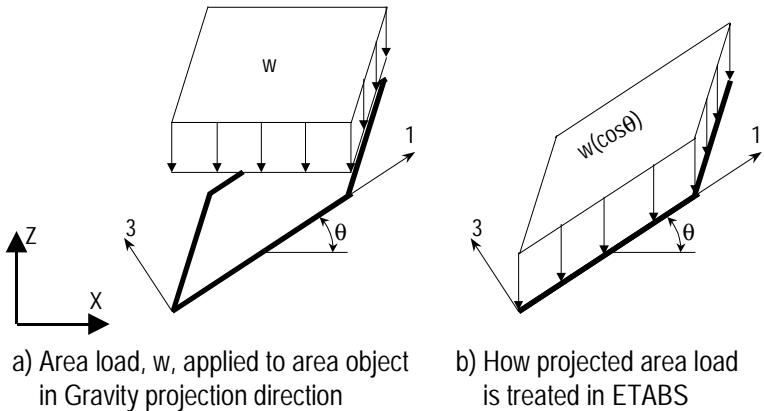
- **Mesh it:** This tags the area object to be automatically meshed, as required, by ETABS into the analysis model. By default all membrane floor area objects have this tag when they are created.
- **Don't mesh it:** This tags the area object *not* to be automatically meshed by ETABS into the analysis model.
- **Cancel:** This gives you a way to get out of the command without having to assign a mesh it or a don't mesh it tag to the area object.

14 Uniform Surface Load Assignments to Area Objects

Use the **Assign menu > Shell/Area Loads > Uniform** command to bring up the Uniform Surface Loads dialog box and assign uniform loads to selected area objects. The following bullet items discuss the four areas in the Frame Distributed Loads dialog box:

- **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.
- **Uniform Load:** Here you specify the uniform load value and the direction of the load. The following directions are possible:
 - ✓ Local-1

Figure 14-7:
Uniform surface load, w , acting on an area object in the Gravity projection direction



- ✓ 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 14-7 shows an example of how ETABS considers projected loads on area objects. Figure 14-7a illustrates a projected uniform surface load of intensity w . The direction of the load is the ETABS gravity projection direction. Note that this is equivalent to a force of $w(\cos\theta)$ acting over the entire surface of the area object in the gravity direction as shown in Figure 14-7b.

- **Options:** The following three assignment 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 then this option increases the total uniform load on the area object assuming, of course, 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 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 uniform load assignments made to the selected area object(s). When this option is selected the items in the Uniform Load area of the dialog box 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 dialog box is opened.

Temperature Load Assignments to Area Objects

Use the **Assign menu > Shell/Area Loads > Temperature** command to bring up the Area Object Temperatures dialog box 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 dialog box:

- **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**

Note:

When end point temperatures are specified to be included in the area object temperature load ETABS assumes that the temperature change varies linearly over the surface of the area object based on the specified changes at the corner points.

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. You either consider the corner point temperatures or you do not.

menu > Static Load Cases command to define load case names.

- **Object Temperature:** Here you specify 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, then enter 0 (zero) for the uniform temperature change. A positive temperature change corresponds to an increase in the temperature of an object.
- **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 dialog box*. The following three assignment options are available:

- ✓ **Add to existing object temperatures:** Adds the specified uniform temperature change to the area object. If one or more uniform temperature change assignments have already been made then this option increases the total uniform temperature change on the area object assuming, of course, 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 below for more information.

- ✓ **Replace existing object temperature:** Replaces the currently specified uniform temperature change, if any, with the new uniform temperature change assignment. If there is not an existing assignment then the new assignment is still made. This is the default option. Note that only the temperature changes in the load case that is specified above are replaced.

This option has no affect on the corner point temperature option. See the Corner Point Temperature Option bullet item below 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 dialog box 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 above are deleted.

This option has no affect on the corner point temperature option. See the corner Point Temperature Option bullet item below for more information.

Note that the default option is Replace and that the program defaults to this every time the dialog box is opened.

- **Corner Point Temperature Option:** If you check the Include Effect of Point Temperatures check box in this area then ETABS 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. ETABS 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 simultaneously 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 don't want to include the effect of point temperatures then simply leave the box unchecked.

Note that the effect of the corner point temperatures is not additive to itself. You either consider the corner point temperatures or you do not. You control this by either checking or unchecking the box. Thus the options in the Object Temperature Options area have no meaning and consequently no affect on the option of including the effect of the point temperatures.

Group Name Assignments

Groups are discussed in detail in Chapter 26. To define a group first select the objects that you want to be part of the group. Then click the **Assign menu > Group Name** command. The Assign Group dialog box appears. Either highlight an existing group name in the dialog box 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.

Note:

If the name of the group appearing in the edit box in the Groups area of the Assign Groups dialog box does not match any of the group names listed in that area then the **OK** button is not active until you click the **Add New Group** button to add that group name to the list of groups.

Important note: If you highlight an existing group name then 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 dialog box lists the names of all the currently defined groups. The Click To area of the dialog box 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.

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 dialog box when that group name is highlighted in the dialog box. See the subsection titled "View by Colors" in Chapter 10 for additional information.

**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 then first select the group, next select the objects you want to assign to the group and then make the assignment.

14

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 and enter the Assign Group dialog box 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 these cases you enter the Assign Groups dialog box without first making a selection, make the desired name changes, color changes or deletions and then click the **OK** button. Since you entered the dialog box without a selection ETABS 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 dialog box 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 dialog box.

Clear Display of Assigns

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 then the frame section assignments are displayed for all line objects in the model. Sometimes you will want to get rid of these assignment displays. You can use the **Assign menu > Clear Display of Assigns** command to do this.

Note that you can also remove the display of assignments by clicking the **Show Undeformed Shape** button, , on the main (top) toolbar or by clicking the **Display menu > Show Undeformed Shape** command.



Chapter 15

15

The ETABS Analyze Menu

The Analyze menu in ETABS provides basic features for starting and controlling your building analysis. This chapter discusses the commands available on the Analyze menu.

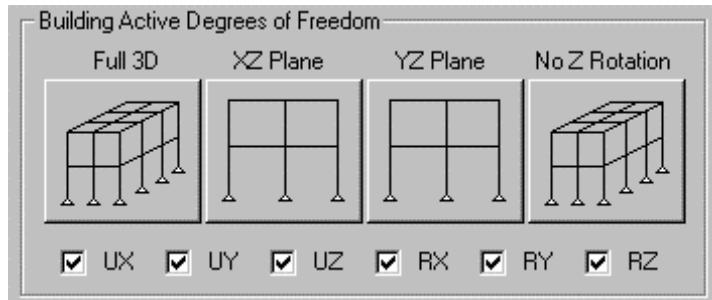
Analysis Options

Click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options dialog box where you can set various parameters for your analysis. In this dialog box you can specify parameters for building active degrees of freedom, dynamic analysis and P-Delta analysis. Each of these items is discussed in subsections below.

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 dialog box (shown below) you specify which of these degrees of freedom are active for your model. A check in the box associated with a degree of freedom

means that degree of freedom is 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.
- **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. Oftentimes, 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 to this they run analyses 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 these analyses. This No Z Rotation feature sets the degrees of freedom of your model appropriately to run an analysis with Z-rotations locked.



Tip:

The degree of freedom buttons provide a fast and easy way to set the building active degrees of freedom for your analysis.

Dynamic Analysis Parameters

To set the dynamic analysis parameters click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options dialog box. Then 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 dialog box. The following bullet items discuss the various areas in this dialog box:

- **Number of Modes:** In this box specify the number of Eigen or Ritz modes that you want ETABS to capture. For more information see the subsections titled "Number of Modes" in Chapter 33. Note that there are two subsections with this title in Chapter 33. One is under the section titled "Eigenvector Analysis" and the other under the section titled "Ritz-Vector Analysis."
- **Type of Analysis:** Choose either eigenvector or ritz-vector analysis in this area. See the sections titled "Eigenvector Analysis" and "Ritz-Vector Analysis" in Chapter 33 for discussion of these types of analyses.

If you are running response spectrum or time history analysis then 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 dialog box 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 . See the subsection titled "Frequency Range" in Chapter 33 for more information.
 - ✓ **Cutoff Frequency (Radius):** This is the radius of the cyclic frequency range, also known as the cutoff frequency, f_{max} . See the subsection titled "Frequency Range" in Chapter 33 for more information.



Tip:

*If you are running response spectrum or time history analysis then we **strongly recommend** that you use ritz-vectors. It is **especially important** that you use ritz-vectors when performing nonlinear time history analysis.*

- ✓ **Relative Tolerance:** This is the relative convergence tolerance, \mathcal{E} . See the subsection titled "Convergence Tolerance" in Chapter 33 for more information.
- ✓ **Include Residual Mass Modes:** If you check this box then ETABS computes residual-mass (missing-mass modes). The purpose of these modes is to approximate high-frequency behavior when the mass participation ratio for a given direction of acceleration load is less than 100%. See the subsection titled "Residual Mass Modes" in Chapter 33 for more information.

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 dialog box is only active if you select "Ritz Vectors" in the Type of Analysis area. In this area you specify the starting ritz vectors. Refer to the section titled "Ritz-Vector Analysis" and particularly to the subsection titled "Starting Load Vectors" in Chapter 33 for more information. Also refer to the section titled "Acceleration Loads" in Chapter 33.

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. The acceleration loads are designated as m_{ux} , m_{uy} , and m_{uz} in the section titled "Acceleration Loads" in Chapter 33. You can use any of these 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 dialog box. If the load is in the List of Loads list box then 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 both link properties are assigned and the Ritz Vector analysis type is specified.

If you check the Include Nonlinear Link Vectors box then ETABS 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 ETABS to capture the modes associated with these special starting vectors. ETABS does not add additional modes to the number you requested when you check the Include Nonlinear Link Vectors check box.

P-Delta Analysis Parameters

To set the P-Delta analysis parameters click the **Analyze menu > Set Analysis Options** command to bring up the Analysis Options dialog box. 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 dialog box. For more information on P-Delta analysis, including information relevant to this dialog box, see the section titled "Initial P-Delta Analysis" in Chapter 33.

The following bullet items discuss the various areas in this dialog box:

- **Method:** Initial P-Delta analysis in ETABS 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

**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.

15

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 though it also works for other cases as well.

The reason we provide this method is to allow 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, then in general, we recommend that you use the Iterative Based on Load Cases option.

- ✓ **Iterative Based on Load Cases:** 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 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 dialog box. The Maximum Iterations item and the Relative Tolerance -Displacements item are discussed in the subsection titled "Iterative Solution" in Chapter 33. Note that the maximum number of iterations specified is the maximum number of additional analyses after the first analysis is run.

- **P-Delta Load Combination:** This area is active if you select the Iterative Based on Load Cases option in the Method area of the dialog box. Here you specify the single load combination to be used for the initial P-Delta analysis of the structure.

As an example, suppose 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

15

For this case, the P-Delta effect due to the 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 since there is no lateral load.

It is also possible to accurately account for the P-Delta effect due to the deformation of the members between their ends in the ETABS 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 ETABS design postprocessors assume this is what you have done and includes these factors, where appropriate, in the design.

If you did want to try and account for the P-Delta effect due to the deformation of the members between their ends in the ETABS analysis then you should first break up all of your columns into at least two objects between story levels. Then you should run each of the six load cases above separately with a different P-Delta load combination for each. Again, it is recommended that this effect be accounted for instead by using factors in your design as is done in the ETABS design postprocessors.

Run Analysis

You can run an analysis of your building either by clicking the **Analyze menu > Run Analysis** command, or by clicking the **Run Analysis** button, , on the main (top) toolbar, or by pressing the F5 function key on your keyboard. When you execute this command using one of the above methods the Run Options dialog box appears with three choices. Those choices are:

- **Run:** This option opens the Analysis Window on top of the main ETABS window and runs the analysis. Information concerning the analysis scrolls by in the Analysis Window as the run progresses. See the subsection below titled "Analysis Window" for more information.

This option runs your analysis in such a way that if you switch away from ETABS to another program while the analysis is running you may not be able to switch back to ETABS until the analysis completes. Also, there is no way to cancel an analysis run once 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 closes the main ETABS window and runs with just the Analysis Window open. Information concerning the analysis run scrolls by in the Analysis Window as the run progresses. See the subsection below titled "Analysis Window" for more information.



Tip:

The **Run Minimized** option has the advantage of providing a **Cancel** button while the analysis is running that allows you to easily abort the analysis at any time.

This option runs your analysis in such a way that if you switch away from ETABS to another program while the analysis is running you *are* able to switch back to ETABS 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.

Analyze Window

As your ETABS 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 you should 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 this information disappears. However, it is saved, in text form, in the .log file. See the section titled "The ETABS Log File" in Chapter 43 for information on this file.

Run Static Nonlinear Analysis

The **Analyze menu > Run Static Nonlinear Analysis** command runs a static nonlinear analysis. For this command to be available you must have previously done the following:

- Run a regular static analysis (linear) of the building.
- Use the **Define menu > Static Nonlinear/Pushover Cases** command to define one or more static nonlinear load cases.

Note that you can run a static nonlinear analysis and then make changes to your hinge definitions or your static nonlinear load case definitions without having to unlock your model. This allows you to run several different static nonlinear analyses without having to rerun the regular static analysis (linear) each time.



Chapter 16

The ETABS Display Menu

16

General

The Display menu in ETABS provides options for displaying input and output information both on the model and onscreen in a tabular form. This chapter discusses those options.

Undeformed Shape

Clicking the **Display menu > Show Undeformed Shape** command or the **Show Undeformed Shape** button, on the main (top) toolbar 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 it clears the display of the results and returns the view to an undeformed shape view.

Note that this command only affects the active window.

Loads

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 discussed in a separate subsection below. The loads that you specify are only displayed in the currently active window.

16

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.

Joint/Point Loads

Click the **Display menu > Show Loads > Joint/Point** command to bring up the Show Joint/Point Loads dialog box. The following bullet items discuss the various areas in this dialog box.



Tip:

You can right click on any object and select the Loads tab as an alternate way of viewing the loads on an object.

- **Load case:** Choose the static load case whose joint/point loads you want to display from the drop-down box. Note that static load cases are defined using the **Define menu > Static Load Cases** command.
- **Load type:** Choose the type of load that you want to display from this area. The choices are forces, displacements or temperature values. You can display one of these types of loads at a time.

- **Show loading values:** When the Show Loading Values check box is *unchecked* then forces and displacements are indicated by arrows in the appropriate direction only. When the Show Loading Values check box is *checked* then 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.

Frame/Line Loads

Click the **Display menu > Show Loads > Frame/Line** command to bring up the Show Frame/Line Loads dialog box. The following bullet items discuss the various areas in this dialog box.

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 case:** Choose the static load case whose frame/line loads you want to display from the drop-down box. Note that static load cases are defined using the **Define menu > Static Load Cases** command.
- **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. 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. These 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 then 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 then 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 this 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* then forces and moments are indicated by arrows in the appropriate direction only. When the Show Loading Values check box is *checked* then 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.

Shell/Area Loads

Click the **Display menu > Show Loads > Shell/Area** command to bring up the Show Shell/Area Loads dialog box. The following bullet items discuss the various areas in this dialog box.

- **Load case:** Choose the static load case whose shell/area loads you want to display from the drop-down box. Note that static load cases are defined using the **Define menu > Static Load Cases** command.
- **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. The choices are:
 - ✓ **Uniform load values:** This is a uniform surface load on an area object. If you choose this option you then also specify the loading direction for which you want the loads displayed by choosing a direction from the associated drop-down box named Direction.
 - ✓ **Temperature values:** This includes all of the temperature loads applied to the area object.

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.

Important Note: When temperature loads are displayed numbers are shown at each corner of the area object. These 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, then the displayed temperatures for the area 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 corners of the area object.

If upon assigning the temperature load you indicated that the effects of point temperature *were* to be included then 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 this case the displayed temperatures for an area object may be different at each corner.

Input Table Mode

Click the **Display menu > Set Input Table Mode** command to bring up the Print Input Tables dialog box. Here you can specify the types of input data that you want to display in a tabular, database form on the screen. See Chapter 40 for documentation of the items that can be tabulated from this dialog box.

16



The items you specify in the Print Input Tables dialog box are displayed in a tabular database format. Select the type of input data that you want to view from the drop-down box. A table for that data appears. When the table is longer than the dialog box is deep two methods exist for scrolling through the table. You can use the scrollbar that appears at the right side of the table or you can use the arrow buttons that are in the dialog box.

The arrow buttons are shown in the sketch to the left where they are labeled 1, 2, 3 and 4 for reference. Arrow button 1 jumps you up to the top of the table. Arrow button 2 takes you up one line in the table. Note that there is an arrow to the left of the table indicating the current line. Arrow button 3 takes you down one line in the table. Arrow button 4 jumps you down to the bottom of the table.

When you are finished viewing a table either click the drop-down box to view another table or click the **OK** button to close the database.

You can not directly copy or print any of the information in the database tables. However, note that the items that can be tabulated on the screen using the **Display menu > Set Input Table Mode** command are exactly the same as the items that can be

printed to a printer or to a file using the **File menu > Print Tables > Input** command.

Deformed Shape

Click the **Display menu > Show Deformed Shape** command or the **Display Static Deformed Shape** button, , on the main (top) toolbar to bring up the Deformed Shape dialog box. Here you can specify the load case whose deformed shape you would like to plot. The following bullet items describe the various areas in the Deformed Shape dialog box.

- **Load:** Choose the load case whose deformed shape you would like to plot from the drop-down box. 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 you can 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 this 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 which 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 you can 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 then the first step of the time history is displayed. If you specify a time that is after the time history finishes then 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 then 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 you can 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 dialog box 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, [\ll], to display the previous step in the analysis and the right arrow key on the status bar, [\gg], 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 titled "Output Colors" in Chapter 18 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. See the section titled "Load Combinations" in Chapter 27 for discussion of load combinations.

If the load combination is a single-valued load combination then the displayed results are meaningful. If the load combination is multi-valued then the displayed results have little meaning. In this 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 then 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 funny looking deformed shape plots.

When a deformed shape is displayed for a load combination you can use the left and right arrow keys on the status bar, [\ll][\gg], to quickly display deformed shapes for other load combinations.

- **Scaling:** Here you can specify the scaling that is used to scale the plotted deformations. If you specify a scaling factor of 100 then 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 ETABS automatically determines a scaling factor for the deformed shape plot. If you want ETABS to automatically determine a scaling factor then leave the Auto option selected. Otherwise select the Scale Factor option and specify your own scale factor.

ETABS 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 then specify your own factor.

- **Options:** Here you can 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.

16

Note:

The cubic curve feature only affects line objects with frame section properties.



ETABS 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 then the frame elements are simply drawn as straight lines connecting the appropriate points/joints.

If you specify that cubic curves are to be considered then ETABS 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 done in ETABS, 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 you can right click on any point object to bring up the Joint Displacements dialog box that displays the displacements (translation and rotations) for that point object in the global coordinate system and in the current units. In this dialog box you can 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 ETABS.

16

Important Note: When an analysis is run ETABS 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 in order 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 these center of mass points are zero since 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 dialog box to toggle these center of mass joints on and off. You can use the **Set Building View Options** button on the main (top) toolbar to access the dialog box.

Finally, when a deformed shape is displayed you can 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 ETABS includes sound with the animation. You can use the **Options menu > Sound** command to toggle this sound on and off.

Mode Shape

Click the **Display menu > Show Mode Shape** command or the **Show Mode Shape** button, , on the main (top) toolbar to bring up the Mode Shape dialog box. Here you can specify the mode whose deformed shape you would like to plot. The following bullet items describe the various areas in the Mode Shape dialog box.

- **Mode number:** Here you can specify the mode number whose deformed shape you would like to plot. 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 then ETABS defaults to the highest mode number in the analysis.

- **Scaling:** Here you can specify the scaling that is used to scale the plotted mode shapes. By default ETABS automatically determines the scaling factor for the mode shape plot. If you want ETABS to automatically determine a scaling factor then 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.

ETABS calculates the default deformation (scale factor of 1 deformation) 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 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 then specify your own factor.

- **Options :** Refer to the bullet item titled "Options" in the previous section titled "Deformed Shape" for discussion of the cubic curve item.

When a mode shape is displayed you can 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 ETABS 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 you can 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.



Note:

You can animate mode shapes in ETABS.

Member Force and Stress Diagrams

Click the **Display menu > Show Member Forces/Stress Diagram** command or click the **Display Member Force Diagram** button,  , on the main (top) toolbar to display support and spring reactions; frame element, pier and spandrel forces, shell forces and stresses and link element forces. These items are discussed in detail in the subsections below.

Support and Spring Reactions

You can display support and spring reactions directly on your ETABS model. To do this click the **Display menu > Show Member Forces/Stress Diagram > Support/Spring Reactions** command to bring up the Point Object Reaction Forces dialog box. The following bullet items discuss the areas in this dialog box:

- **Load:** Choose the load case whose support or spring reactions you would like to display from the drop-down box. 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 you also specify a time for which you want to see the reactions. For static nonlinear cases you also specify a step at which you want to see the reactions.

Following is a description of what is plotted for each of these items.

- ✓ **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 you can 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 you can 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 then the first step of the time history is displayed. If you specify a time that is after the time history finishes then 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 then 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 you can 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 dialog box 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 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 the support or spring reactions for the entire push-over 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).

See the section titled "Load Combinations" in Chapter 27 for discussion of single-valued and multi-valued load combinations. If a load combination is a single-valued load combination then its reaction values are displayed. If a load combination is multi-valued then 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 then the -3 value is displayed. If an adj-

cent 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 you can use the left and right arrow keys on the status bar, , to quickly display reactions for other load combinations.

- **Type:** Here you indicate whether to display support reactions or spring forces.

When support or spring reactions are displayed you can 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.

Frame Element, Pier and Spandrel Forces

16

You can display column, beam, brace, pier and spandrel forces directly on your ETABS model. To do this click the **Display menu > Show Member Forces/Stress Diagram > Frame/Pier/Spandrel Forces** command to bring up the Member Force Diagram dialog box. The following bullet items discuss the areas in this dialog box:

- **Load:** Choose the load case whose member forces you would like to display from the drop-down box. 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 you also specify a time for which you want to see the forces. For static nonlinear cases you also specify a step at which you want to see the forces.
- **Component:** Here you 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 these types of objects then you can make the other objects invisible using the View menu > Set Building View Options command.

16

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 ETABS automatically determines a scaling factor for the plotted forces. If you want ETABS to automatically determine a scaling factor then leave the Auto option selected. Otherwise select the Scale Factor option and specify your own scale factor.

ETABS 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 then 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.

**Tip:**

When forces are displayed on the model you can 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 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 first before you can check the Fill Diagram check box.
- ✓ **To display force diagrams unfilled with no text values:** Leave both diagrams unchecked.
- ✓ **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 first 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 you can 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) then the arrow keys step you through all of the other static load cases (linear).
 - ✓ If you are currently viewing a response spectrum case then 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 then 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 then the arrow keys take you through all of the other steps in the static nonlinear case.
- ✓ If you are currently viewing a load combination then 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 "Output Colors" in Chapter 18 for more information.
- For frame elements exact force values are plotted at all output station locations. These exact force values are then connected by straight-line segments to complete the force diagram. See the section titled "Frame Output Station Assignments to Line Objects" in Chapter 14 for more information.
- For wall pier and spandrel elements exact force values are plotted at the ends of the element. These 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.

See Chapter 35 for discussion of frame element output conventions. See Chapter 38 for discussion of wall pier and spandrel output conventions.

Shell Forces and Stresses

You can display internal shell element forces and stresses directly on your ETABS model. To do this click the **Display menu > Show Member Forces/Stress Diagram > Shell Stresses/Forces** command to bring up the Element Force/Stress Contours for Shells dialog box.

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). See Chapter 36 for additional information.

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. See Chapter 36 for more information.



The following subsections discuss the areas in the Element Force/Stress Contours for Shells dialog box:

Load

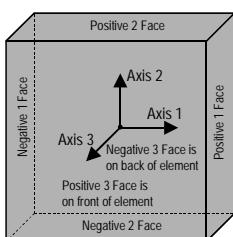
Choose the load case whose shell element forces or stresses you would like to display from the drop-down box. Note that you can plot shell element forces or stresses for any static load case, response spectrum case, time history case, static nonlinear case, or load combination. For time history cases you also specify a time for which you want to see the forces or stresses. For static nonlinear cases you also specify a step at which you want to see the forces or stresses.

Component Type

Here you specify whether you want to see the shell element internal forces or the internal stresses. See Chapter 36 for more information.

Component

Here you specify which component of force or stress you would like to see. See Chapter 36 for a complete description of these components. For shell element internal forces the possible components are:

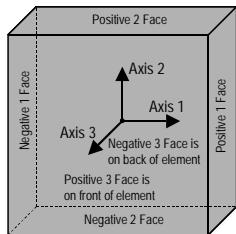


- **F11:** Direct force per unit length acting at the midsurface of the element on the positive and negative 1 faces in the 1-axis direction.
- **F22:** Direct force per unit length acting at the midsurface of the element on the positive and negative 2 faces in the 2-axis direction.

- **F12:** Shearing force per unit length acting at the midsurface 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 midsurface 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 midsurface 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 mid-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 midsurface 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 midsurface 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 midsurface 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 midsurface 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 midsurface 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 midsurface 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.

For shell element internal *stresses* the possible components are:



- **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. You can 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 dialog box.

In the Contour Range area of the Element Force/Stress Contours for Shells dialog box you can specify two values. They are:

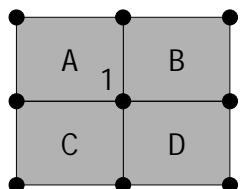
- **Min:** Any element with a force or stress less than the value shown here is displayed in the color associated with Min in the Contours area of the Assign Output Colors dialog box. Note that the color associated with Min is the top color in the dialog box.
- **Max:** Any element with a force or stress 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 dialog box. Note that the color associated with Max is the bottom color in the dialog box.

When you specify the Min and the Max values ETABS equal spaces the intermediate range values between the specified Min and Max values.

If you set both the Min and the Max values to zero then this tells ETABS to create its own range. In this case ETABS creates a stress range with rounded off (even) values that the actual maximum and minimum stresses just fit within. Note that setting Min and max to zero is the default.

Stress Averaging

Stress Averaging: Here you specify 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. These 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 these values become.

If the force or stress contours are plotted with no stress averaging at the common points then you will typically see abrupt changes in force or stress from element to element. Stress averaging tends to get rid of these abrupt changes in the plot and smoothes the contours out.

ETABS 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 ETABS 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 underlined portion of the previous paragraph. Suppose you are looking at stresses in a location where a wall intersects a floor. Further suppose 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, then 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, then 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.

In ETABS you have the option of having no stress averaging, stress averaging at all joints or stress averaging at specific points you have selected just prior to plotting the shell forces or stresses.

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 then you will have to view them in a 3D view.

Finally, when shell element forces and stresses are plotted for multi-valued load combinations, ETABS displays whichever of the maximum and minimum values has the largest absolute value.

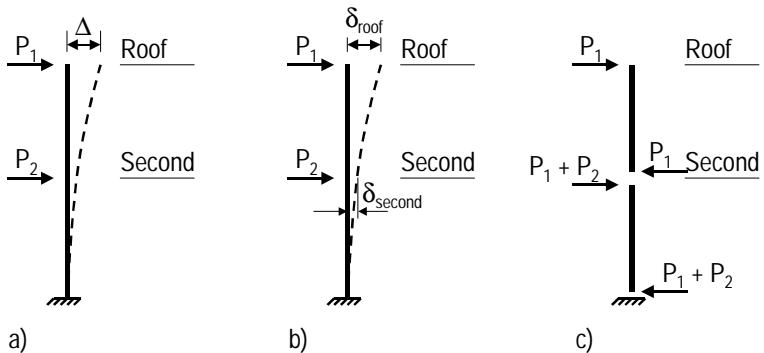
Link Element Forces

You can display link forces directly on your ETABS model. To do this click the **Display menu > Show Member Forces/Stress Diagram > Link Forces** command to bring up the Member Force Diagram dialog box. This dialog box works as described in the previous subsection titled "Frame Element, Pier and Spandrel Forces."

See Chapter 37 for discussion of link element output conventions. 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, ETABS displays whichever

Figure 16-1:
Example of the theory used for energy diagrams



of the maximum and minimum values has the largest absolute value.

Energy Diagram

16

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.

Click the **Display menu > Show Energy Diagram** command to display energy diagrams which 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.

Consider the two story structure shown in Figure 16-1a 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 dashed.

Now consider the same structure, shown in Figure 16-1b, with a single load P (typically a unit load) applied to it and a resulting displaced shape, δ , shown dashed. Maxwell's Reciprocal Theorem states that:

$$\mathbf{P}\Delta = \mathbf{P}_1\delta_{\text{roof}} + \mathbf{P}_2\delta_{\text{second}} \quad \text{Eqn. 16-1}$$

See a structural analysis textbook for details on Maxwell's Reciprocal Theorem.

In this very simple example, Equation 16-1 could be reduced to an element level where the elements are illustrated in Figure 16-1c as shown in Equation 16-2.

$$\begin{aligned} P\Delta = & [P_1\delta_{\text{roof}} - P_1\delta_{\text{second}}] + \\ & [(P_1 + P_2)\delta_{\text{second}} - (P_1 + P_2)\delta_{\text{base}}] \end{aligned} \quad \text{Eqn. 16-2}$$

Noting that δ_{base} is equal to zero, Equation 16-2 reduces to that shown in Equation 16-3.

$$P\Delta = [P_1\delta_{\text{roof}} - P_1\delta_{\text{second}}] + [(P_1 + P_2)\delta_{\text{second}}] \quad \text{Eqn. 16-3}$$

In Equation 16-3 the first term in brackets is the energy in the top element and the second term is the energy in the bottom element. The energy in both of these elements sums to the total $P\Delta$ energy.

When you request that ETABS show the energy diagram it reports the equivalent of the values shown in brackets in Equation 16-3 for each element in the structure. Note the following about the energy values that ETABS reports:

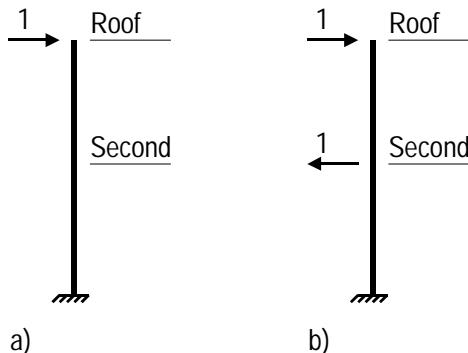
- The energy values that ETABS reports are based on all six degrees of freedom of the element, not just the one degree of freedom discussed in Figure 16-1 and Equations 16-1 through 16-3.
- The energy values that ETABS reports are determined as follows:
 - ✓ ETABS determines the energy per unit volume associated with each element in the structure.
 - ✓ ETABS normalizes all of the calculated energy values such that the largest one has a value of 100.

Note:

*Energy dia-
grams are nor-
malized.*

As previously stated the energy diagrams are helpful as an aid to determine which elements should be stiffened to control lateral displacements in your structure. When you request that ETABS show an energy diagram, the Energy Diagram dialog box appears. In this dialog box you input a load case associated with forces and one associated with displacements.

Figure 16-2:
Example displacement load cases for energy diagram



The load case associated with forces is the load case for which you want to control displacements. In the discussion above it is the load case shown in Figure 16-1a. The load case associated with displacements is the one associated with Figure 16-1b in the example above. Typically this load case consists of one or more unit loads.

16

Figure 16-2 shows a couple of load cases you might use for your displacement load cases. Figure 16-2a shows a load case that is appropriate if you are interested in controlling the roof displacement. Figure 16-2b shows a load case that is appropriate if you are interested in controlling the interstory displacement between the roof and the second story level.

Response Spectrum Curves

After running a time history analysis select one or more point objects and click the **Display menu > Show Response Spectrum Curves** command to bring up the Response Spectrum Generation dialog box. Here you can 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 dialog box has five separate tabs in it. The following subsections discuss each of those tabs.

Define Tab

The following bullet items discuss the three areas on the Define tab of the Response Spectrum Generation dialog box.

- **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

The following bullet items discuss the three areas on the Axes tab of the Response Spectrum Generation dialog box.

- **Abscissa:** This is the horizontal axis of the response spectrum. It can either 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 16-4a and 16-4b, respectively.

$$\text{PSV} = \frac{2\pi}{T} \text{ SD} \quad \text{Eqn. 16-4a}$$

$$\text{PSA} = \left(\frac{2\pi}{T} \right)^2 \text{ SD} \quad \text{Eqn. 16-4b}$$

Options Tab

The following bullet items discuss the two areas and the check box on the Options tab of the Response Spectrum Generation dialog box.

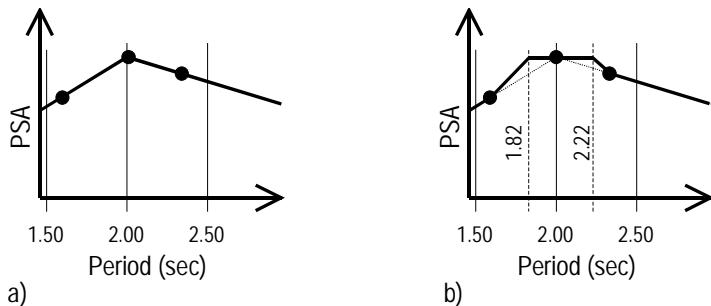
- **Abscissa:** This is the horizontal axis of the response spectrum. It can be plotted with either an arithmetic scale or a log scale.

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 16-3a. 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 this case ETABS artificially widens the peak such that there is a constant peak value from 1.82 seconds to 2.22 seconds. The response spectrum with spectrum widening is shown in Figure 16-3b. The calculations are shown in the figure.

Figure 16-3:
Example of spectrum
widening



Calculation for widened spectrum range shown in (b)

$$T = 2.00 \text{ sec, therefore } f = 1 / T = 1 / 2.00 = 0.5 \text{ Hz}$$

$$\text{Calculate 10\% spectrum widening frequency: } 0.10f = 0.10 * 0.5 = 0.05 \text{ Hz}$$

$$\text{Calculate frequency range: } f_{\max} = 0.5 + 0.05 = 0.55 \text{ Hz, } f_{\min} = 0.5 - 0.05 = 0.45 \text{ Hz}$$

$$\text{Calculate period range: } T_{\min} = 1 / f_{\max} = 1 / 0.55 = 1.82 \text{ sec, } T_{\max} = 1 / f_{\min} = 1 / 0.45 = 2.22 \text{ sec}$$

- **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/second². 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 discuss 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 ETABS for your building.
- ✓ **User:** The user periods and frequencies are those specified by you in the User Frequencies area of the tab.
- **User Frequencies/Periods:** You specify user frequencies or periods in this area. 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.

16

Damping Tab

In the damping area you can 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.

Time History Traces



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.

16

After running a time history analysis you can click the **Display menu > Show Time History Traces** command to bring up the Time History Display Definition dialog box. Here you can 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.

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 dialog box and used in this discussion 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, ETABS automatically creates time history display functions for all selected objects. Once inside the Time History Display Definition dialog box 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. ETABS creates the time history display function automatically for the point. You can easily modify the component of displacement displayed if ETABS did not default to the one you want.

The following bullet items discuss various items in the Time History Display Definition dialog box:

- **Time history case:** This is the name of the time history case for which you want to display a time history trace.
- **Choose functions:** In this area you 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.

The list box titled List of Functions lists all of the currently defined time history functions. You can highlight a function in this list box and click the **Add** button to move the function into the Vertical Functions list box.

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.

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.

You can highlight a function in either the List of Functions list box or the Vertical Functions list box and click the **Modify>Show** button to adjust the components that are plotted for that function. For example you may want to plot displacements in the Y direction instead of the X direction. The **Modify>Show** button allows you to make this adjustment.

- **Modify List of Functions button:** This button allows you to add functions to the list of functions or delete them. It also gives you an alternate path, and slightly longer path, for modifying the components that are plotted for a function.

Most of the definitions and component information you encounter here is self-explanatory. Standard energy formulas are used for the ETABS energy measures. All energies are integrated over the full structure. Following are equations describing each type of energy available in ETABS that can be plotted as a time history function:

$$IE = \int F(t) v(t) dt \quad \text{Eqn. 16-5a}$$

$$KE = \int m a(t) v(t) dt \quad \text{Eqn. 16-5b}$$

$$PE = \int k u(t) v(t) dt \quad \text{Eqn. 16-5c}$$

$$MDE = \int c v^2(t) dt \quad \text{Eqn. 16-5d}$$

$$NDE = \int [D_{\text{force}} v_{\text{avg}}(t) - D_{\text{stiff}} u(t) v(t)] dt \quad \text{Eqn. 16-5e}$$

$$LE = \int [L_{\text{force}} v_{\text{avg}}(t) - L_{\text{stiff}} u(t) v(t)] dt \quad \text{Eqn. 16-5f}$$

$$EE = IE - KE - PE - MDE - NDE - LE \quad \text{Eqn. 16-5g}$$

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.

- **Axes labels:** The axes labels that you type here appear on the screen plots and on printed plots.

- **Time range:** You can specify 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 then the horizontal axis range override overrides the specified Time Range.

For example, suppose you have 10 seconds of earthquake output. Further suppose 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 then 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.

Note:

The individual line options can be different for each vertical function

- **Individual 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.

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.

Static Pushover Curve

Click the **Display menu > Show Static Pushover Curve** command to display the force-displacement (pushover) curve obtained from a static nonlinear analysis. You can also view the pushover curve in the Acceleration-Displacement Response Spectrum (ADRS) format and overlay it with various response spectra thus allowing you to perform capacity-spectrum analysis in ETABS.

There is a File menu at the top of the window that displays the pushover curve. This File menu allows you to print graphics of the plot or to print tables either to a file or to a printer.

Documentation of ETABS static nonlinear analysis is beyond the scope of this manual.

Section Cut Forces

16

Click the **Display menu > Show Section Cut Forces** command to display section cut forces. This command brings up the Select Section Cuts dialog box where you can select one or more sections cuts whose forces you would like to see.

Note that section cuts are defined by first using the **Assign menu > Group Names** command to define a group that is used for defining the section cut and then using the **Define menu > Section Cuts** command to define the section cut itself. See the section titled "Section Cuts" in Chapter 26 and the section titled "Section Cuts" in Chapter 11 for more information.

When you click the **OK** button the section cut forces are displayed in a tabular form on the screen. The output sign convention used for section cut forces is described in Chapter 39. Note that the section cut forces are reported in the local coordinate system for the section cut. Also note that the location where the section cut forces are reported is output along with the force information.

Output Table Mode

Click the **Display menu > Set Output Table Mode** command or click the **Display Output Tables** button, , on the main (top) toolbar to bring up the Print Output Tables dialog box. Here you can specify the types of output data that you want to display in a tabular, database form on the screen. See Chapter 41 for documentation of the items that can be tabulated from this dialog box.

The items you specify in the Print Output Tables dialog box are displayed in a tabular database format. Select the type of output data that you want to view from the drop-down box. A table for that data appears. When the table is longer than the dialog box is deep two methods exist for scrolling through the table. You can use the scrollbar that appears at the right side of the table or you can use the arrow buttons that are in the dialog box.

16

1 2 3 4



The arrow buttons are shown in the sketch to the left where they are labeled 1, 2, 3 and 4 for reference. Arrow button 1 jumps you up to the top of the table. Arrow button 2 takes you up one line in the table. Note that there is an arrow to the left of the table indicating the current line. Arrow button 3 takes you down one line in the table. Arrow button 4 jumps you down to the bottom of the table.

When you are finished viewing a table either click the drop-down box to view another table or click the OK button to close the database.

You can not directly copy or print any of the information in the database tables. However, note that the items that can be tabulated on the screen using the **Display menu > Set Output Table Mode** command are exactly the same as the items that can be printed to a printer or to a file using the **File menu > Print Tables > Output** command.



Chapter 17

The ETABS Design Menu

17

Overview

The Design menu serves as your gateway to the integrated design postprocessors that are a part of your ETABS package. The design postprocessors available are:

Steel Frame Design - See Chapter 45

Concrete Frame Design - See Chapter 46

Composite Beam Design - See Chapter 47

Shear Wall Design - See Chapter 48

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 postprocessors.

The menu commands for each of the design postprocessors are discussed in Chapters 45 through 48 as referenced above.

Note that you use the **File menu > Print Tables** command to print design output from the various design postprocessors in a tabular form.

17

Overwrite Frame Design Procedure

Background

The frame design procedure identifies the design post processor that will design a particular element. The design procedure is reported in the Line Information dialog box which appears when you right click on a line object. See the section titled "Right Click Information for Line Objects" in Chapter 24 for more information on this.

The design procedure is always one of the following four items:

- Steel Frame Design
- Concrete Frame Design
- Composite Beam Design
- Null (no design)

The design procedure for a line object is automatically determined when the analysis is run. It is not necessarily determined when the line object is drawn or when you modify your model.

See the following section titled "The Overwrite Frame Design Procedure Command" for more information.

ETABS Default Design Procedure Assignments

The default design procedure assignments are determined by ETABS as follows:

- **Null:** If the line object is *not* assigned a frame section property then 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 then its default design procedure is Concrete Frame Design.
- **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 below then 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 Z-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.



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 then you will have to make all of the beams horizontal. You can easily do this using the Edit menu > Align Points/ Lines/Edges command and selecting the "Align to Z Coordinate of" option.

- **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 then its default design procedure is Steel Frame Design.

The Overwrite Frame Design Procedure Command

The **Design menu > Overwrite Frame Design Procedure** command allows you to change the design postprocessor for a frame element as follows:



Tip:

If you don't want a frame element to be designed then use the **Design menu > Overwrite Frame Design Procedure** command to assign it a Null (no design) design procedure.

17

- 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 dialog box 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 Z-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 ETABS 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.

When you click the **Design menu > Overwrite Frame Design Procedure** command the Overwrite Frame Design Procedure dialog box 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 titled "ETABS Default Design Procedure Assignments" earlier in this chapter.



Tip:

You can 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.

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 then later right click on it the Design Procedure item in the Line Information dialog box 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 18

The ETABS Options Menu

18

General

The Options menu in ETABS provides you with control over some of the basic features of ETABS. 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 discusses all of the items on the Options menu except for the Preference items related to the design postprocessors. The design postprocessors are documented in separate manuals.

Preferences

The ETABS preferences control a variety of items that affect the look and feel of the program, the default behavior of the design postprocessors and how ETABS considers live load reduction.

Dimensions and Tolerances

The **Options menu > Preferences > Dimensions/Tolerances** command brings up the Preferences dialog box where you can specify preferences for various dimension and tolerance items. The following items can be specified:

 **Note:**

The auto merge tolerance is used internally by ETABS to determine such things as when two point objects are in the same location or when a point object falls on a line object.

- **Auto merge tolerance:** This item is used as a basic tolerance check in the model. It is entered in length units. The ETABS 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 then 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, then 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 associated **Snap to Invisible Grid** button on the side toolbar. This item is entered in length units. The ETABS default for this item is 48 inches in English units or 1 meter in metric units. See the subsection titled "ETABS Snap Options" in Chapter 12 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 ETABS default for this item is 48 inches in English units or 1 meter in metric units. See the section titled "The ETABS Nudge Feature" in Chapter 9 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 ETABS default for this item is 3 pixels.

- **Screen snap to tolerance:** When using the snap features in ETABS your mouse pointer must be within this distance of a snap location to snap to it. This item is entered in pixels. The ETABS default for this item 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 ETABS default for this item 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 ETABS default for this item is 4 pixels.

- **Maximum graphic font size:** The default text font size used in ETABS 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 ETABS default for this item is 12 points.



Note:

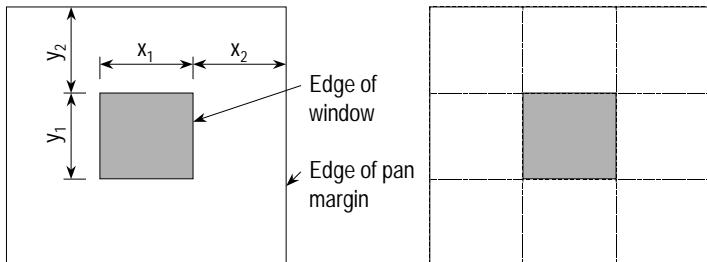
A pixel is the smallest graphic unit (dot) that can be displayed on the screen. A typical screen resolution is 1024 pixels by 768 pixels.



Note:

In printing a point is a unit of type equal to 0.01384 inch or approximately $\frac{1}{72}$ inch.

Figure 18-1:
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

This font size does not apply to the grid line identification labels whose size is determined by the specified size of the bubble.

- **Minimum graphic font size:** The default text font size used in ETABS is determined based on the average story height of your model. As you zoom out of your model the font 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 ETABS default for this item 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 whose size is determined by the specified size of the bubble.

- **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 ETABS default for this item is 50%. This is a recommended value.

See Figure 18-1 an example of the pan margin. In the figure the window is shown shaded. Figure 18-1a shows an example of 100% pan margin. Note that the dimension x_2 is equal to 100% of x_1 and similarly y_2 is equal to 100% of y_1 . Figure 18-1b illustrates that setting the pan margin to 100% allows you to potentially cover nine



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.

times more screen area than when the pan margin is set to 0%. If the pan margin is set to 0% you can not 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 ETABS model to run.

See the subsection titled "Pan Feature" under the section titled "Viewing Tools Available in ETABS" in Chapter 10 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 on the main (top) toolbar. This parameter is entered in percent. The magnification of all objects in a view are increased or decreased by this percent. The ETABS default for this item is 10%.

See the subsection titled "Zoom Features" under the section titled "Viewing Tools Available in ETABS" in Chapter 10 for additional information on the auto zoom step.

- **Shrink factor:** When you shrink the objects in a view using the **Shrink Object Toggle** button, , located on the main (top) toolbar all line objects are shrunk by this specified percentage. The ETABS default for this item is 70%. Note that you can also shrink objects by selecting the **View menu > Set Building View Options** command, or its associated button on the main toolbar, and checking the Object Shrink check box in the Special Effects area of the Set Building View Options dialog box.

Typically area objects are not shrunk by this percentage. Instead they are shrunk 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



Note:

The shrink factor typically only applies to line objects, not area objects.

has zero length) the area object reverts to using the percentage specified here.

The **Reset Defaults** button resets all of the dimension and tolerance values to their ETABS default values.

Output Decimals

Click the **Options menu > Preferences > Output Decimals** command to bring up the Preferences dialog box where you can specify preferences for the number of decimal places desired in the numeric output for various items. You can control the number of decimal places for the following types of items:

- **Displacements:** These are translational displacements which are reported in the current length units (e.g., inches). The ETABS default for displacements is four decimal places.
- **Rotations:** These are output rotations which are always reported in radians. The ETABS default for rotations is five decimal places.
- **Forces:** These are reported in the current force units (e.g., kips). The ETABS default for forces is two decimal places.
- **Moments:** These are reported in the current force-length units (e.g., kip-inch). The ETABS default for moments is three decimal places.
- **Forces per length:** These are reported in the current force/length units (e.g., kip/inch). The ETABS 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 ETABS default for moments per length is three decimal places.
- **Stresses:** These are reported in the current force/length² units (e.g., kip/inch² also known as ksi). The ETABS default for stresses is three decimal places.

- **Lengths:** These are reported in the current length units (e.g., inches). The ETABS default for lengths is three decimal places.
- **Rebar areas:** These are reported in the current length² units (e.g., inches²). The ETABS default for rebar areas is three decimal places.
- **Dimension line text:** Here you have the option 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 then you have the further option of reporting to either the nearest 1/8 inch or 1/16 inch. The ETABS 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. See Chapter 20 for discussion of the database units.

To modify any of the output decimal preferences simply enter the number of decimal places desired in the edit box next to each of these items in the Preferences dialog box.

The **Reset Defaults** button resets all of the output decimal values to their ETABS 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 then you can set your own output decimal preferences in a .edb file that you use for model initialization. Note that if you do this the **Reset Defaults** button resets the values to the built-in ETABS defaults, not the values that were in your initialization file. See the section titled "Starting a New Model" in Chapter 8 for more information on the initialization file.

Reinforcement Bar Sizes

Overview

You can define the reinforcing bar (rebar) name, diameter and area in the Reinforcing Bar Sizes dialog box. To access this dialog box click the **Options menu > Preferences > Reinforcement Bar Sizes** command.

The ETABS 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, you can add additional reinforcing bar definitions. You can also delete reinforcing bar definitions, including the default ones as long as they are not being used somewhere by the program.

The reinforcing bar data is used in the following locations of the program:

- The reinforcing bar data is used to define reinforcing steel in concrete column frame sections in the Reinforcement Data dialog box. To access this dialog box click the **Define menu > Frame Sections** command, highlight a concrete beam section (e.g., CSEC1), click the **Modify>Show Property** button, click the **Reinforcement** button and select the Column tab in the Reinforcement Data dialog box. In this instance ETABS uses the specified area of the bar.
- The reinforcing bar data is used to define reinforcing steel in the Section Designer utility. Section Designer is used to define wall pier sections and unusual frame sections. In this instance ETABS uses both the specified area of the bar and the diameter of the bar. The diameter is used to determine the location of the center of the bar when the face of the bar is aligned at a certain cover.

To access Section Designer for defining wall pier sections select a defined wall pier, click the **Design menu > Shear Wall Design > Assign Pier Sections for Checking** command, click the **Add Pier Section** button, fill in the Pier Section Data dialog box and click the **Sec-**

tion Designer button. See the Shear Wall Design Manual for more information.

To access Section Designer for frame sections click the **Define menu > Frame Sections** command, click Add SD Section in the Add drop-down box, fill in the SD Section Data dialog box and click the **OK** button. See the section titled "Adding Frame Section Properties using Section Designer" in Chapter 11 for more information.

Reinforcing Bar Sizes Dialog Box

The following bullet items describe how to modify the bar sizes shown in the Reinforcing Bar Sizes dialog box.

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 dialog box. 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 dialog box. 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 dialog box. Click the **OK** button.
- **To delete a bar:** Click on the bar ID that you want to delete in the Rebar area of the dialog box. 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 dialog box. Click the **OK** button.

The **Reset Defaults** button resets all of the area and diameter values for the ETABS default rebar to their default values. It also adds back in any default rebar sizes that you deleted. This button has no affect on other user-defined rebar you may have added.

Live Load Reduction

General

Note:

Live load reduction does not apply to floor and ramp-type area objects.

Click the **Options menu > Preferences > Live Load Reduction** command to bring up the Live Load Reduction Factor dialog box 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 titled "Static Load Cases" in Chapter 11 for more information. The following subsections describe the four areas in this dialog box. Do not overlook the upcoming subsection titled "Application Area in the Live Load Reduction Factor Dialog Box" that contains crucial information about the application of live load reduction in ETABS.

Important Note: ETABS 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 subsection below titled "Live Load Reduction Formulas" is always 1 for floor-type and ramp-type area objects.

Method Area in the Live Load Reduction Factor Dialog Box

In the Method area of the Live Load Reduction Factor dialog box you can choose the formula or method, if any, to be used for live load reduction. The available options for this are described in the following four subsections.

No Live Load Reduction

In this case no live load reduction is done even if you have defined static load cases as reducible live load type load cases.

Tributary Area Live Load Reduction

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 18-1.

$$\text{RLLF} = 1 - 0.8(A - 150)$$

Eqn. 18-1

where,

Tip:

You can overwrite the RLLF factor on an element by element basis in the design overwrites.

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 subsection below titled "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 Dialog Box."

Note that no check is done to limit the RLLF based on Equation 7-2 in Section 1607.5 of the 1997 UBC.

Influence Area Live Load Reduction

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 18-2.

$$\text{RLLF} = \left(0.25 + \frac{15}{\sqrt{A_I}} \right)$$

Eqn. 18-2

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 tribu-

trary area. The influence area for a beam, brace or wall is taken as two times the tributary area. See the subsection below titled "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 Dialog Box."

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 18-3.

$$\text{RLLF} = 1 - r (A - A_{\min})$$

Eqn. 18-3

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, $1/\text{length}^2$. The default value is 0.08 in $1/\text{ft}^2$ 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 Dialog Box."

Minimum Factor Area in the Live Load Reduction Factor Dialog Box

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 element with load from only one story level and 0.4 for elements with load from more than one story level.
- **Influence area method:** 0.5 for element with load from only one story level and 0.4 for elements with load from more than one story level.
- **User-defined method:** 0.6 for element with load from only one story level and 0.4 for elements with load from more than one story level.

Note:

Live loads are not reduced for basic analysis output. Reduced live loads are only used for determining design forces in the design postprocessors.

Application Area in the Live Load Reduction Factor Dialog Box

Important Note: In ETABS the live load is currently only reduced for design forces that are used in the ETABS design post-processors. 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.

The Application area of the Live Load Reduction Factor dialog box reinforces the above important note. The only application available in the area is Design Forces. You can not make any changes in this area. If you do not want live load reduction to apply to your design forces in the design postprocessors then select the No Live load Reduction option in the Method area of the dialog box.

**Note:**

For columns, the live load reduction can be specified to apply to the axial load only or to all force components.

Application to Columns Area in the Live Load Reduction Factor Dialog Box

For columns, the live load reduction can be specified to apply to the axial load only or to all force components. By default ETABS 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.

Tributary Area

ETABS calculates the tributary area for a frame, shell or link element from the floor-type and/or ramp-type area objects that load the element. If no floor-type and/or ramp-type area object loads a particular element then the tributary area for that element is calculated as zero by ETABS.

Note that live load reduction can be overwritten on an element by element basis in each of the design postprocessors.

Colors

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 discuss display colors and output colors.

Display Colors

Note:

The color of Null-type area and line objects is controlled by the Background item.

Note:

The color of the bounding plane line (cyan) is built into the program. You can not 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.

Click the **Options menu > Colors > Display** command to bring up the Assign Display Colors dialog box where you can set the display colors for various items in your model. The following bullet items discuss the areas in this dialog box.

- **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.
 - ✓ **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.
 - ✓ **Floor:** These are floor-type area objects. The color applies to the object edges and fill color.
 - ✓ **Ramp:** 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 dialog box.
 - ✓ **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 ID's. 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 titled "Editing Coordinate System Grid Line Data" in Chapter 9 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.
- **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.

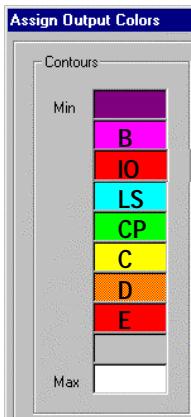
Note that you can click the **View menu > Set Building View Options** command, or the **Set Building View Options** button on the main (top) toolbar, to access the Set Building View Options dialog box and toggle the display of extruded shapes on or off.

- **Reset Defaults button:** This button resets the colors to the built-in ETABS 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.

Output Colors

Click the **Options menu > Colors > Output** command to bring up the Assign Output Colors dialog box where you can set the display colors for various output items. The following bullet items discuss the areas in this dialog box.

- **Contours:** In this area you can specify ten different colors which 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 displaying the deformed shape of a static nonlinear analysis. The color boxes used for this are shown in the sketch to the left. Documentation of nonlinear static analysis is beyond the scope of this manual.

- **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 postprocessor 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.

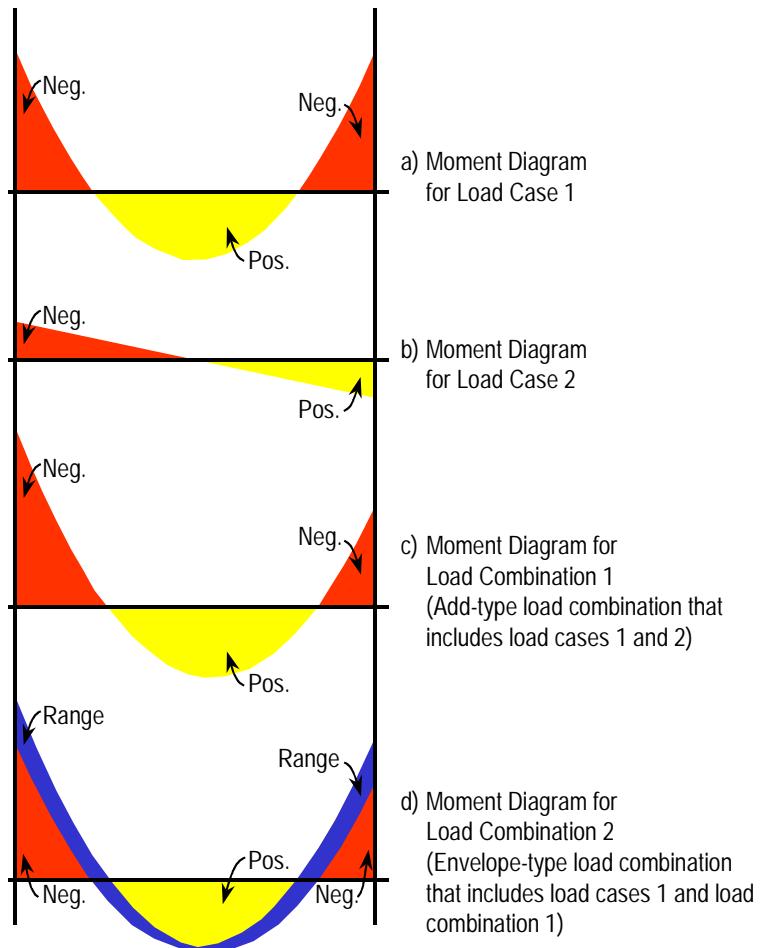
- ✓ Any frame element that does not have its Design Procedure the same as the design postprocessor that is currently displaying the results is shown in a color that is opposite of the background color. See the section titled "Right Click information for Line Objects" in Chapter 24 for discussion of the Design Procedure.
- ✓ Any frame element with the correct Design Procedure which 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 color 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 18-2. (Note that these diagrams are plotted with positive values on the tension side of the element.)

Figure 18-2a shows the moment diagram for a load case designated 1 which is gravity load. Figure 18-2b shows the moment diagram for a load case designated 2 which is a lateral load. Figure 18-2c shows the moment diagram for a load *combination* designated 1. This load combination is created as an add-type load combination

Figure 18-2:
Example of diagram
fill colors where the
diagrams are plotted
with positive values
on the tension side of
the element



and it adds the results of load case 1 and load case 2. See the subsection titled "Types of Load Combinations" in Chapter 27 for more information. Note how the fill colors are different for positive and negative moment in these moment diagrams.

Figure 18-2d 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 18-2d 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.
- **Reset Defaults button:** This button resets the colors to the built-in ETABS 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.

Other Option Items

This section describes each of the other option items that are available on the Options menu.

18

Windows

Tip:

You can display your model in from one to four windows. Each window can display a completely different view.

You can 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.

Startup Tips

When you first open ETABS 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 ETABS 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 ETABS window closes the Tip of the Day window. For example, as soon as you start the ETABS graphical interface you can immediately click on the file menu and the Startup Tips window closes and the File menu appears.

Bounding Plane

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 then a cyan line appears in the plan view showing the location of the elevation.

Table 18-1 lists the circumstances where the bounding plane (line) appears.

Table 18-1:
*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

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 18-2 earlier in this chapter for an example of moment diagrams plotted on the tension side of a member.

Sound

Click the **Options menu > Sound** command to toggle the sound produced by ETABS when it is displaying animation of deformed shapes and mode shapes on or off.

Lock Model

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 can not make any changes to it that will affect the analysis results except as noted below.

Exception: While the model is locked you can run one or more nonlinear static analyses.

Note:

ETABS automatically locks your model when you run an analysis.

When you run an analysis ETABS automatically locks the model. This is done to keep you from changing the model such that your analysis results are invalidated. After you have run an analysis, if you want to make changes to your model, then you must first unlock it. Unlocking the model deletes all of your analysis results. The results are deleted because once you make changes to the model they are no longer valid.

If you want to save analysis results and make changes to your model, then after you have run the analysis use the **File menu > Save As** command to save the file with a new name. You can then make changes to the file with the new name. The original file remains with its results.

Aerial View Window

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 titled "The ETABS Aerial View" in Chapter 4 for more information.

Floating Property Window

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 the sections titled "Floating Properties of Object Window for Line Objects" and "Floating Properties of Object Window for Area Objects" in Chapter 12 for more information.

Crosshairs

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.



Chapter 19

The ETABS Help Menu

The ETABS Help File

19

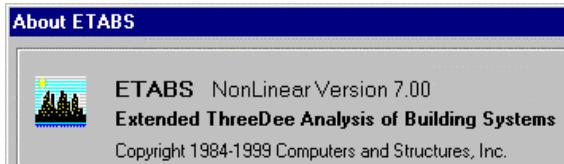
You can click on the **Help menu > Search for Help on** command at any time to open the ETABS help file and get detailed, searchable information about ETABS. One of the best ways to enter the help file is to press the F1 key on your keyboard.

If you are in a dialog box at the time you press the F1 key then you will jump immediately to context sensitive help for that dialog box. If you are not in a dialog box when the F1 key is pressed then the ETABS help file is opened just as it is when you click the **Help menu > Search for Help on** command.

About ETABS

Clicking on the **Help menu > About ETABS** command brings up the About ETABS dialog box that provides some useful and important information about ETABS and about your computer.

When you are calling, faxing or e-mailing us for technical support or for information on product upgrades you should *always* tell us the version of the program that you are using. You find this version number by Clicking on the **Help menu > About ETABS** command and reading the first (top) line on the left-hand side of the dialog box. It will say something like "ETABS Nonlinear Version x.xx" or "ETABS Plus Version x.xx." An example is shown below.



In addition, when you have technical support questions that may be related to the physical memory in your computer, or to your computer's operating system, you should always tell us about your computer's memory and operating system. You can find this information in the bottom of the left-hand side of the About ETABS dialog box. An example of this information is shown below.



See Chapter 3 for more information on technical support.



References

References

ASCE, 1995

R

Minimum Design Loads for Buildings and Other Structures - ASCE 7-95, American Society of Civil Engineers, New York, New York, 1995.

BOCA, 1996

The BOCA National Building Code/1996, Building Officials and Code Administrators International, Inc., Country Club Hills, Illinois, 1996.

CEN, 1994

ENV 1998-1-1:1994, Eurocode 8: Design Provisions for Earthquake Resistance of Structures - Part 1-1: General Rules- Seismic Actions and General Requirements for Structures, European Committee for Standardization, Brussels, Belgium, 1994.

R. W. Clough, I. P. King and E. L. Wilson, 1963

“Structural Analysis of Multistory Buildings,” *Journal of the Structural Division, ASCE*, Vol. 89, No. 8, 1963.

R. D. Cook, D. S. Malkus and M. E. Plesha, 1989

Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, New York, 1989.

A. K. Gupta, 1990

“Response Spectrum Method,” *Blackwell Scientific Publications, Ltd.*, 1990.

IBC, 1997

International Building Code 2000, International Code Council, Birmingham, Alabama, November, 1997.

NBCC, 1995

National Building Code of Canada, National Research Council of Canada, Ottawa, Canada, 1995.

NEHRP, 1997

NEHRP Recommended Provisions for Seismic Regulations for New Buildings and Other Structures (FEMA 302), Building Seismic Safety Council, Washington, D.C., 1997.

N. M. Newark and W. J. Hall, 1981

Earthquake Spectra and Design, Earthquake Engineering Research Institute, Berkeley, California, 1982.

NZS, 1992

Code of Practice for General Structural Design and Design Loadings for Buildings, Known as the Loadings Standard, Standards New Zealand, Wellington, New Zealand, 1992.

SEAOC, 1996

Recommended Lateral Force Requirements and Commentary, Structural Engineers Association of California, Sacramento, California, 1996.

UBC, 1994

Uniform Building Code, International Conference of Building Officials, Whittier, California, 1994.

UBC, 1997

Uniform Building Code, International Conference of Building Officials, Whittier, California, 1997.

D. W. White and J. F. Hajjar, 1991

“Application of Second-Order Elastic Analysis in LRFD: Research to Practice,” *Engineering Journal*, AISC, Vol. 28, No. 4, pp. 133–148.

E. L. Wilson, 1993

“An Efficient Method for the Base Isolation and Energy Dissipation Analysis of Structural Systems,” *ATC 17-1, Proceedings of Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control*, Applied Technology Council, Redwood City, California, 1993.

R

E. L. Wilson, 1997

Three Dimensional Dynamic Analysis of Structures with Emphasis on Earthquake Engineering, Computers and Structures, Berkeley, California, 1997.

E. L. Wilson and M. R. Button, 1982

“Three Dimensional Dynamic Analysis for Multicomponent Earthquake Spectra,” *Earthquake Engineering and Structural Dynamics*, Vol. 10.

E. L. Wilson, H. H. Dovey and A. Habibullah, 1981a

“Theoretical Basis for CTABS80: A Computer Program for Three-Dimensional Analysis of Building Systems,” *Technical Report K-81-2*, Computers/Structures International, Oakland, California, 1981.

E. L. Wilson and A. Ibrahimbegovic 1989

“Simple Numerical Algorithms for the Mode Superposition Analysis of Linear Structural Systems with Nonproportional Damping,” *Computers and Structures*, Vol. 33, No. 2, 1989.

E. L. Wilson, A. D. Kiureghian and E. Bayo, 1981b

“A Replacement for the SRSS Method in Seismic Analysis,” *Earthquake Engineering and Structural Dynamics*, Vol. 9, 1981.

E. L. Wilson and I. J. Tetsuji, 1983

“An Eigensolution Strategy for Large Systems,” *Computers and Structures*, Vol. 16.

E. L. Wilson, M. W. Yuan, and J. M. Dickens, 1982

“Dynamic Analysis by Direct Superposition of Ritz Vectors,” *Earthquake Engineering and Structural Dynamics*, Vol. 10, pp. 813–823.



Index

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

\$set file, 3-5, 4-12, 8-20, 8-29, 20-4
2D view, 10-5, 10-6
3D view, 10-2, 10-8

A

acceleration loads, 33-11
active degrees of freedom, 15-1
add to model from template, 9-7
additional mass
 area, 14-56
 line, 14-38
 point, 14-14
aerial view window, 4-8, 18-23
align points, lines and edges, 9-20, 9-29
Analyze menu commands
 Set Analysis Options, 15-1
 Run Analysis, 15-8
 Run Static Nonlinear Analysis, 15-9
analysis, types
 eigenvector, 33-3
 linear elastic static, 33-2
 linear time history, 33-14

modal, 33-2
nonlinear time history, 33-14, 33-17
nonlinear static, 33-22
p-delta, 33-18
periodic time history (linear), 33-14
response spectrum (linear), 33-12
ritz-vector, 33-8
analysis log file, 43-1
analysis sections, 45-1, 45-13, 46-2, 46-10, 47-1,
 47-12, 47-13
angle drawing constraint, 12-22
area object, 23-1
 assignments to, 14-48, 23-5
drawing, 12-9
right click information, 23-6
type, 23-2
Assign menu commands
Joint/Point
 Rigid Diaphragm, 14-2
 Panel Zone, 14-3
 Restraints (Supports), 14-9
 Point Springs, 14-10
 Link Properties, 14-13
 Additional Point Mass, 14-14

ETABS User's Manual – Volume 1

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

- Frame/Line
 - Frame Section, 14-22
 - Frame Releases/Partial Fixity, 14-23
 - Frame Rigid Offsets, 14-24
 - Frame Output Stations, 14-28
 - Local Axes, 14-29
 - Frame Property Modifiers, 14-31
 - Link Properties, 14-32
 - Frame Nonlinear Hinges, 14-32
 - Pier Label, 14-34
 - Spandrel Label, 14-35
 - Line Springs, 14-36
 - Additional Line Mass, 14-38
 - Automatic Frame Mesh/No Mesh, 14-39
- Shell/Area
 - Wall/Slab/Deck Section, 14-48
 - Opening, 14-49
 - Rigid Diaphragm, 14-49
 - Local Axes, 14-50
 - Shell Stiffness Modifiers, 14-51
 - Pier Label, 14-52
 - Spandrel Label, 14-53
 - Area Springs, 14-54
 - Additional Area Mass, 14-56
 - Automatic Membrane Floor Mesh/No Mesh, 14-57
- Joint/Point Loads
 - Force, 14-16
 - Ground Displacement, 14-18
 - Temperature, 14-20
- Frame/Line Loads
 - Point, 14-40
 - Distributed, 14-42
 - Temperature, 14-46
- Shell/Area Loads
 - Temperature, 14-60
 - Uniform Surface, 14-58
- Group Names, 14-63
- Clear Display of Assigns, 14-64
- auto merge tolerance (preference), 18-2
- auto relabel all, 23-5
- auto select list, 11-11, 24-1
- auto zoom step (preference), 18-5
- automatic seismic loads
 - 1994 UBC, 28-6
 - 1995 NBCC (Canadian), 28-21
 - 1996 BOCA, 28-17
 - 1997 NEHRP, 28-30
 - 1997 UBC, 28-10
 - 1997 UBC isolated, 28-14
 - IBC2000, 28-25
 - user defined, 28-36
- automatic wind loads
 - 1994 UBC, 29-6
 - 1995 NBCC (Canadian), 29-14
 - 1996 BOCA, 29-11
 - 1997 UBC, 29-8
 - ASCE 7-95, 29-16
 - user defined, 29-19
- avi file, 8-26
- axes, 10-15, 21-2

B

- beams, secondary, 8-10, 12-6
- black objects on white background, 10-18, 44-2
- bounding plane, 18-3, 18-21
- breaking (dividing, meshing) line objects, 9-35
- buckling, 33-20, 33-21

C

- Cancel button, 4-14
- center of rigidity, 41-12
- charts
 - response spectrum function, 11-29
 - response spectrum curve from time history results, 16-29
 - time history function, 11-38
 - time history trace, 16-34
- clear display of assignments, 14-64
- codes, building
 - 1992 NZS 4203 (New Zealand), 11-37
 - 1994 UBC, 11-34, 28-6, 29-6
 - 1995 NBCC (Canada), 11-35, 28-21, 29-14
 - 1996 BOCA, 11-35, 28-17, 29-11
 - 1997 NEHRP, 11-36, 28-30

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

1997 UBC, 11-34, 28-10, 28-14, 29-8
 1998 Eurocode 8, 11-37
 ASCE 7-95, 29-16
 IBC2000, 11-36, 28-25
 colors
 display colors, 18-15
 frame element steel stress ratio colors, 18-17
 object fill colors, 10-18, 18-15
 output colors, 18-17
 shell element stress contour colors, 18-17
 view model by element or group colors, 10-17
 comments, user, 8-29
 composite beam design process, 47-3
 concrete frame design process, 46-4
 constraints
 drawing, 12-21
 rigid diaphragm, 14-2, 14-49, 23-9, 23-16, 25-5
 control key (Ctrl), 4-9, 4-10
 cookie cut meshing tools, 31-3
 coordinate systems, 21-1
 copying geometry, 9-2
 create a new model, 4-11, 6-1, 8-1
 crosshairs, 18-23
 coupled springs, 14-12
 cumulative center of mass, 41-12
 current units, 20-3

D

damping
 in link elements, 11-27
 modal, 11-27, 11-50, 33-15
 database, Microsoft Access, 8-26, 42-1
 database, section properties, 11-7
 database units, 20-4
 decimal places (preference), 18-6
 deck span direction, 14-49, 32-14
 default.edb file, 6-5, 8-2
 Define menu commands
 Material Properties, 11-1
 Frame Sections, 11-6
 Wall/Slab/Deck Sections, 11-21
 Link Properties, 11-26
 Frame Nonlinear Hinge Properties, 11-27

Section Cuts, 11-27
 Response Spectrum Functions, 11-29
 Time History Functions, 11-38
 Static Load Cases, 11-46
 Response Spectrum Cases, 11-50
 Time History Cases, 11-56
 Static Nonlinear/Pushover Cases, 11-63
 Load Combinations, 11-63
 Mass Source, 11-64
 deformed shape, display, 16-7
 deleting objects, 9-7
 deleting a story level, 9-18
 deselect, 13-6
 Design menu commands
 Steel Frame Design
 Select Design Group, 45-7
 Select Design Combo, 45-8
 View/Revise Overwrites, 45-8
 Set Lateral Displacement Targets, 45-9
 Start Design/Check of Structure, 45-10
 Interactive Steel Frame Design, 45-11
 Display Design Info, 45-11
 Make Auto Select Section Null, 45-11
 Change Design Section, 45-12
 Reset Design Section to Last Analysis, 45-13
 Verify Analysis vs Design Section, 45-13
 Reset All Steel Overwrites, 45-14
 Delete Steel Design Results, 45-14
 Concrete Frame Design
 Select Design Combo, 46-7
 View/Revise Overwrites, 46-7
 Start Design/Check of Structure, 46-8
 Interactive Concrete Frame Design, 46-8
 Display Design Info, 46-9
 Change Design Section, 46-9
 Reset Design Section to Last Analysis, 46-10
 Verify Analysis vs Design Section, 46-10
 Reset All Concrete Overwrites, 46-11
 Delete Concrete Design Results, 46-11
 Composite Beam Design
 Select Design Group, 47-7
 Select Design Combo, 47-8
 View/Revise Overwrites, 47-9
 Start Design/Check of Structure, 47-10

ETABS User's Manual – Volume 1

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

- I Interactive Composite Beam Design, 47-10
- Display Design Info, 47-11
- Make Auto Select Section Null, 47-11
- Change Design Section, 47-11
- Reset Design Section to Last Analysis, 47-12
- Verify Analysis vs Design Section, 47-13
- Reset All Composite Beam Overwrites, 47-13
- Delete Composite Beam Design Results, 47-13
- Shear Wall Design
 - Select Design Combo, 48-11
 - View/Revise Pier Overwrites, 48-12
 - View/Revise Spandrel Overwrites, 48-15
 - Define Pier Sections for Checking, 48-17
 - Assign Pier Sections for Checking, 48-17
 - Start Design/Check of Structure, 48-17
 - Interactive Wall Design, 48-17
 - Display Design Info, 48-18
 - Reset All Pier/Spandrel Overwrites, 48-19
 - Delete Wall Design Results, 48-19
- Overwrite Frame Design Procedure, 17-2
- design postprocessors, 6-5, 6-6, 6-9, 17-1, 24-2
- design procedure, 17-2, 24-7
- design process
 - composite beam design, 47-3
 - concrete frame design, 46-4
 - shear wall design, 48-8
 - steel frame design, 45-3
- design sections, 45-1, 45-13, 46-2, 46-10, 47-1, 47-12, 47-13
- developed elevation, 6-6, 10-7, 10-9, 12-11, 12-12
- diaphragm, rigid
 - assign to area object, 14-49
 - assign to point objects, 14-2
 - display diaphragm extent, 10-29
- dimension lines, 9-40, 12-17
- dimensions, measurements, 10-15
- dimensions, preferences, 18-2
- displacement
 - ground (input static load), 14-18
 - deformed shape (static load), 16-7
- Display menu commands
 - Show Undeformed Shape, 16-1
- Show Loads
 - Joint/Point, 16-2
 - Frame/Line, 16-3
 - Shell/Area, 16-5
- Set Input Table Mode, 16-6
- Show Deformed Shape, 16-7
- Show Mode Shape, 16-12
- Show Member Forces/Stress Diagram
 - Support/Spring Reactions, 16-14
 - Frame/Pier/Spandrel Forces, 16-17
 - Shell Stresses/Forces, 16-20
 - Link Forces, 16-26
- Show Energy Diagram, 16-27
- Show Response Spectrum Curves, 16-29
- Show Time History Traces, 16-34
- Show Static Pushover Curve, 16-39
- Show Section Cut Forces, 16-39
- Set Output Table Mode, 16-40
- divide lines, 9-35
- Draw menu commands
 - Select Object, 12-1
 - Reshape Object, 9-37
 - Draw Point Objects, 12-3
 - Draw Line Objects
 - Draw Lines (Plan, Elev, 3D), 12-4
 - Create Lines in Region or at Clicks (Plan, Elev, 3D), 12-5
 - Create Columns in Region or at Clicks (Plan), 12-6
 - Create Secondary Beams in Region or at Clicks (Plan), 12-6
 - Create Braces in Region or at Clicks (Elev), 12-7
 - Draw Area Objects
 - Draw Areas (Plan, 3D), 12-9
 - Draw Rectangular Areas (Plan, Elev), 12-10
 - Create Areas at Click (Plan, Elev), 12-10
 - Draw Walls (Plan), 12-10
 - Create Walls in Region or at Click (Plan), 12-12
 - Draw Developed Elevation Definition, 12-12
 - Draw Dimension Line, 12-17
 - Snap to
 - Grid Intersections and Points, 12-18

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

Line Ends and Midpoints, 12-19
 Intersections, 12-19
 Perpendicular Projections, 12-19
 Lines and Edges, 12-19
 Fine Grid, 12-19
 Constrain Drawn Line to
 None, 12-22
 Constant X, 12-21
 Constant Y, 12-21
 Constant Z, 12-21
 Constant Angle, 12-22
 draw mode, 4-12
 dynamic load participation ratio, 41-8
 dxf export, 8-24, 8-25

E

e2k file, 3-5, 4-12, 8-21, 8-22, 8-29, 20-3
 ebk file, 20-4
 edb file, 3-5, 4-12, 8-2, 8-7, 8-19, 8-20, 8-22, 8-24, 20-4
 emf file, 8-26, 44-2
 earthquake load *See automatic seismic loads*
 edge select mode, 9-31
 Edit menu commands
 Undo, 4-14
 Redo, 4-14
 Cut, 9-2
 Copy, 9-2
 Paste, 9-2
 Delete, 9-7
 Add to Model from Template
 Add 2D Frame, 9-8
 Add 3D Frame, 9-8
 Replicate, 9-9
 Edit Grid Data, 9-14
 Edit Story Data
 Edit, 9-17
 Insert Story, 9-17
 Delete Story, 9-18
 Edit Reference Planes, 9-18
 Edit Reference Lines, 9-18
 Merge Points, 9-19
 Align Points/Lines/Edges, 9-20

Move Points/Lines/Areas, 9-29
 Expand/Shrink Areas, 9-30
 Merge Areas, 9-32
 Mesh Areas, 31-2
 Join Lines, 9-33
 Divide Lines, 9-35
 Auto Relabel All, 23-5
 eigenvalue, 33-3
 eigenvector analysis, 33-3
 elevation, developed, 10-7, 12-11, 12-12
 elevation, story level, 8-6, 9-17
 elevation view, 10-6
 energy diagram, 16-27
 export options
 Save Model as ETABS7 .e2k Text File, 8-22
 Save Model as SAP2000 .s2k Text File, 8-23
 Save Story as SAFE .f2k Text File, 8-23
 Save Story as ETABS7 .edb File, 8-24
 Save Story Plan as .DXF, 8-24
 Save as 3D .DXF, 8-25
 Save Input/Output as Access Database File, 8-26
 Save Graphics as Enhanced MetaFile, 8-26
 expand area objects, 9-30
 extend line objects, 9-25

F

fast restraints, 14-10
 fax number, 3-4
 File menu commands
 New Model, 8-1
 Open, 8-19
 Save, 8-20
 Save As, 8-20
 Import
 Open ETABS7 .e2k Text File, 8-21
 Open ETABS6 Text File, 8-21
 Overwrite Story from SAFE .f2k Text File, 8-22
 Overwrite Story from ETABS7 .edb File, 8-22
 Export
 Save Model as ETABS7 .e2k Text File, 8-22
 Save Model as SAP2000 .s2k Text File, 8-23

ETABS User's Manual – Volume 1

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

Save Story as SAFE .f2k Text File, 8-23
Save Story as ETABS7 .edb File, 8-24
Save Story Plan as .DXF, 8-24
Save as 3D .DXF, 8-25
Save Input/Output as Access Database File, 8-26
Save Graphics as Enhanced MetaFile, 8-26
Create Video
 Time History Animation, 8-26
 Cyclic Animation, 8-27
Print Preview for Graphics, 8-27
Print Graphics, 8-27
Print Tables
 Input, 8-28
 Analysis Output, 8-29
 Steel Frame Design, 8-28
 Concrete Frame Design, 8-28
 Composite Beam Design, 8-28
 Shear Wall Design, 8-28
User Comments and Session Log, 8-29
Display Input/Output Text Files, 8-29
Exit, 8-30
floating property window, 12-8, 12-12
font size, 18-3, 18-4
frame element internal forces, 35-2
frame section properties, 11-6, 14-22
frequency, 33-4, 41-3
functions, response spectrum
 1992 NZS 4203 (New Zealand), 11-37
 1994 UBC, 11-34
 1995 NBCC (Canada), 11-35
 1996 BOCA, 11-35
 1997 NEHRP, 11-36
 1997 UBC, 11-34
 1998 Eurocode 8, 11-37
 from text file, 11-30
 IBC2000, 11-36
 user-defined, 11-32
functions, time history
 cosine, 11-44
 from a file, 11-38
 ramp, 11-44
 sawtooth, 11-45
 sine, 11-43

triangular, 11-46
user-defined, 11-41
user-defined periodic, 11-41

G

getting started
 creating a model, 6-1, 8-1
 installing ETABS, 2-1
global force balance, 43-3
graphical user interface, 4-1
grid line systems, 9-14, 21-1
ground displacement, 14-18
groups, 14-63, 26-1
groups, designing by, 26-3
groups, section cuts, 26-4
groups, selecting by, 13-4, 26-3

H

Help menu commands
 Search for Help on, 19-1
 About ETABS, 19-1
height, story level, 8-6, 9-17

I

import options
 Open ETABS7 .e2k Text File, 8-21
 Open ETABS6 Text File, 8-21
 Overwrite Story from SAFE .f2k Text File, 8-22
 Overwrite Story from ETABS7 .edb File, 8-22
incremental analysis, 33-22
initial p-delta analysis, 33-18
initialization of model, 8-2
inserting a story level, 9-17
installation
 hardware key device, 2-11
 network server, 2-5
 network workstation, 2-6
 sentinel driver, 2-9
 single user, 2-4
 troubleshooting, 2-14
interactive composite beam design, 47-14

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

interactive steel frame design, 45-14
 interactive concrete frame design, 46-11
 interactive shear wall design, 48-17
 intersecting line select mode, 13-3
 invert selection, 13-5
 invisible grid for snapping, 12-19

J

join lines, 9-33
 joint *See point object*

K

keyboard commands *See inside front cover*

L

limits, viewing, 10-10
 line objects
 assignments to, 14-22, 24-5
 drawing, 12-3
 right click information, 24-7
 type, 24-3
 line, reference, 9-18
 line thickness (preference)
 printer, 18-3
 screen, 18-3
 linear static analysis, 33-2
 link element
 assigned to line objects, 14-32
 assigned to panel zones, 14-5
 assigned to point objects (grounded), 14-13
 internal deformations, 37-4
 internal forces, 37-6
 internal nonlinear springs, 37-2
 properties, 11-26
 live load reduction, 11-47, 18-10
 loads, assignment
 area object
 temperature load, 14-60
 uniform surface load, 14-58
 line object
 distributed load, 14-42

point load, 14-40
 temperature load, 14-46
 point object
 force load, 14-16
 ground displacement, 14-18
 temperature load, 14-20
 loads, displaying on model
 joint/point, 16-2
 frame/line, 16-3
 shell/area, 16-5
 load cases
 response spectrum, 11-50, 27-4
 static, 11-46, 27-2
 static nonlinear (pushover), 11-63, 27-6
 time history, 11-56, 27-5
 load combinations, 27-6
 load transformation, 32-1
 local axes
 area object, 14-50, 23-17
 coordinate system, 9-15, 21-4
 frame element, 35-1
 line object, 14-29, 24-29
 link element, 14-13, 37-2
 panel zone, 14-8
 point object, 25-12
 shell element, 36-1
 section cut, 11-29, 26-6
 wall pier, 38-2
 wall spandrel, 38-5
 locking model, 4-13, 18-22
 log file, 43-1

M

magnifying the view *See zoom*
 major axis, 14-30
 major direction, 14-30
 mass, additional
 area, 14-56
 line, 14-38
 point, 14-14
 mass per unit volume, 11-4
 mass source, 11-64, 27-11
 material property, 11-1

ETABS User's Manual – Volume 1

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

maximum graphic font size (preference), 18-3
measurements

angle between two lines, 10-16
perimeter and area of an area, 10-15
line length, 10-15
membrane property, 11-21, 11-22, 30-1
merge
 areas, 9-32
 points, 9-19
 tolerance, 9-19, 18-2

mesh areas
 automatic, 30-1
 manual, 31-1

mesh line objects *See divide lines*

Microsoft Access *See database, Microsoft Access*

Microsoft Excel *See spreadsheet, copying geometry to and from*

minimum graphic font size (preference), 18-4

minor axis, 14-30
minor direction, 14-30
modal analysis, 33-2
modal direction factors, 41-5
modal effective mass factors, 41-5
modal participation factors, 41-4
modal periods and frequencies, 41-3
mode shapes, 16-12, 41-3

modifiers
 frame property, 14-31
 shell stiffness, 14-51

modulus
 shear, 11-3
 Young's, 11-2

mouse, using, 4-9
moving objects, 9-29

N

nonlinear
 frame hinge properties (pushover), 11-27
 link properties, 11-26, 37-2
 static analysis (pushover), 33-22
 time history analysis, 33-17
nsrvx.exe, 2-13
nudging objects, 9-42

O

OK button, 4-14
openings, 14-49
Options menu commands
 Preferences
 Dimensions/Tolerances, 18-2
 Output Decimals, 18-6
 Reinforcement Bar Sizes, 18-7
 Live Load Reduction, 18-10
Colors
 Display, 18-15
 Output, 18-17
Windows, 18-20
Show Tips at Startup, 18-20
Show Bounding Plane, 18-21
Moment Diagrams on Tension Side, 18-22
Sound, 18-22
Lock Model, 18-22
Show Aerial View Window, 18-23
Show Floating Property Window, 18-23
Show Crosshairs, 18-23
output
 conventions, 34-1, 35-1, 36-1, 37-1, 38-1, 39-1,
 41-13
 decimals (preferences), 18-6
 display colors, 18-15
 displayed on screen, 16-1
 onscreen output tables, 16-6, 16-40
 printed to printer or file, 8-27
 stations, 14-28
out file, 43-3
overlapping area objects, 23-15
overturning moments, 41-13
overwrite frame design procedure, 17-2

P

pan, 10-13
pan margin (preference), 18-4
panel zone
 assignments, 14-3
 displacements, 34-3
 internal deformations, 34-4

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

internal forces, 34-5
 p-delta analysis, 33-18
 period, 41-3
 perspective view, 10-8
 phone number, 3-4
 piers
 labels, 14-34, 14-52, 48-1
 output forces, 16-17, 38-4
 plan fine grid spacing (preference), 18-2
 plan nudge value (preference), 18-3
 plane
 bounding, 18-21
 reference, 9-18
 plate bending, 11-22
 point object
 assignments to, 14-1, 25-3
 drawing, 12-3
 on top of another point object, 25-12
 output conventions, 34-1
 right click information, 25-3
 polyline, 24-31
 preferences, 18-1
 print
 graphics, 8-27
 preview, 8-27
 tables, 8-28

Q

quadrilaterals, drawing *See Draw menu commands*
 quick keys *See inside front cover*
 quitting ETABS, 8-30

R

reactions, 16-14, 34-2
 readme.txt, 2-5
 rectangles, drawing *See Draw menu commands*
 redo, 4-14
 reduction, live load, 18-10
 reference lines, 9-18
 reference planes, 9-18
 refresh view, 10-14

refresh window, 10-14
 reinforcing
 bar sizes (preference), 18-7
 beam, 11-17
 column, 11-19
 relabeling objects, 23-5
 replicating objects, 9-9
 reshaper tool, 9-37
 residual mass modes, 33-7
 response spectrum analysis, 33-12
 response spectrum analysis output
 damping and accelerations, 41-10
 modal amplitudes, 41-11
 base reactions, 41-11
 response spectrum curve
 from time history analysis results, 16-29
 input function, 11-29
 restore previous selection, 13-5
 restraints, 14-9
 right click
 on area object, 23-6
 on line object, 24-7
 on point object, 25-3
 right hand rule, 23-18
 rigid diaphragm assignment
 to area object, 14-49
 to point objects, 14-2
 ritz-vector analysis, 33-8
 rotate 3D view, 10-5
 rubber band line, 12-4, 12-11, 13-3
 rubber band window, 12-3, 12-5, 12-6, 12-7, 12-12, 13-2

S

saving model, 8-20
 screen selection tolerance (preference), 18-3
 screen snap to tolerance (preference), 18-3
 screen line thickness (preference), 18-3
 secondary beams, 12-6
 section cuts
 defining, 11-27, 26-4
 output forces, 26-7, 39-1
 section designer, 11-11, 48-20

ETABS User's Manual – Volume 1

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

seismic load *See automatic seismic loads*
select all, 13-5
select edge of area object, 9-31
Select menu commands
 Select at Pointer/in Window, 13-2
 Select using Intersecting Line, 13-3
 Select on XY Plane, 13-4
 Select on XZ Plane, 13-4
 Select on YZ Plane, 13-4
 Select by Groups, 13-4
 Select by Frame Sections, 13-4
 Select by Wall/Slab/Deck Sections, 13-4
 Select by Link Properties, 13-4
 Select by Line Object Type, 13-5
 Select by Area Object Type, 13-5
 Select by Story Level, 13-5
 Select All, 13-5
 Select Invert, 13-5
 Deselect, 13-6
 Get Previous Selection, 13-5
 Clear Selection, 13-6
self-mass, 11-4, 11-64
self-weight, 11-4, 11-24, 11-47
select mode, 4-12
sequential analysis *See incremental analysis*
shear wall design process, 48-8
shell element faces, 36-2
shell element section properties, 11-21
shell element internal forces, 36-3
shell element internal stresses, 36-8
shift key, 4-9, 4-10
show all, 10-11
show selection only, 10-11
shrink area objects, 9-30
shrink factor (preference), 18-5
similar stories feature, 12-2, 22-3
snap to options
 intersections, 12-19
 invisible grid, 12-19
 lines and edges, 12-19
 middle and ends, 12-19
 perpendicular, 12-19
 points, 12-18
sound, 18-22

spandrels
 labels, 14-35, 14-53, 48-5
 output forces, 16-17, 38-5
spreadsheet, copying geometry to and from, 9-3
spring
 forces, 16-14
 properties
 area, 14-54
 line, 14-36
 point, 14-10
starting a new model, 4-11, 6-1, 8-1
starting load vectors, 33-10
static analysis, 33-2
static load participation ratio, 41-7
static pushover curve, 16-39
status bar, 4-5
steel frame design process, 45-3
story levels, 8-6, 9-17, 22-1
story shears, 41-13
support, technical
 e-mail support, 3-5
 fax support, 3-4
 phone support, 3-4
 training, at CSI in Berkeley, 3-6
supports
 area spring, 14-54
 line spring, 14-36
 point spring, 14-10
reactions, 16-14
restraints, 14-9

T

tables displayed onscreen
input tables, 16-6
output tables, 16-40
static nonlinear analysis tables, 16-39
time history trace tables, 16-38
telephone numbers
 fax, 3-4
 voice, 3-4
templates, 8-6, 9-7, 11-42
thick plate, 11-22

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

- time history
 - analysis, 33-14
 - case, 11-56
 - function, 11-38
 - trace, 16-34
 - time history types
 - linear, 33-14
 - nonlinear, 33-14
 - periodic, 33-14
 - tips, showing at startup, 18-20
 - tolerances, preferences, 18-2
 - toolbar buttons
 - main (top) toolbar buttons
 - New Model, 8-1
 - Open .EDB File, 8-19
 - Save Model, 8-20
 - Undo, 4-14
 - Redo, 4-14
 - Refresh Window, 10-14
 - Lock/Unlock Model, 18-22
 - Run Analysis, 15-8
 - Rubber Band Zoom, 10-11
 - Restore Full View, 10-12
 - Restore Previous Zoom, 10-12
 - Zoom In One Step, 10-13
 - Zoom Out One Step, 10-13
 - Pan, 10-14
 - 3D View, 10-2, 10-5
 - Plan View, 10-5
 - Elevation View, 10-6
 - Rotate 3D View, 10-5
 - Perspective Toggle, 10-8
 - Move Up in List, 10-5
 - Move Down in List, 10-5
 - Object Shrink Toggle, 10-18
 - Set Building View Options, 10-16
 - Show Undeformed Shape, 16-1
 - Display Static Deformed Shape, 16-7
 - Display Mode Shape, 16-12
 - Display Member Force Diagram, 16-14
 - Display Output Tables, 16-40
 - side toolbar buttons
 - Pointer, 12-1
 - Reshaper, 9-37
- Draw Point Objects (displays flyout button), 12-3
 - Create Points (plan, elev, 3D), 12-3
 - Draw Line Objects (displays flyout buttons), 12-3
 - Draw Lines (plan, elev, 3D), 12-4
 - Create Lines in Region or at Clicks (all views), 12-5
 - Create Columns in Region or at Clicks (plan), 12-6
 - Create 2ndary Beams in Region or at Clicks (plan), 12-6
 - Create Braces in Region or at Clicks (elev), 12-7
 - Draw Area Objects (displays flyout buttons), 12-9
 - Draw Areas (plan, 3D), 12-9
 - Draw Rectangular Areas (plan, elev), 12-10
 - Create Areas at Click (plan, elev), 12-10
 - Draw Walls (plan), 12-10
 - Create Walls in Region or at Click (plan), 12-12
 - Select All, 13-5
 - Restore Previous Selection, 13-5
 - Clear Selection, 13-6
 - Set Intersecting Line Select Mode, 13-3
 - Snap to Points, 12-18
 - Snap to Middle and Ends, 12-19
 - Snap to Intersections, 12-19
 - Snap to Perpendicular, 12-19
 - Snap to Lines and Edges, 12-19
 - Snap to Invisible Grid, 12-19
 - training, at CSI in Berkeley, 3-6
 - trim line objects, 9-25

U

- undeformed shape, display, 16-1
- undo, 4-14
- units, 20-1
- unlocking model, 4-13, 18-22
- unstable end releases, 14-24

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

V

video, create and playback, 8-26

View menu commands

Set 3D View, 10-2

Set Plan View, 10-5

Set Elevation View, 10-6

Set Building View Limits, 10-10

Set Building View Options, 10-16

Rubber Band Zoom, 10-11

Restore Full View, 10-12

Previous Zoom, 10-12

Zoom In One Step, 10-13

Zoom Out One Step, 10-13

Pan, 10-13

Measure

Line, 10-15

Area, 10-15

Angle, 10-16

Change Axes Location, 10-15

Show Selection Only, 10-11

Show All, 10-11

Save Custom View, 10-9

Show Custom View, 10-9

Refresh Window, 10-14

Refresh View, 10-14

Y

Young's modulus, 11-2

Z

zoom

in one step, 10-13

out one step, 10-13

previous zoom, 10-12

restore full view, 10-12

rubber band zoom, 10-11

I

W

walls

drawing *See Draw menu commands, Draw*

Area Objects

assigning pier labels to area objects, 14-52

assigning pier labels to line objects, 14-34

assigning spandrel labels to area objects, 14-53

assigning spandrel labels to line objects, 14-35

wind load *See automatic wind loads*

windows, Options menu command, 18-20

windowing, 13-2

world wide web address, 3-3

ETABS®

Three Dimensional Analysis and Design of Building Systems

ETABS USER'S MANUAL Volume 2



**Computers and Structures, Inc.
Berkeley, California, USA**

First Edition
December 1999

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., 1978-1999.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
ETABS is a registered 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.

ACKNOWLEDGMENT

Thanks are due to all of the numerous structural engineers, who over the years have given valuable feedback that has contributed toward the enhancement of this product to its current state.

Special recognition is due Dr. Edward L. Wilson, Professor Emeritus, University of California at Berkeley, who was responsible for the conception and development of the original ETABS and whose continued originality has produced many unique concepts that have been implemented in this version.



Volume 2 Contents



Tip:

If you are just getting started with ETABS Version 7 we suggest that you read Chapters 1 through 6 in Volume 1 and then use the rest of the manual (Volumes 1 and 2) as a reference guide on an as-needed basis. If you are not responsible for installing ETABS then you can skip Chapter 2.

The Table of Contents for Volume 2 of this manual consists of a chapter list that covers both Volumes 1 and 2 followed by an expanded table of contents for Volume 2 only. The chapter list devotes one line to each chapter. It shows you the chapter number (if applicable), chapter title and the pages that the chapter covers. Subheadings are provided in the chapter list section to help give you a sense of how this manual is divided into several different parts.

Following the chapter list is the expanded table of contents for Volume 2. Here all section headers and subsection headers are listed along with their associated page numbers for each chapter in Volume 2.

When searching through the manual for a particular chapter, the highlighted tabs at the edge of each page may help you locate the chapter more quickly.

If you are new to ETABS we suggest that you read Chapters 1 through 6 in Volume 1 and then use the rest of the manual (Volumes 1 and 2) as a reference guide on an as-needed basis. If you are not responsible for installing ETABS then you can skip Chapter 2.

ETABS User's Manual Chapter List

Volume 1

Volume 1 Contents

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	Chapter List.....	i to vi
N. A.	Expanded Table of Contents.....	vi to xx



Note:

Chapter 4 provides a comprehensive description of the various parts of the ETABS graphic interface.

Introduction and Getting Started Information

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
1	Introduction	1-1 to 1-10
2	Installation	2-1 to 2-14
3	Getting Help	3-1 to 3-7



Note:

Chapter 6 provides information on how to create an ETABS model.

General Overview of ETABS

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
4	The ETABS Graphical User Interface	4-1 to 4-16
5	Overview of an ETABS Model.....	5-1 to 5-5
6	ETABS Modeling Tips	6-1 to 6-10

The ETABS Menus

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
7	Overview of the ETABS Menus	7-1 to 7-2
8	The ETABS File Menu	8-1 to 8-30
9	The ETABS Edit Menu.....	9-1 to 9-43
10	The ETABS View Menu	10-1 to 10-30
11	The ETABS Define Menu.....	11-1 to 11-65
12	The ETABS Draw Menu.....	12-1 to 12-22
13	The ETABS Select Menu.....	13-1 to 13-6
14	The ETABS Assign Menu	14-1 to 14-64
15	The ETABS Analyze Menu	15-1 to 15-9
16	The ETABS Display Menu	16-1 to 16-40
17	The ETABS Design Menu.....	17-1 to 17-6
18	The ETABS Options Menu.....	18-1 to 18-23
19	The ETABS Help Menu.....	19-1 to 19-2

**Note:**

Chapters 8 through 19 document most of the menu commands and toolbar button shortcuts available in ETABS.

Other Information

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	References.....	References-1 to References-4
N. A.	Index	Index-1 to Index-12

Volume 2***Volume 2 Contents***

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	Chapter List.....	i to vi
N. A.	Expanded Table of Contents.....	vi to xxi

Detailed Information on Selected ETABS Topics

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
20	Units	20-1 to 20-5
21	Coordinate Systems	21-1 to 21-7
22	Story Level Data.....	22-1 to 22-5
23	Area Objects	23-1 to 23-21
24	Line Objects	24-1 to 24-31
25	Point Objects.....	25-1 to 25-12
26	Groups and Section Cuts	26-1 to 26-12
27	Load Cases, Load Combinations and Mass.....	27-1 to 27-13
28	Automatic Seismic Loads.....	28-1 to 28-37
29	Automatic Wind Loads	29-1 to 29-19
30	Automatic Meshing of Area and Line Objects ..	30-1 to 30-12
31	Manual Meshing of Area Objects	31-1 to 31-16
32	Transformation of Loads into the ETABS Analysis Model	32-1 to 32-32
33	Overview of ETABS Analysis Techniques.....	33-1 to 33-24

Note:

Volume 2 provides detailed information on a variety of ETABS subjects.



ETABS Analysis Output**Note:**

Chapters 34 through 43 document the ETABS analysis output.

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
34	Point Object Output Conventions.....	34-1 to 34-5
35	Frame Element Output Conventions.....	35-1 to 35-5
36	Shell Element Output Conventions	36-1 to 36-14
37	Link Element Output Conventions.....	37-1 to 37-7
38	Wall Pier and Spandrel Output Conventions.....	38-1 to 38-6
39	Section Cut Output Conventions.....	39-1 to 39-4
40	Printed Input Tables.....	40-1 to 40-3
41	Printed Output Tables	41-1 to 41-14
42	Database Input/Output Tables	42-1 to 42-2
43	The ETABS Log and Out Files.....	43-1 to 43-6
44	Inserting ETABS Output into Written Reports	44-1 to 44-3

Note:

Chapters 45 through 48 provide information on how to use the ETABS design postprocessors

Introduction to the ETABS Design Postprocessors

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
45	Steel Frame Design	45-1 to 45-17
46	Concrete Frame Design.....	46-1 to 46-15
47	Composite Beam Design	47-1 to 47-19
48	Shear Wall Design	48-1 to 48-28

Other Information

<u>Chapter</u>	<u>Title</u>	<u>Pages</u>
N. A.	References.....	References-1 to References-4
N. A.	Appendix 1 - The ETABS Menu Structure.....	A1-1 to A1-13
N. A.	Index	Index-1 to Index-12

ETABS User's Manual - Volume 2 - Expanded Contents

CHAPTER 20: UNITS

General	20-1
Different Sets of Units Available in ETABS	20-2
Current Units versus Database Units	20-3
Current Units	20-3
Database Units	20-4
Changing the Database Units	20-4

CHAPTER 21: COORDINATE SYSTEMS

Overview	21-1
Upward and Horizontal Directions	21-1
The Global Coordinate System	21-2
Additional Coordinate Systems	21-3
Local Coordinate Systems	21-5
Grid Lines	21-5

CHAPTER 22: STORY LEVEL DATA

- Definition of a Story Level 22-1
- Editing Story Level Data 22-2
- Similar Story Levels 22-3
 - The Story Data Dialog Box 22-3
 - The Similar Stories Drop-Down Box 22-4
- Effect of Story Levels on Drawn Objects 22-5

CHAPTER 23: AREA OBJECTS

- General 23-1
- Area Object Labeling and Area Type 23-2
 - Area Type 23-3
 - Automatic Area Object Labeling 23-4
 - Relabeling Objects 23-5
- Assignments Made to Area Objects 23-5
- Right Click Information for Area Objects 23-6
 - Location Tab in the Area Information Dialog Box 23-7
 - Assignments Tab in the Area Information Dialog Box 23-9
 - Loads Tab in the Area Information Dialog Box 23-13
- Overlapping Area Objects 23-15
- Plan Views of Walls 23-16
- Default Area Object Local Axes 23-17
 - Default Orientation for Horizontal Area Objects 23-17
 - Default Orientation for Vertical Area Objects 23-17
 - Default Orientation for Other Area Objects 23-18
- The Right Hand Rule 23-18
- Positive Direction of Coordinate System Axes 23-19
 - Global Coordinate System 23-19
 - Local Coordinate System 23-19

CHAPTER 24: LINE OBJECTS

- General 24-1
- Frame Section Properties 24-1
- Line Object Labeling and Line Type 24-3
 - Line Type 24-3
 - Automatic Line Object Labeling 24-4
- Assignments Made to Line Objects 24-5
- Right Click Information for Line Objects 24-7
 - Location Tab in the Line Information Dialog Box 24-8
 - Assignments Tab in the Line Information Dialog Box 24-9
 - Loads Tab in the Line Information Dialog Box 24-17
- Overlapping Line Objects 24-26
- Plan Views of Vertical Line Objects 24-28
- Default Line Object Local Axes 24-29
 - Vertical Line Objects 24-29
 - Horizontal Line Objects 24-29
 - Other Line Objects 24-30
- Polylines 24-31

CHAPTER 25: POINT OBJECTS

- General 25-1
- Automatic Point Object Labeling 25-2
- Assignments Made through the Assign Menu 25-3
- Right Click Information for Point Objects 25-3
 - Location Tab in the Point Information Dialog Box 25-4
 - Assignments Tab in the Point Information Dialog Box 25-5
 - Loads Tab in the Point Information Dialog Box 25-10

Point Objects Overlapping Other Objects 25-12

Point Object Local Axes 25-12

CHAPTER 26: GROUPS AND SECTION CUTS

General 26-1

Defining Groups 26-2

Selecting Groups 26-3

Designing by Groups 26-3

Section Cuts 26-4

Defining a Section Cut with a Group 26-4

Location that Section Cut Forces are Summed About 26-6

Local Axes for Section Cuts 26-6

How ETABS Calculates Section Cut Forces 26-7

CHAPTER 27: LOAD CASES, LOAD COMBINATIONS AND MASS

Load Cases 27-1

Static Load Case 27-2

Design Type 27-3

Self-Weight Multiplier 27-4

Response Spectrum Load Case 27-4

Time History Load Case 27-5

Static Nonlinear Load Case 27-6

Live Load Reduction 27-6

General 27-6

Load Combinations 27-6

General 27-6

Output Values Produced by Load Combinations 27-7

Types of Load Combinations 27-8

Examples 27-9

- Design Load Combinations 27-10
- Restrictions for Load Case and Load Combination Labels 27-11
- Mass 27-11

CHAPTER 28: AUTOMATIC SEISMIC LOADS

- General 28-1
- Defining Automatic Seismic Load Cases 28-2
- Automatic Seismic Load Cases 28-3
 - Distribution of Automatic Seismic Loads at a Story Level 28-3
 - Direction and Eccentricity Data 28-3
 - Story Range Data 28-5
 - 1994 UBC Seismic Loads 28-6
 - Options for 1994 UBC Building Period 28-6
 - Other Input Factors and Coefficients 28-7
 - Algorithm for 1994 UBC Seismic Loads 28-7
 - 1997 UBC Seismic Loads 28-10
 - Options for 1997 UBC Building Period 28-10
 - Other Input Factors and Coefficients 28-10
 - Algorithm for 1997 UBC Seismic Loads 28-11
 - 1997 UBC Isolated Building Seismic Loads 28-14
 - Other Input Factors and Coefficients 28-14
 - Algorithm for 1997 UBC Isolated Building Seismic Loads 28-15
 - 1996 BOCA Seismic Loads 28-17
 - Options for 1996 BOCA Building Period 28-17
 - Other Input Factors and Coefficients 28-18
 - Algorithm for 1996 BOCA Seismic Loads 28-19
 - 1995 NBCC Seismic Loads 28-21
 - Options for 1995 NBCC Building Period 28-21
 - Other Input Factors and Coefficients 28-22

Algorithm for 1995 NBCC Seismic Loads	28-23
IBC 2000 Seismic Loads	28-25
Options for IBC 2000 Building Period	28-25
Other Input Factors and Coefficients	28-26
Algorithm for IBC 2000 Seismic Loads	28-27
1997 NEHRP Seismic Loads	28-30
Options for 1997 NEHRP Building Period	28-30
Other Input Factors and Coefficients	28-31
Algorithm for 1997 NEHRP Seismic Loads	28-33
User-Defined Seismic Loads	28-36
Input Factors and Coefficients	28-36
Algorithm for User-Defined Seismic Loads	28-36

CHAPTER 29: AUTOMATIC WIND LOADS

General	29-1
Defining Automatic Wind Load Cases	29-2
Automatic Wind Load Cases	29-3
Wind Direction	29-3
Wind Exposure Height	29-4
Wind Exposure Width and Wind Load Application Point	29-5
1994 UBC Wind Loads	29-6
Input Wind Coefficients	29-6
Algorithm for 1994 UBC Wind Loads	29-7
1997 UBC Wind Loads	29-8
Input Wind Coefficients	29-8
Algorithm for 1997 UBC Wind Loads	29-9
1996 BOCA Wind Loads	29-11
Input Wind Coefficients for 1996 BOCA	29-11
Algorithm for 1996 BOCA Wind Loads	29-12

1995 NBCC Wind Loads	29-14
Input Wind Coefficients	29-14
Algorithm for 1995 NBCC Wind Loads	29-14
ASCE 7-95 Wind Loads	29-16
Input Wind Coefficients for ASCE 7-95	29-16
Algorithm for ASCE 7-95 Wind Loads	29-16
User-Defined Wind Loads	29-19

CHAPTER 30: AUTOMATIC MESHING OF AREA AND LINE OBJECTS

General	30-1
Automatic Meshing of Line Objects	30-3
Automatic Meshing of Area Objects	30-6
General	30-6
How ETABS Automatically Meshes Floors	30-7
Viewing the Automatic Floor Mesh	30-7
Examples of Automatic Floor Meshing	30-8

CHAPTER 31: MANUAL MESHING OF AREA OBJECTS

General	31-1
Cookie Cut Meshing Tools	31-3
Cookie Cut at Selected Line Objects	31-3
Cookie Cut at Selected Points at X Degrees	31-4
Meshing Tools for Quadrilaterals and Triangles	31-7
Background Information	31-7
Four-sided Area Objects	31-7
Three-sided Area Objects	31-8
Mesh Quadrilaterals and Triangles into N by M Areas	31-8
Mesh Quadrilaterals and Triangles at Intersections and Selected Points on Edges	31-11

- Mesh at Intersection with Visible Gridlines 31-12
- Mesh at Selected Point Objects on Edges 31-14
- Mesh at Intersections with Selected Line Objects 31-15
- Example with Combined the Mesh Sub-options 31-16

CHAPTER 32: TRANSFORMATION OF LOADS INTO THE ETABS ANALYSIS MODEL

- Background 32-1
- Valid Loading 32-2
 - Point Objects 32-2
 - Line Objects 32-3
 - Area Objects 32-3
- Introduction to Load Transformation 32-4
- Load Transformation for Area Objects 32-6
- Vertical Load Transformation for Floors with Deck Properties 32-11
 - Rectangular Interior Meshed Element 32-12
 - General Interior Meshed Element 32-15
 - Exterior Meshed Element 32-18
 - The Effect of Deck Openings on Load Transformation 32-24
- Vertical Load Transformation for Floors with Membrane Slab Properties 32-26

CHAPTER 33: OVERVIEW OF ETABS ANALYSIS TECHNIQUES

- General 33-1
- Linear Static Analysis 33-2
- Modal Analysis 33-2
 - Eigenvector Analysis 33-3
 - Number of Modes 33-4
 - Frequency Range 33-4
 - Convergence Tolerance 33-6
 - Residual Mass Modes 33-7

- Ritz-Vector Analysis 33-8
 - Number of Modes 33-9
 - Starting Load Vectors 33-10
- Acceleration Loads 33-11
- Response Spectrum Analysis 33-12
- Time History Analysis 33-14
 - Mode Superposition 33-15
 - Modal Damping 33-15
 - Time Steps 33-16
 - Nonlinear Time-History Analysis 33-17
- Initial P-Delta Analysis 33-18
- Iterative Solution 33-20
- Buckling 33-21
- Practical Application 33-21
- Nonlinear Static Analysis 33-22

CHAPTER 34: POINT OBJECT OUTPUT CONVENTIONS

- Overview 34-1
- Displacements 34-1
- Support Reactions 34-2
- Spring Forces 34-3
- Grounded Link Element Forces 34-3
- Panel Zone Output 34-3
 - Panel Zone Displacements 34-3
 - Panel Zone Internal Deformations 34-4
 - Panel Zone Internal Forces 34-5

CHAPTER 35: FRAME ELEMENT OUTPUT CONVENTIONS

- General 35-1

Frame Element Internal Forces 35-2

CHAPTER 36: SHELL ELEMENT OUTPUT CONVENTIONS

General 36-1

Faces of Shell Elements 36-2

Shell Element Internal Forces 36-3

Shell Element Internal Stresses 36-8

Other Formulas Relating Shell Element Internal Forces to Internal
Stresses 36-12

CHAPTER 37: LINK ELEMENT OUTPUT CONVENTIONS

General 37-1

Link Element Assignments to Point and Line Objects 37-1

Internal Nonlinear Springs 37-2

Link Element Force-Deformation Relationships 37-3

Link Element Internal Deformations 37-4

Link Element Internal Forces 37-6

CHAPTER 38: WALL PIER AND SPANDREL OUTPUT CONVENTIONS

General 38-1

Wall Pier Output Locations and Sign Convention 38-2

 Wall Pier Local Axes 38-2

 Two-Dimensional Pier 38-2

 Three-Dimensional Pier 38-3

 Pier Element Internal Forces 38-4

Wall Spandrel Output Locations and Sign Convention 38-4

 Wall Spandrel Local Axes 38-5

 Spandrel Element Internal Forces 38-5

CHAPTER 39: SECTION CUT OUTPUT CONVENTIONS

Overview 39-1

Section Cut Forces 39-1

CHAPTER 40: PRINTED INPUT TABLES

Features of Dialog Box 40-1

Input Data Categories 40-2

Building Data 40-2

Object Data 40-2

Static Loads 40-3

CHAPTER 41: PRINTED OUTPUT TABLES

General 41-1

Building Modes 41-3

Building Modal Info 41-3

Modal Periods and Frequencies 41-3

Modal Participation Factors 41-4

Modal Direction Factors 41-5

Modal Effective Mass Factors 41-5

Static and Dynamic Load Participation Ratios 41-7

 Static Load Participation Ratio 41-7

 Dynamic Load Participation Ratio 41-8

 Final Comments on Static and Dynamic Load Participation Ratios 41-10

Damping and Accelerations 41-10

Modal Amplitudes 41-11

Base Reactions 41-11

Building Output 41-12

 Cumulative Center of Mass 41-12

- Center of Rigidity 41-12
- Story Shears and Overturning Moments 41-13

CHAPTER 42: DATABASE INPUT/OUTPUT TABLES

- General 42-1

CHAPTER 43: THE ETABS LOG AND OUT FILES

- General 43-1
- The ETABS Log File 43-1
- The ETABS Out File 43-3
- Global Force Balance 43-3

CHAPTER 44: INSERTING ETABS OUTPUT INTO WRITTEN REPORTS

- General 44-1
- Tabular Output 44-1
- Graphical Output 44-2

CHAPTER 45: STEEL FRAME DESIGN

- Analysis Sections and Design Sections 45-1
- Steel Frame Design Procedure 45-3
- Steel Frame Design Menu Commands 45-7
 - Select Design Group 45-7
 - Select Design Combo 45-8
 - View/Revise Overwrites 45-8
 - Set Lateral Displacement Targets 45-9
 - Start Design/Check of Structure 45-10
 - Interactive Steel Frame Design 45-11
 - Display Design Info 45-11
 - Make Auto Select Section Null 45-11

- Change Design Section 45-12
- Reset Design Section to Last Analysis 45-13
- Verify Analysis vs Design Section 45-13
- Reset All Steel Overwrites 45-14
- Delete Steel Design Results 45-14
- ETABS Interactive Steel Frame Design 45-14

CHAPTER 46: CONCRETE FRAME DESIGN

- Analysis Sections and Design Sections 46-2
- Concrete Frame Design Procedure 46-4
- Concrete Frame Design Menu Commands 46-7
 - Select Design Combo 46-7
 - View/Revise Overwrites 46-7
 - Start Design/Check of Structure 46-8
 - Interactive Concrete Frame Design 46-8
 - Display Design Info 46-9
 - Change Design Section 46-9
 - Reset Design Section to Last Analysis 46-10
 - Verify Analysis vs Design Section 46-10
 - Reset All Concrete Overwrites 46-11
 - Delete Concrete Design Results 46-11
- ETABS Interactive Concrete Frame Design 46-11

CHAPTER 47: COMPOSITE BEAM DESIGN

- Analysis Sections and Design Sections 47-1
- Composite Beam Design Procedure 47-3
- Composite Beam Design Menu Commands 47-7
 - Select Design Group 47-7
 - Select Design Combo 47-8

- View/Revise Overwrites 47-9
- Start Design/Check of Structure 47-10
- Interactive Composite Beam Design 47-10
- Display Design Info 47-11
- Make Auto Select Section Null 47-11
- Change Design Section 47-11
- Reset Design Section to Last Analysis 47-12
- Verify Analysis vs Design Section 47-13
- Reset All Composite Beam Overwrites 47-13
- Delete Composite Beam Design Results 47-13
- Interactive Composite Beam Design and Review 47-14
 - Member Identification Area of Dialog Box 47-14
 - Story ID 47-14
 - Beam Label 47-14
 - Design Group 47-14
 - Section Information Area of Dialog Box 47-15
 - Auto Select List 47-15
 - Optimal 47-15
 - Last Analysis 47-15
 - Current Design/Next Analysis 47-16
 - Acceptable Sections List Area of Dialog Box 47-16
 - Redefine Area of Dialog Box 47-17
 - Sections Button 47-17
 - Overwrites Button 47-18
 - Temporary Area of Dialog Box 47-18
 - Combos Button 47-18
 - Show Details Area of Dialog Box 47-19
 - Diagrams Button 47-19
 - Details Button 47-19

CHAPTER 48: SHEAR WALL DESIGN

Overview 48-1

Wall Pier Labeling 48-1

General 48-1

Assigning Wall Pier Labels 48-2

Wall Spandrel Labeling 48-5

General 48-5

Assigning Wall Spandrel Labels 48-5

Shear Wall Design Procedure 48-8

Menu Commands for Shear Wall Design 48-11

Select Design Combo 48-11

View/Revise Pier Overwrites 48-12

View/Revise Spandrel Overwrites 48-15

Define Pier Sections for Checking 48-17

Assign Pier Sections for Checking 48-17

Start Design/Check of Structure 48-17

Interactive Wall Design 48-17

Combos Button 48-18

Overwrites Button 48-18

Display Design Info 48-18

Reset All Pier/Spandrel Overwrites 48-19

Delete Wall Design Results 48-19

Using Section Designer to Define Pier Reinforcing 48-20

Local Axes Definition and Orientation 48-20

Initial Definition of a Wall Pier Section 48-21

Starting Section Designer 48-21

Creating a Pier Section from Scratch 48-22

Creating a Pier from the Geometry of an Existing Analysis Pier Section 48-23

Modifying the Geometry of the Concrete Section 48-23

Revising Rebar Size, Cover and Spacing 48-24

 General 48-24

 Methodology 48-25

Modifying Material Properties 48-26

Tips and Tricks 48-26

 Distort Feature 48-26

 Interaction Diagrams and Moment-Curvature Plots 48-27

 Pier Orientation 48-27

 Assigning Pier Sections 48-28

REFERENCES

APPENDIX 1 - THE ETABS MENU STRUCTURE

INDEX



Chapter 20

Units

General

Except for the instances mentioned in the following paragraph ETABS uses consistent units throughout the model. You can temporarily change the current units used in the ETABS graphical interface at any time whether you are creating a model, modifying a model, or viewing output results. The current units for the model are always shown in the drop-down box located on the right-hand side of the status bar at the bottom of the ETABS window. You can change the current units on the fly by simply selecting another set of units from the drop-down box. Several different sets of both English and metric units are available. You can also change the current units using drop-down boxes located in some of the ETABS dialog boxes.



Tip:

Typically the current units are of concern to you, not the database units.

The instances where ETABS may not use consistent units throughout the model are when considering areas of reinforcing steel for concrete frame design and shear wall design. In these cases the units for the rebar are specified in the design preferences. The units can either be specified the same as the current

units (i.e., consistent units), or they can be specified as one of several predefined, unchanging units, such as, in²/ft for distributed reinforcing steel in shear walls. See, for example, the **Options menu > Preferences > Shear Wall Design** command.

Typically in ETABS you only need concern yourself with the current units that are shown in the drop-down box on the status bar. However, to fully understand the units in ETABS it is necessary to recognize the difference between the current units and the database units. The database units are discussed later in this chapter.

Different Sets of Units Available in ETABS

Several different sets of both English units and Metric units are available in ETABS. A set of units consists of force, length, temperature and time units.



Tip:

You can change the current units at any time by clicking in the drop-down box located on the right-hand side of the status bar at the bottom of the ETABS window and selecting a new set of units.

Seconds are always used as the time unit for all sets of units in ETABS. Degrees Fahrenheit is used as the temperature unit for all sets of English units and degrees centigrade is used as the temperature unit for all sets of metric units. The different sets of units that are available in ETABS are tabulated below:

<u>English Units</u>	<u>Metric Units</u>
lb, in, °F, sec	KN, m, °C, sec
lb, ft, °F, sec	KN, cm, °C, sec
kip, in, °F, sec	KN, mm, °C, sec
kip, ft, °F, sec	Kgf, m, °C, sec
	Kgf, cm, °C, sec
	Kgf, mm, °C, sec
	N, m, °C, sec
	N, cm, °C, sec
	N, mm, °C, sec
	Ton, m, °C, sec
	Ton, cm, °C, sec
	Ton, mm, °C, sec

Two types of angular units are also used in ETABS.

- Degrees are always used for specifying geometry, including local axes definitions, for both English and Metric sets of units.
- Radians are always used for specifying rotational displacements in the input data for both English and Metric sets of units.
- Output rotations are always reported in radians for both English and Metric sets of units.

Current Units versus Database Units

To fully understand the units in ETABS it is helpful to recognize the difference between the **current** units and the **database** units. Each of these is described below.

Current Units

The current units are the units that are displayed in the dropdown box located on the right hand side of the status bar at the bottom of the ETABS window. Your model is always displayed in the current units. The location of the mouse cursor, which is displayed in the status bar, is shown in the current units. All model dimensions and properties displayed in any dialog boxes are in the current units except for possibly areas of reinforcing steel as discussed previously in this chapter. Any output data that is printed or displayed onscreen is in the current units (except possibly reinforcing steel areas). In short, all aspects of your entire model are always displayed in the current units (except possibly reinforcing steel areas).

When you use the **File menu > Export > Save Model as ETABS.e2k Text File** command the *.e2k file is saved in the current units.

Database Units

When you create a new model by clicking **File menu > New Model** whatever set of units is the current units becomes what is called the database units for the newly created model. The database units are set as soon as you click the File menu > New Model command.

ETABS always saves the binary database input file (the one with the .edb extension) for your model and the associated backup text input file (the one with the .Set extension) in the database units regardless of what the current units may be at the time you save.

There is also a *.ebk file that is created automatically by ETABS after you successfully open an existing *.edb file. The *.ebk file is intended to be a backup of your *.edb file as it was when you last successfully opened it. The *.ebk file is a binary file, not a text file. The *.ebk file is always saved in the database units for the model.

Whenever you open an existing model the current units displayed in the drop-down box on the status bar are changed to display the database units for that model when the open process is successfully completed.

Once the database units are created for a model they cannot be changed unless you export your model to a text file and then import it again as described later in this chapter in the section titled “Changing the Database Units.”

Changing the Database Units

Typically you will not have to worry about the database units for your model, however, occasionally you may wish to change them. There are not very many reasons why you might want to change the database units, but two possible ones are:

- You want your model to initially appear in a different set of units when you open it.
- You want your backup text file with the .Set extension to be saved in a different set of units.



Note:

Typically you will not need to change database units for a model.

In the unlikely event that you want to change the database units for a model, you can use the following four-step procedure:

- Open the model for which you want to change the database units in the ETABS graphical interface.
- Set the current units to be what you want the new database units for the model to be.
- Use the **File menu > Export > Save Model as ETABS7.e2k Text File** command to export a text input file of your model. Note that the model is exported to this file in the current units.
- Use the **File menu > Import > Open ETABS7.e2k Text File** command to import this text file. The database units for the imported model become the same as the units associated with the *.e2k text input file. The imported model is displayed in whatever the current units are at the time it is imported.



Chapter 21

Coordinate Systems

Overview



Tip:

In ETABS the Z-axis is always vertical, with +Z being upward.

There are three types of coordinate systems in ETABS. They are the global coordinate/grid system, additional coordinate/grid systems and local coordinate systems. The global coordinate/grid system applies to the overall model. Additional coordinate systems may apply to the overall model or to a portion of the model. Each object in the model has its own local coordinate system.

Upward and Horizontal Directions

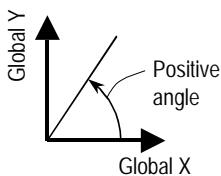
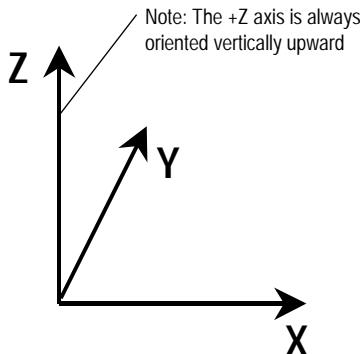


Tip:

Loading specified as “Gravity” acts in the global -Z direction.

ETABS always assumes that Z is the vertical axis, with **+Z being upward**. Local coordinate systems for elements, and ground-acceleration loading are defined with respect to this upward direction. Self-weight loading always acts downward, in the -Z direction. Also loading specified as “Gravity” loading always acts downward, in the global -Z direction.

Figure 21-1:
The global coordinate system



The X-Y plane is horizontal. The primary horizontal direction is +X. Angles in the horizontal plane are measured from the positive half of the X axis, with positive angles appearing counter-clockwise when you are looking down at the X-Y plane as shown in the sketch to the left.

The Global Coordinate System

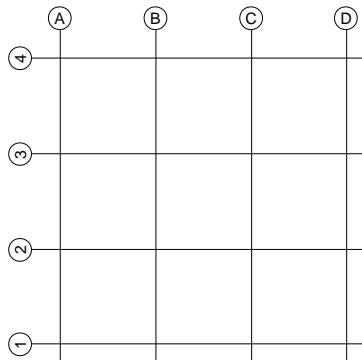
Note:

The right hand rule applies to all coordinate systems in ETABS. This includes the global, additional and local coordinate systems.

The global coordinate system is a three-dimensional, right-handed, rectangular coordinate system. The three axes denoted X, Y and Z, are mutually perpendicular and satisfy the right-hand rule. The location and orientation of the global system are arbitrary. The +Z direction is by default always upward; you can not change this. Locations in the global coordinate system can be specified using x, y, and z coordinates. See Figure 21-1 for an illustration of the global coordinate system. See the section titled "The Right Hand Rule" in Chapter 23 for additional information.

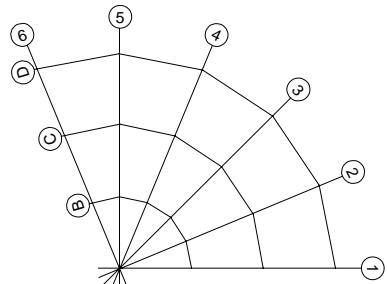
All other coordinate systems in the model, both additional and local, are ultimately defined with respect to the global coordinate system, either directly or indirectly. The coordinates of all points in the model are ultimately converted to global x, y, and z coordinates for storage in the input file, regardless of which coordinate system they may have originally been specified in.

Figure 21-2:
Plan view of grid lines from rectangular and cylindrical coordinate systems



Plan View of Grid Lines from a Rectangular Coordinate System

Note: Vertical grid lines occur at all intersections of grid lines from the same coordinate system



Plan View of Grid Lines from a Cylindrical Coordinate System

Additional Coordinate Systems

Note:

Additional coordinate systems can be either rectangular or cylindrical.



Tip:

All additional coordinate systems and element local coordinate systems are defined with respect to the global coordinate system.



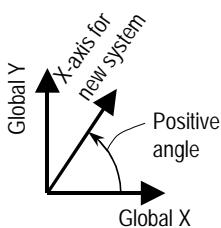
When you create a model the default coordinate system is the global system. Often, for simple models, this one coordinate system is all you will need to create your model. However, for more complex models, additional coordinate systems can be defined to supplement the global coordinate system and thus simplify locating objects in the model. Understanding these different coordinate systems is crucial to properly defining your model. You can specify additional coordinate systems using the **Edit** menu > **Edit Grid Data** command.

Additional coordinate systems may be rectangular or cylindrical as shown in Figure 21-2. Just like the global coordinate system, the +Z direction axis is always upward by default in all additional coordinate systems and you can not change this.

All additional coordinate systems are always defined with respect to the global coordinate system. There are three things to do when you define an additional coordinate system. They are:

- Specify the additional coordinate system as either rectangular or cylindrical.

- Locate the origin of the additional coordinate system with respect to the origin of the global system by specifying a translation. The translation is specified from the global origin to the additional system origin in global X and Y coordinates. The sign is important when specifying the translation.
- Specify the direction of the additional grid line system with respect to that of the global system by indicating a rotation about the Z-axis. The rotation is measured from the positive global X-axis to the positive X-axis for a rectangular additional coordinate system, or to the theta equals zero degrees axis for a cylindrical additional coordinate system. Positive rotations appear counter clockwise when the positive Z-axis is pointing toward you.



Note that additional coordinate systems can not be translated along the Z-axis with respect to the global coordinate system, nor can they be rotated about a horizontal axis.

As an example, following is a sequence of steps you might use to create a new cylindrical coordinate system in an existing model:

- Click the **Edit menu > Edit Grid Data** command to bring up the Coordinate Systems dialog box.
- Click the **Add New System** button to bring up the Coordinate System Definition dialog box.
- Select the Cylindrical option and specify the number and spacing (uniform) of grid lines.
- Click the **Edit Grid** button to bring up a dialog box where you can modify the grid data.
- If necessary modify the grid line spacing.
- Click the **Locate System Origin** button to bring up the Locate System Origin dialog box.

- Specify the global X and Y translation and the rotation about the global Z-axis of the origin of the new coordinate system relative to the origin of the global coordinate system.
- Click the **OK** button three times to exit all dialog boxes.

Local Coordinate Systems

Local coordinate systems are provided for each type of object in ETABS. Object local coordinate systems are typically defined with some reference to the global coordinate system. Refer to the following chapters for additional discussion of local coordinate systems:

- Chapter 23: Area Objects.
- Chapter 24: Line Objects.
- Chapter 25: Point Objects.

Grid Lines

Grid lines can be defined in ETABS to assist you in graphically locating objects in your model. Your mouse pointer can snap to grid lines as you are drawing objects.



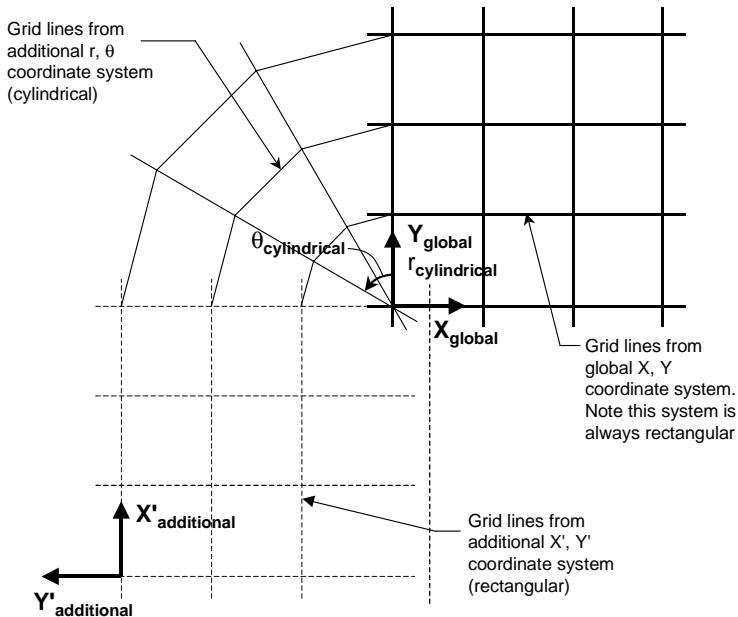
Note:

Vertical grid lines are provided at intersections of horizontal grid lines that are part of the same coordinate system.

The global coordinate system and all specified additional coordinate systems each have a system of grid lines associated with them. The grid lines associated with each coordinate system are oriented parallel to the axes of that system. Thus for the global and each additional coordinate system there are vertical grid lines and there are horizontal grid lines in two directions.

You have complete control over the location and label of the grid lines that fall in a horizontal plane. The vertical grid lines are provided automatically at each intersection point of the horizontal grid lines. The vertical grid lines (grid lines that are parallel to the global Z-axis) extend over the full height of the model, they have no label, and you cannot edit their location.

Figure 21-3:
Grid lines associated with the global coordinate system and additional coordinate systems combined to help define a complex building



Shortcut:

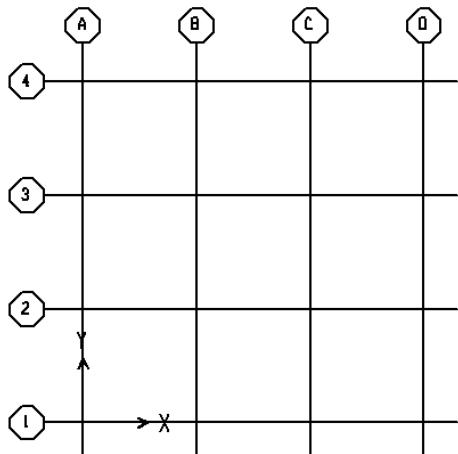
Double click on any grid line to bring up a dialog box for editing grid lines.

Note:

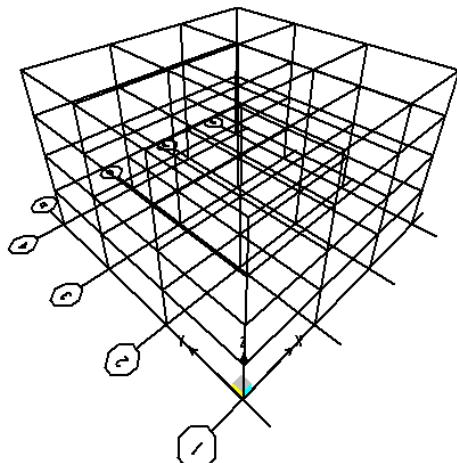
Grid lines associated with all coordinate systems are visible simultaneously.

For the global coordinate system and rectangular additional coordinate systems the grid lines are oriented in three mutually orthogonal directions. Two of those directions lie in a horizontal plane; the third direction is vertical. For cylindrical additional coordinate systems the grid lines in the horizontal plane consist of a set of concentric circles about the coordinate system origin point and a set of radial lines branching out from the coordinate system origin point. Examples of rectangular and cylindrical coordinate systems with grid lines are shown in Figure 21-2. Figure 21-3 shows a plan view demonstrating how a grid line system associated with the global coordinate system might be combined with two additional grid line systems, one rectangular and one cylindrical, to help define a complex building.

In Figure 21-3 the origin of the cylindrical coordinate system is at the same location as the global coordinate system origin. The cylindrical coordinate system origin is rotated 90° with respect to the global coordinate system origin. The origin of the additional rectangular system is also rotated 90° with respect to the origin of the global coordinate system as well as translated away from the origin of the global system.



Plan View



Three Dimensional View

(Above)

Figure 21-4:
Example three dimensional grid line system

When you define an additional coordinate system in ETABS and specify grid lines for it, the grid lines are displayed along with those of the global system and along with those of any other additional coordinate system that may have been defined. In other words, all specified grid lines are always displayed at the same time, regardless of the coordinate system they are associated with.

So far the example grid line systems have all been illustrated in plan. The grid line systems are actually three-dimensional. A vertical grid line exists at each intersection of horizontal grid lines from the same grid (coordinate) system. In addition, in a three-dimensional view, the horizontal grid lines exist at each story level as illustrated in Figure 21-4. As shown in Figure 21-4, the three-dimensional view only shows the bubbles identifying the grid lines at the base of the building so that the drawing does not become too cluttered. You can toggle the grid lines on and off as needed using the **View menu > Set Building View Options** command to display the Set Building View Options dialog box and checking or unchecking the Grid lines check box in the Other Visibility Options area.

Shortcut:

Click the  button on the main toolbar to access the Set Building Display Options dialog box.





Chapter 22

Story Level Data

Since ETABS is written specifically for buildings, the concept of story levels is logical, convenient and useful. When working with a model in the graphical user interface, you can work in a plan view of any story level. Thus story levels assist you in identifying, locating and viewing specific areas of your model. Story levels are also used to help identify specific objects in the model. For example a story level and a column label identify a specific column object. In a tall building a specific column may be identified as column label C23 at the 14th story level. This type of identification is particularly useful for output that is displayed or printed in a tabular form.

Definition of a Story Level

In ETABS a story level is a horizontal plane cut through a building at a specified elevation. With the exception of shear wall spandrel beams, the objects in ETABS that are associated with a particular story level always are located at or below the story level elevation and above the story level below. Refer to

the Shear Wall Design manual for a discussion of association of spandrel beams with story levels.

Typically it is most convenient to locate story levels at the top of steel elevation in a steel framed building, at the bottom of slab elevation (not bottom of ribs) in a concrete building with a ribbed slab and at the center of the slab in a concrete building with flat slabs.

Editing Story Level Data



Tip:

*To edit the story level data click **Edit menu** > **Edit Story Data** > **Edit** to access the Story Data dialog box.*

The story level data can be viewed and edited in the Story Data dialog box. There are two ways to access this dialog box:

- If you have clicked **File menu** > **New Model** and are currently in the process of defining a template model in the Building Plan Grid System and Story Definition dialog box then in the Story Dimensions area of the dialog box select the Custom Story Data option and then click the **Edit Story Data** button.
- If you are working on an existing model you can click **Edit menu** > **Edit Story Data** > **Edit** to access the Story Data dialog box.

Following is a list of data that is included for each story level in the building in the Story Data dialog box:

- **Label:** This is a label identifying the story level. Default values for this label are STORY1, STORY2, etc. You can change the label for any story level, for example, you may want to label your story levels 1st, 2nd, etc. Note that the bottom of the building is identified as story level BASE; you can not change this.
- **Height:** This is 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 can not change this.

- **Elevation:** This is 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 can not change them. These story level elevations are provided for informational purposes.
- **Similar To:** This is a tag that indicates the story level is similar to another story level for drawing, assignment and selection purposes when working in plan view. See the section titled "Similar Story Levels" later in this chapter for more information.

Note that the **Edit menu > Edit Story Data > Insert Story** and the **Edit menu > Edit Story Data > Delete Story** commands are also available to insert and delete story levels, respectively.

Similar Story Levels

The ETABS similar story feature is active in plan view only. It is not active when you are working in an elevation view or a three-dimensional view. The options you set for the similar stories feature affect objects drawn, assignments made and selections made in plan view. You set the options for the similar stories feature in the Story Data dialog box and in the similar stories drop-down box located on the ETABS status bar.



Note:

The similar stories feature is only active when you are working in a plan view. It is not active in elevation views or in three-dimensional views.

The Story Data Dialog Box

In the Story Data dialog box (**Edit menu > Edit Story Data > Edit**) one of the items that you specify at each story level is labeled "Similar To." You either specify another story level name or None for this item.

When you specify Story X as similar to Story Y then ETABS assumes that the similarity is two-way. That is, Story X is assumed to be similar to Story Y and Story Y is assumed to be similar to Story X.

You can change the “Similar To” assignment for a story level at any time by editing it in the Story Data dialog box. Changing the “Similar To” assignment for a story level has no affect on previously drawn objects or on previous assignments to objects. Only objects drawn or assignments made after the change are affected by the change.

The Similar Stories Drop-Down Box

The similar stories drop-down box is located on the right-hand side of the status bar at the bottom of the ETABS window. When you are working in a plan view the options available in this window are One Story, Similar Stories or All Stories. These options have the following meaning:

- **One Story:** An object drawn in plan only occurs at the level that it is drawn at. An assignment made in a plan view only applies to the object(s) actually selected. A selection made in a plan view only applies to the object(s) actually selected.
- **Similar Stories:** An object drawn in plan occurs at all story levels designated as similar to the level where the object is drawn. An assignment made to an object in a plan view also occurs at all levels designated as similar to the story where the assignment is actually made where there is an object of the same type in the same plan location as the selected object. When an object is selected in plan view, objects of the same type in the same location at different story levels that are designated as similar to the story where the selection is actually made are also selected.
- **All Stories:** An object drawn in plan occurs at all story levels. An assignment made to an object in a plan view also occurs at all story levels where there is an object of the same type in the same plan location as the selected object. When an object is selected in plan view, objects of the same type in the same location at all other story levels are also selected.

You can change the similarity option in the drop-down box in the status bar any time you are in a plan view. When you are in



Tip:

The similar stories feature can significantly speed the modeling process.

an elevation view or a three-dimensional view the similar stories drop-down box displays the word "Inactive" to remind you that the feature is inactive in these views. When you switch from plan view to an elevation or three-dimensional view and then back to a plan view ETABS will retain the similarity option that it last had in the drop-down box in the status bar for the previous plan view.

Effect of Story Levels on Drawn Objects

When you draw an area or line object in an elevation view that crosses a story level elevation, ETABS automatically breaks the object up into multiple objects. The area and line objects are broken at the story levels.

Area Objects

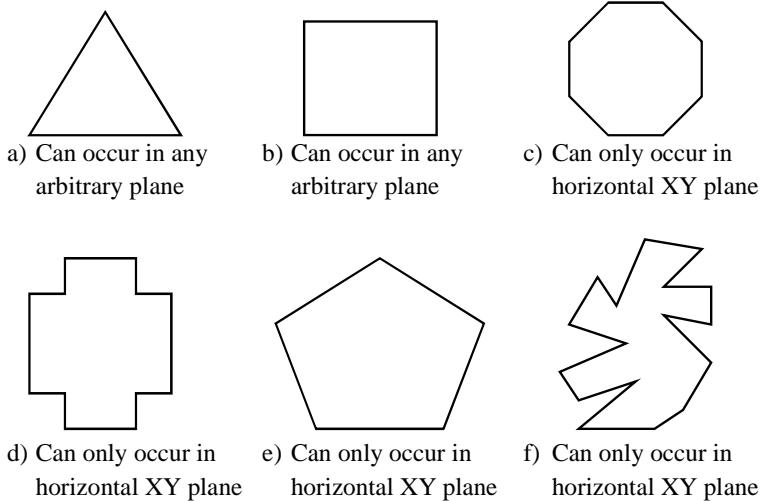
General

Area objects are defined by three or more corner points (nodes) connected by straight-line segments. Typically area objects are 3 or 4-noded although if they are horizontal (parallel to the global XY plane) they can have an unlimited number of corners.

Typically all of the nodes of an area object must be in one plane, however, ETABS does allow 4-noded elements to have a slight twist, that is, for one of the corner points to be slightly out of the plane defined by the other three points. Area objects with more than 4 corner points must be horizontal and have all corner points in the same plane.

Examples of valid area objects are shown in Figure 23-1. The area objects shown in Figure 23-1a and b can occur in any arbitrary plane. The area objects shown in Figure 23-1c, d, e and f can only occur in a horizontal plane because they have more than four corner points.

Figure 23-1:
Examples of valid
area objects



Area Object Labeling and Area Type

When you first draw an area object ETABS assigns an area type and a label to the area object. ETABS also assigns a section property to the object as it is drawn. You can control the section property assigned when the object is drawn through the floating Properties of Object box. See the section titled "Drawing Area Objects" in Chapter 12 for more information on this method of area object property assignment. Property assignments can be modified later by selecting the area object and clicking the **Assign menu > Shell/Area > Wall/Slab/Deck Section** command. See the subsection titled "Wall, Slab and Deck Section Assignments to Area Objects" in Chapter 14 for more information on this method of assigning and modifying area object properties.

The following two subsections discuss the area type and label assignments.

Area Type

If a wall, slab or deck section property is *not* assigned to the area object as it is drawn then the section is assigned an area type of "Null." If a wall, slab or deck section property is assigned to the area object as it is drawn then the section is assigned an area-type of either "Floor", "Wall" or "Ramp." The following logic is used to determine which of these frame types to assign:

Note:

Area objects are always given an area type by ETABS based on their orientation and property assignment. The possible area types are Floor, Ramp, Wall and Null.

- **Floor:** If the area object is horizontal (parallel to the global XY plane) then it is given an area type of "Floor."
- **Wall:** If the area object is vertical (parallel to the global Z-axis) then it is given an area type of "Wall."
- **Ramp:** If the area object is not vertical and not horizontal then it is given an area type of "Ramp."

The area type is used to determine the prefix used in the automatic labeling of the area object, as described in the following subsection. The area type also determines how objects may be treated for design.

You can not directly change the area type associated with an area object. However, if you change the orientation of the object then ETABS automatically changes the area type. For example, suppose you have a rectangular area object defined in a horizontal plane. This is given an area type of Floor by ETABS. Now suppose you drop two of the corner points associated with the area object one story level to create a sloping area object. ETABS immediately changes the area type for the object from Floor to Ramp. As mentioned in the following subsection, the label is also changed.

Automatic Area Object Labeling

ETABS automatically labels area objects based on their area type as follows:

- Area objects with an area type of Floor are given a label starting with the letter F followed by a number, such as F21.
- Area objects with an area type of Wall are given a label starting with the letter W followed by a number, such as W21.
- Area objects with an area type of Ramp are given a label starting with the letter R followed by a number, such as R21.
- Area objects with an area type of Null are given a label starting with the letter A followed by a number, such as A21.



Note:

ETABS automatically labels area objects.

If the area type of an area object changes then ETABS automatically relabels the area object using the correct designation as described above. There are no exceptions to this rule.

In general you can not assign your own labels to area objects. Only the automatic ETABS labels are allowed. There is one exception to this. If you import a text file into ETABS then that text file can have area object labels different from the automatic labels described above. Be aware though that in this case, if you cause the area type assigned to an area object to change then ETABS will automatically relabel the object. Also if you move the object it will be relabeled.

ETABS uses an intelligent labeling scheme. Typically it gives the same label to area objects located at the same plan location and at the same elevation relative to their respective story level, but at different story levels. Note that area objects are always identified by a story level and a label. This type of labeling scheme makes it much easier for you to locate objects. For example area object W2 at one level will be directly above area object W2 at another story level.

Relabeling Objects

Note:

You can not selectively relabel objects. You either relabel all of them or none of them.

You can use the **Edit menu > Auto Relabel All** command to automatically relabel all area, line and point objects. Note that this is an all or nothing command. You can not selectively relabel objects. Unlike other commands on the Edit menu you do not have to select the objects before relabeling them.

ETABS relabels the objects in the following order. Working in 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. Using this relabeling scheme you will typically know that if you find area object W2, then area object W3 is probably somewhere close by. Note that before relabeling the object labeling is in the order that the objects were defined.

Typically we recommend that after you finish creating your model you should use the **Edit menu > Auto Relabel All** command to get optimum labeling for the model. Keep in mind that once you do this if you later add or subtract objects from the model and then relabel again, what was area object W3 may no longer be area object W3.

Assignments Made to Area Objects

Tip:

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

As previously mentioned you can assign specific section properties to area objects as you draw them. The properties assigned are controlled through the floating Properties of Object box. See the section titled "Drawing Area Objects" in Chapter 12 for more information.

You can modify previous section property assignments and make many other types of assignments to area objects through the Assign menu. The types of area object assignments you can make include:

- Assign wall, slab and deck section properties.
- Designate an area object as an opening.
- Designate an area object as a rigid diaphragm.

- Specify the local axes orientation.
- Specify shell stiffness modifiers.
- Assign pier labels.
- Assign spandrel labels.
- Assign area spring supports (useful for soil supports).
- Assign additional area mass (if the mass source is specified as From Element and Additional Masses using the **Define menu > Mass Source** command).
- Indicate if a membrane floor element is *not* to be automatically meshed by ETABS.
- Assign uniform surface loads.
- Assign temperature loads.

Use either the **Assign menu > Shell/Area** or the **Assign menu > Shell/Area Loads** command to make these assignments. See the section titled "Assignments to Area Objects" in Chapter 14 for more information.

Right Click Information for Area Objects



Tip:

Right click on an area object to view useful information that identifies the object, describes its location and reveals its assignments.

Right click on an area object to view useful information that identifies the object, describes its location and reveals its assignments. Right clicking on an area object means to position your mouse pointer over the object and click the right (not left as you usually do) button on the mouse. When you do this the Area Information dialog box appears.

The units drop-down box in the Area Information dialog box allows you to change the current units without leaving the dialog box. Any change in units that you make while in this dialog box is permanent; that is, the change remains when you exit the dialog box.

The area at the top of the Area Information dialog box that is named Identification provides basic information that identifies the area object. This information includes:

- **Label:** This is the area object label assigned by ETABS. See the previous subsection in this chapter titled "Automatic Area Object Labeling" for more information.
- **Story:** This is the story level associated with the area object.
- **Area Type:** This is the area object type. It can be Floor, Wall, Ramp or Null. See the previous subsection in this chapter titled "Area Type" for more information.



Tip:

Click on a tab in the Area Information dialog box to see the data associated with the tab.

The Area Information dialog box also has three different tabs located just above the Identification area. Each of these tabs displays different types of information. The tabs are labeled Location, Assignments and Loads. These labels are relatively self-explanatory. The Location tab includes information that locates the area object. The Assignments tab includes information on all area object assignments except for loads. The Loads tab includes information on all load assignments to the area object. The items displayed in each of these tabs are detailed in the subsections below.

Location Tab in the Area Information Dialog Box

The following geometric information about the area object is displayed on the Location tab:

- **Area:** This is the total area of the area object in Length² units.
- **Perimeter:** This is the total length of the perimeter of the area object in Length units.
- **Centroid X:** This is the global X coordinate of the centroid of the area object. This value is only provided for Floor-type area objects and horizontal Null-type area objects.

- **Centroid Y:** This is the global Y coordinate of the centroid of the area object. This value is only provided for Floor-type area objects and horizontal Null-type area objects.
- **Polar Inertia ($I_x + I_y$):** This is the $I_x + I_y$ value for the area object calculated about the centroid of the area object and reported in Length⁴ units. This value is only provided for Floor-type area objects and horizontal Null-type area objects.

This is essentially the mass moment of inertia (MMI) of the area object assuming that the mass per unit area of the object is 1. Note that you multiply the $I_x + I_y$ item by the mass per unit area of the area object to get the MMI for the area object. Refer to the general diaphragm shown in Figure 14-3 in Chapter 14 for more information.

- **No. of Points:** This is the total number of corner points in the area object.

In addition, for each corner point of the area object the following information is displayed on the Location tab:

- **Point n:** Here n in the label varies from 1 to the total number of corner points. The data entered in this box is the label of the corner point object.
- **Story:** This is the story level associated with the corner point object.
- **X:** This is the global X coordinate of the corner point object.
- **Y:** This is the global Y coordinate of the corner point object.
- **Delta Z:** This is the vertical distance from the story level that the point object is associated with to the point object. The value of Delta Z is always between zero and the value of the story height associated with the point object, inclusive. Delta Z is always positive even though it is measured from the story level downward.

Assignments Tab in the Area Information Dialog Box

Following are the items that appear (or in some cases may appear) on the Assignments tab:

- **Shell section prop.:** If a wall, slab or deck section property is *not* assigned to the object then this item displays "None." Otherwise it displays the name of the wall, slab or deck section property. The shell section property is assigned using the **Assign menu > Shell/Area > Wall/Slab/Deck Section** command. See the subsection titled "Wall, Slab and Deck Section Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.
- **Opening:** This item only appears if the area object is designated as an opening using the **Assign menu > Shell/Area > Openings** command. In this case it either displays "Loaded opening" or "Unloaded opening" depending on the type of opening. See the subsection titled "Opening Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

When an opening is assigned to an area object the only two assignments shown on the Assignments tab are Opening and Group. No other assignments are shown on the tab in this case because no other assignments to the object are meaningful when it is designated as an opening.

- **Rigid diaphragm:** If a rigid diaphragm constraint is *not* assigned to the object then this item displays No. Otherwise it displays the name of the rigid diaphragm constraint. The rigid diaphragm is assigned using the **Assign menu > Shell/Area > Rigid Diaphragm** command. See the subsection titled "Rigid Diaphragm Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

- **Local axis 2 angle:** Refer to the section titled "Default Area Object Local Axes" later in this chapter for a description of the default area object local axes. Note that you can use the **Assign menu > Shell/Area > Local Axes** command to rotate the local 1 and 2 axes from their default orientation (rotation about the local 3 axis). If the local 1 and 2 axes have not been rotated from their default orientation then this item says "Default." Otherwise it displays "X° from Default" where X is the number of degrees the axes have been rotated from their default position. See the subsection titled "Local Axes Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.
- **Stiffness Modifiers:** The default value for the shell stiffness modifiers is 1 which indicates that no modification are made. If all of the shell stiffness modifiers are 1 then this item displays "None." Otherwise this item is replaced with one or more of the six items listed below. Any of the six stiffness modifiers that are not 1 are listed.
 - ✓ **Stiff. Modifier, f11.** If this item is not 1 then the item and the modifier value are listed.
 - ✓ **Stiff. Modifier, f22.** If this item is not 1 then the item and the modifier value are listed.
 - ✓ **Stiff. Modifier, f12.** If this item is not 1 then the item and the modifier value are listed.
 - ✓ **Stiff. Modifier, m11.** If this item is not 1 then the item and the modifier value are listed.
 - ✓ **Stiff. Modifier, m22.** If this item is not 1 then the item and the modifier value are listed.
 - ✓ **Stiff. Modifier, m12.** If this item is not 1 then the item and the modifier value are listed.

The stiffness modifiers are assigned using the **Assign menu > Shell/Area > Shell Stiffness Modifiers** command. See the subsection titled "Shell Stiffness Modifi-

ers Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

- **Pier:** If a pier label is *not* assigned to the area object then this item displays "No." Otherwise it displays the name of the pier label. The pier label is assigned using the **Assign menu > Shell/Area > Pier Label** command. See the subsection titled "Pier Label Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information. Note that pier labels can only be assigned to wall-type area objects.
- **Spandrel:** If a spandrel label is *not* assigned to the area object then this item displays "No." Otherwise it displays the name of the spandrel label. The spandrel label is assigned using the **Assign menu > Shell/Area > Spandrel Label** command. See the subsection titled "Spandrel Label Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information. Note that spandrel labels can only be assigned to wall-type area objects.
- **Area Springs:** If no area springs are assigned to the object then this item displays "None." Otherwise this item is replaced with one or more of the three items listed below. Any of the three local directions of the area springs that has an assignment is listed.
 - ✓ **Area Spring Local-1.** If there is an area spring stiffness assigned in the local 1 direction then this item and the spring stiffness are listed.
 - ✓ **Area Spring Local-2.** If there is an area spring stiffness assigned in the local 2 direction then this item and the spring stiffness are listed.
 - ✓ **Area Spring Local-3.** If there is an area spring stiffness assigned in the local 3 direction then this item and the spring stiffness are listed.

The area springs are assigned using the **Assign menu > Shell/Area > Area Springs** command. See the subsection titled "Area Spring Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

- **Area Mass:** If an additional area mass is *not* assigned to the object then this item displays "None." Otherwise it displays the value of the additional area mass. The area mass is assigned using the **Assign menu > Shell/Area > Additional Area Mass** command. See the subsection titled "Additional Area Mass Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

Note that the area mass value is displayed here regardless of the mass source you have specified using the **Define menu > Mass Source** command. If you designate that the mass source is from a specified load combination then the value shown here is not used by the program. See the section titled "Mass Source" in Chapter 11 and the section titled "Mass" in Chapter 27 for more information.

- **Automatic Mesh:** This item indicates whether or not the area object is automatically meshed into the analysis model by ETABS. It displays either Yes or No. See Chapter 30 and the subsection titled "Automatic Membrane Floor Mesh/No Mesh Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

Note that ETABS can only automatically mesh floor-type area objects with membrane properties only into the analysis model.

- **Group:** This item displays the groups that the object belongs to. By default in ETABS all objects belong to the default group called ALL. You can not delete the group named ALL and you can not remove an object from the group. Thus even if you have not assigned the area object to any groups you will see ALL for this item indicating that it belongs to the ALL group. If you have

assigned the object to one or more groups then additional Group lines are provided, one for each group to which you have assigned the object.

Loads Tab in the Area Information Dialog Box

Following are the items that may appear on the Loads tab:

- **Load:** If no loads are assigned to the area object then this displays "None." Otherwise this item is not displayed.
- **Static Load Case:** This line indicates the name of the static load case to which the load(s) listed on the immediately following line(s) of the Loads tab is (are) applied. For area objects either uniform or temperature loads may be listed. The bulleted items below list the area object loads you may see immediately following the Static Load Case line. One or more of these load lines may be listed for each static load case depending on the assignments you have made.

Note that the area object loads are assigned using the **Assign menu > Shell/Area Loads** command.

- ✓ **Uniform F1:** This line indicates the value of a uniform surface load in the local 1-axis direction of the area object that is assigned to the object.
- ✓ **Uniform F2:** This line indicates the value of a uniform surface load in the local 2-axis direction of the area object that is assigned to the object.
- ✓ **Uniform F3:** This line indicates the value of a uniform surface load in the local 3-axis direction of the area object that is assigned to the object.
- ✓ **Uniform FX:** This line indicates the value of a uniform surface load in the global X direction that is assigned to the area object.

- ✓ **Uniform FY:** This line indicates the value of a uniform surface load in the global Y direction that is assigned to the area object.
- ✓ **Uniform FGrav:** This line indicates the value of a uniform surface load in the gravity (negative global Z) direction that is assigned to the area object.
- ✓ **Uniform FX Proj:** This line indicates the value of a uniform projected surface load in the global X direction that is assigned to the area object.
- ✓ **Uniform FY Proj:** This line indicates the value of a uniform projected surface load in the global Y direction that is assigned to the area object.
- ✓ **Uniform FGrav Proj:** This line indicates the value of a uniform projected surface load in the Gravity (negative global Z) direction that is assigned to the area object.
- ✓ **Temperature:** This line indicates the value of a temperature change that is assigned to the area object.
- ✓ **Add Point Temp:** This line indicates whether or not you have indicated that ETABS is to consider previously specified point object temperature changes at the corners of the area object. This line either displays "Yes" or "No."

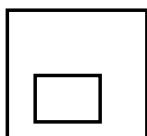
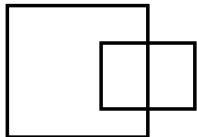
Static load case names are defined using the **Define menu > Static Load Cases** command. See the section titled "Static Load Cases" in Chapter 11 for more information on static load cases.

Uniform surface loads are assigned to area objects using the **Assign menu > Shell/Area Loads > Uniform** command. See the subsection titled "Uniform Surface Load Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

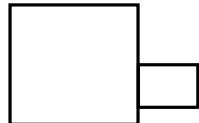
Temperature loads are assigned to area objects using the **Assign menu > Shell/Area Loads > Temperature** command. See the subsection titled "Temperature Load Assignments to Area Objects" under the section titled "Assignments to Area Objects" in Chapter 14 for more information.

Overlapping Area Objects

When drawing area objects to create your model you can overlap (intersect) them and you can draw one area object inside another. However, when your model is complete and ready for analysis the following restrictions must be met. If these restrictions are not met, then an error will occur when you run the analysis.



- No two area objects of any kind can intersect. Intersecting area objects occur when the line representing an edge of one area object intersects the line representing an edge of another area object. The sketch to the left shows an example of two intersecting area objects.
- An area object with structural properties (wall, slab or deck properties assigned) can not be enclosed by another area object with structural properties. The sketch to the left shows an example of one area object that is enclosed by another. It is acceptable for a *null-type* area object to be completely enclosed by another area object (of any type).
- Area objects designated as openings must be completely enclosed by an area object that ETABS is going to automatically mesh into the analysis model. Note that ETABS only automatically meshes area objects with deck section properties and area objects with slab section properties that have membrane behavior only into the analysis model. Further note that ETABS only automatically meshes these types of area objects as long as you have not specified that they are not to be meshed.



- Two area objects with different rigid diaphragm assignments can not intersect, be one inside the other or have a common edge. The sketch to the left shows an example of two area objects with a common edge.

23

Important note: To reiterate, the above restrictions only apply when the analysis is ready to be run. They do not apply while you are in the process of creating the model.

Following is some other useful information about overlapping area objects:

- When one null-type area object is completely enclosed by another area object of any type, any loads on the two objects are additive.
- When one null-type area object is completely enclosed by another area object of any type, additional area mass on the two objects is additive.
- When one null-type area object is completely enclosed by another area object of any type, any area spring stiffnesses on the two objects are additive.

Plan Views of Walls



Tip:

You can draw and/or select a wall in plan view.

In a plan view of a story level, vertical area objects below that are assigned wall properties appear as lines that represent a plan section (to scale) through the wall. While in a plan view you can use either the **Draw menu > Draw Area Objects > Draw Walls (plan)** command or the **Draw menu > Draw Area Objects > Create Walls in Region or at Click (plan)** command, or one of the associated toolbar buttons on the side toolbar to draw walls. This is a very convenient method for drawing walls. See the section titled "Drawing Area Objects" in Chapter 12 for more information.

Default Area Object Local Axes

The following subsections describe the default local axes orientation for vertical, horizontal and other (neither vertical nor horizontal) area objects. Note that you can use the **Assign menu > Shell/Area > Local Axes** command to rotate the local 1 and 2 axes about the local 3 axes from their default orientation.

Default Orientation for Horizontal Area Objects



Tip:

When local axes are displayed on the screen local axis 1 is red, local axis 2 is white and local axis 3 is blue, always. This is the same order as the colors of the American flag: red, white and blue.

For horizontal area objects the local axes have the following default orientation:

- **Local axis 1:** This axis is in the plane of the area object. The positive local 1 axis is in the same direction as the positive global X-axis.
- **Local axis 2:** This axis is in the plane of the area object. The positive local 2 axis is in the same direction as the positive global Y-axis.
- **Local axis 3:** This axis is perpendicular to the plane of the area object. The positive local 3 axis points upward. It is in the same direction as the positive global Z-axis.

Default Orientation for Vertical Area Objects



Tip:

*You can toggle on a view of the area object local axes using the **View menu > Set Building View Options** command or using the  **Set Building View Options** button.*

For vertical area objects the local axes have the following default orientation:

- **Local axis 1:** This axis is in the plane of the area object. The positive local 1 axis is in the same direction as the positive global Z-axis, upward.
- **Local axis 2:** This axis is in the plane of the area object. The projection of the positive local 2 axis onto the global X-axis is in the same direction as the positive global X-axis. If the area object is in the global YZ plane such that there is no projection onto the global X-axis then the positive direction of the local 2 axis is parallel to the positive global Y-axis.

- **Local axis 3:** This axis is perpendicular to the plane of the area object. Its direction is determined from applying the right-hand rule using the directions of the 1 and 2 axes described above. See the section titled “The Right Hand Rule” later in this chapter for more information.

Default Orientation for Other Area Objects

For area objects that are not vertical and are not horizontal (for example, ramp-type area objects) the local axes have the following default orientation:

- **Local axis 1:** This axis is in the plane of the area object. The positive local 1 axis is in the same direction as the positive global X-axis.
- **Local axis 2:** This axis is in the plane of the area object. Its direction is determined from applying the right-hand rule using the directions of the 1 and 3 axes. See the section titled “The Right Hand Rule” later in this chapter for more information.
- **Local axis 3:** This axis is perpendicular to the plane of the area object. The positive local 3 axis has an upward sense. The projection of the positive local 3 axis onto the global Z-axis is in the same direction as the positive global Z-axis, upward.

The Right Hand Rule

The right hand rule is used for two main purposes. One is to determine the positive directions of coordinate system axes. The other is to determine the positive direction of moments and rotations in a coordinate system.

Important note: Do not confuse the positive direction of moments in a coordinate system with the positive directions for analysis output forces. The positive directions for analysis output forces are described for various elements in Chapters 34 through 39.

Positive Direction of Coordinate System Axes

The global (X-Y-Z) and local (1-2-3) coordinate systems used in ETABS are right handed coordinate systems. The right hand rule applies in these coordinate systems. Following is an explanation of the right hand rule as it applies to coordinate system axes in ETABS.



Tip:

As silly as it may sound, make sure you are using your right hand and not your left hand when applying the right hand rule. A common mistake when you are in a hurry is to use the wrong hand.

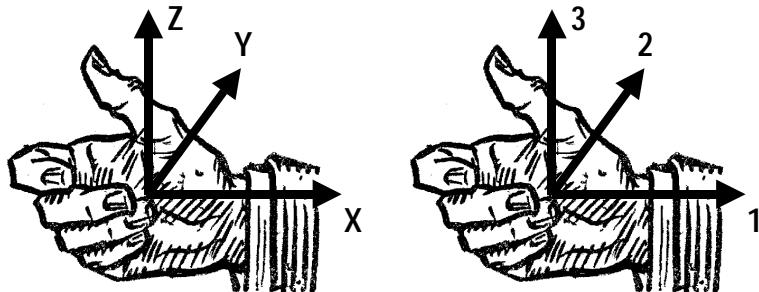
Global Coordinate System

- Take your right hand and point your thumb in the direction of the positive global Z-axis. Your fingers then wrap in such a way that they push the positive global X-axis into the positive global Y-axis. This is illustrated in Figure 23-2a.
- Take your right hand and point your thumb in the direction of the positive global Y-axis. Your fingers then wrap in such a way that they push the positive global Z-axis into the positive global X-axis.
- Take your right hand and point your thumb in the direction of the positive global X-axis. Your fingers then wrap in such a way that they push the positive global Y-axis into the positive global Z-axis.

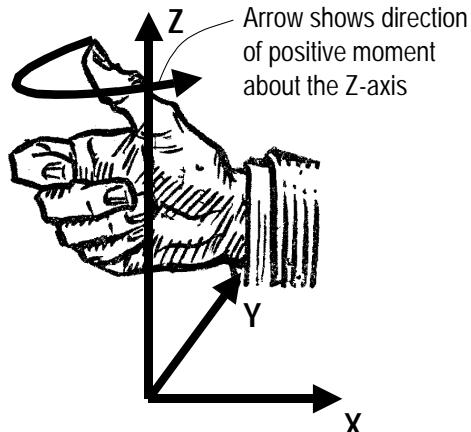
Local Coordinate System

- Take your right hand and point your thumb in the direction of the positive global 3-axis. Your fingers then wrap in such a way that they push the positive global 1-axis into the positive global 2-axis. This is illustrated in Figure 23-2b.
- Take your right hand and point your thumb in the direction of the positive global 2-axis. Your fingers then wrap in such a way that they push the positive global 3-axis into the positive global 1-axis.
- Take your right hand and point your thumb in the direction of the positive global 1-axis. Your fingers then wrap in such a way that they push the positive global 2-axis into the positive global 3-axis.

Figure 23-2:
Illustrations of the
right hand rule



- a) Thumb points in +Z-direction
and fingers wrap +X axis
into +Y axis
- b) Thumb points in +Z-direction
and fingers wrap +1 axis into
+2 axis



- c) Thumb points in +Z-direction
and fingers wrap in direction of
positive moment about the Z-axis

Positive Direction of Moments in a Coordinate System

The positive direction of forces in a coordinate system is the same as the positive direction of the coordinate system axes. The positive direction of moments is determined by applying the right hand rule as described below.

- Take your right hand and point your thumb in the direction of the positive global Z-axis. Your fingers then wrap around the positive global Z-axis in the direction of a

positive moment about the global Z-axis. This is illustrated in Figure 23-2c.

- The same concept applies to the global X, global Y, local 1, local 2 and local 3 axes.



Chapter 24

Line Objects

General

Line objects are defined by two end points connected by a straight-line segment. You can right click on a line object to bring up the Line Information dialog box where you can view information about the line including its length, exact location and assignments. See the section titled “Right Click Information for Line Objects” later in this chapter for more information.

Frame Section Properties

ETABS has a built-in frame section property called CSEC1 which is a 12 inch by 24 inch solid *concrete* element. It also has five default auto select section property lists named LatCol, LatBm, GravCol, GravBm, and SecBm. The CSEC1 property and the five auto select section property lists can not be deleted. They can however be modified, except you can't change the material property (concrete) associated with CSEC1. You can also define other section properties and auto select section lists. For

additional information on auto select section lists see the subsection titled “Adding User-Defined Frame Section Properties” in the section titled “Frame Section Properties” in Chapter 11 of this manual.

24

Auto select section lists are used in the Steel Frame Design and Composite Beam Design postprocessors. Thus the five auto select section lists are filled with steel section properties. You can delete steel sections from the five default auto selection lists as long as you leave at least one steel section in each of them. In other words you can not delete the last steel section from any of the five default auto select section lists.

You can also add as many sections as you want to these lists. We recommend that for optimum performance of the design postprocessors (speed) you try to limit the number of sections you include in an auto select section list. Putting between 10 and 40 sections is quite reasonable for an auto select section list. Putting all of the sections in the AISC database is not a great idea, though it is possible.

The five default auto select section lists are used as default sections in the steel structure template that is available when you click the **File menu > New Model** command or the **Edit menu > Add to Model from Template > Add 3D Frame** command.

The CSEC1 concrete section is used as the default section in the concrete structure templates that are available when you click the **File menu > New Model** command or the **Edit menu > Add to Model from Template > Add 3D Frame** command.

We recommend that you modify these five default auto select section lists and the CSEC1 section to best suit your own purposes and save them in the Default.edb file. Use the **Define menu > Frame Sections** command to modify the section properties. See the subsection titled “Initialization of a New Model” in the section titled “Starting a New Model” in Chapter 8 for more information on the important topic of the Default.edb file and other initialization files.

Line Object Labeling and Line Type

When you first draw a line object ETABS assigns a line type and a label to the line object. ETABS also assigns a section property to the line object as it is drawn. You can control the section property assigned when the object is drawn through the floating Properties of Object box. See the section titled "Drawing Line Objects" in Chapter 12 for more information on this method of line object property assignment. Property assignments can be modified later by selecting the line object and clicking the **Assign menu > Frame/Line > Frame Section** command. See the subsection titled "Frame Section Assignments to Line Objects" in Chapter 14 for more information on this method of assigning and modifying line object properties.

The following two subsections discuss the line type and label assignments.

Line Type

If a frame section property is *not* assigned to the line object as it is drawn then the section is assigned a line type of "Null." If a frame section property is assigned to the line object as it is drawn then the section is assigned a line type of either "Column", "Beam" or "Brace." The following logic is used to determine which of these frame types to assign:

- **Column:** If the line object is vertical (parallel to the global Z-axis) then it is given a line type of "Column."
- **Beam:** If the line object is horizontal (parallel to the global X-Y plane) then it is given a line type of "Beam."
- **Brace:** If the line object is not vertical and not horizontal then it is given a line type of "Brace."

The line type is used to determine the prefix used in the automatic labeling of the line object, as described in the following subsection. The line type also determines how objects may be treated for design.



Note:

Line objects are always given a line type by ETABS based on their orientation and property assignment. The possible line types are Column, Beam, Brace and Null.

You can not directly change the line type associated with a line object. However, if you change the orientation of the object then ETABS automatically changes the line type. For example, suppose you have a line object defined in a horizontal plane. This is given an line type of Beam by ETABS. Now suppose you drop one of the end points of the line object one story level to create a sloping line object. ETABS immediately changes the line type for the object from Beam to Brace. As mentioned in the following subsection, the label is also changed.

Automatic Line Object Labeling

ETABS automatically labels line objects based on their line type as follows:

- Line objects with a line type of Column are given a label starting with the letter C followed by a number, such as C21.
- Line objects with a line type of Beam are given a label starting with the letter B followed by a number, such as B21.
- Line objects with a line type of Brace are given a label starting with the letter D followed by a number, such as D21. The letter D stands for diagonal. The letter B for brace is not used because B is already used for beams.
- Line objects with a line type of Null are given a label starting with the letter L followed by a number, such as L21. The letter L stands for line.

 **Note:**
ETABS automatically labels line objects.

If the line type of a line object changes then ETABS automatically relabels the line object using the correct designation as described above. There are no exceptions to this rule.

In general you can not assign your own labels to line objects. Only the automatic ETABS labels are allowed. There is one exception to this. If you import a text file into ETABS then that text file can have line object labels different from the automatic labels described above. Be aware though that in this case, if you cause the line type assigned to a line object to change, or move the line object, ETABS will automatically relabel the object.

ETABS uses an intelligent labeling scheme. Typically it gives the same label to line objects located at the same plan location and at the same elevation relative to their respective story level, but at different story levels. Note that line objects are always identified by a story level and a label. This type of labeling scheme makes it much easier for you to locate objects. For example line object B2 at one level will be directly above line object B2 at another story level.

You can use the **Edit menu > Auto Relabel All** command to automatically relabel all area, line and point objects. See the subsection titled "Relabeling Objects" in Chapter 23 for additional information on this.

Assignments Made to Line Objects

As previously mentioned you can assign specific section properties to line objects as you draw them. The properties assigned are controlled through the floating Properties of Object box. See the section titled "Drawing Line Objects" in Chapter 12 for more information.

You can modify previous section property assignments and make many other types of assignments to line objects through the Assign menu. The types of line object assignments you can make include:



Tip:

To make an assignment to a line object first select the object and then click the appropriate command on the Assign menu.

- Assign frame section properties.
- Assign frame member end releases or partial end fixity.
- Assign end offsets (rigid or flexible) along the length of frame members to model the finite size of beam-column joints.
- Assign frame member end joint offsets to model such conditions as a beam framing into a column eccentrically (off center).

- Designate frame output stations. These are locations along a frame element where output forces are calculated and design is performed.
- Specify the line object local axes orientation.
- Specify frame property modifiers.
- Assign link element properties.
- Assign frame nonlinear hinges (pushover).
- Assign pier labels.
- Assign spandrel labels.
- Assign line spring supports.
- Assign additional line mass (if the mass source is specified as From Element and Additional Masses using the **Define menu > Mass Source** command).
- Indicate if a frame element is *not* to be automatically meshed by ETABS.
- Assign point loads along line objects.
- Assign distributed (uniform and trapezoidal) loads along line objects.
- Assign temperature loads.

Use either the **Assign menu > Frame/Line** or the **Assign menu > Frame/Line Loads** command to make these assignments. See the section titled "Assignments to Line Objects" in Chapter 14 for more information.

Right Click Information for Line Objects

Right click on a line object to view useful information that identifies the object, describes its location and reveals its assignments. Right clicking on a line object means to position your mouse pointer over the object and click the right (not left as you usually do) button on the mouse. When you do this the Line Information dialog box appears.



Tip:

Right click on a line object to view useful information that identifies the object, describes its location and reveals its assignments.

The units drop-down box in the Line Information dialog box allows you to change the current units without leaving the dialog box. Any change in units that you make while in this dialog box is permanent; that is, the change remains when you exit the dialog box.

The area at the top of the Line Information dialog box that is named Identification provides basic information that identifies the line object. This information includes:

- **Label:** This is the line object label assigned by ETABS. See the previous subsection in this chapter titled "Automatic Line Object Labeling" for more information.
- **Story:** This is the story level associated with the line object.
- **Line Type:** This is the line object type. It can be Column, Beam, Brace or Null. See the previous subsection in this chapter titled "Line Type" for more information.
- **Design Procedure:** This item shows which design post-processor the line object is tagged to be designed by. For line objects the choices for this item are either Steel Frame Design, Concrete Frame Design, Composite Beam Design or Null. If desired, use the **Design menu > Overwrite Frame Design Procedure** command to change the Design Procedure for frame elements. See the section titled "Overwrite Frame Design Procedure" in Chapter 17 for more information.

The Line Information dialog box also has three different tabs located just above the Identification area. Each of these tabs dis-

plays different types of information. The tabs are labeled Location, Assignments and Loads. These labels are relatively self-explanatory. The Location tab includes information that locates the line object. The Assignments tab includes information on all line object assignments except for loads. The Loads tab includes information on all load assignments to the line object. The items displayed in each of these tabs are detailed in the subsections below.

Location Tab in the Line Information Dialog Box



Tip:

Click on a tab in the Line Information dialog box to see the data associated with the tab.

The following geometric information about the line object is displayed on the Location tab:

- **Length:** This is the total end point to end point length of the line object.
- **Start point (I):** The data entered in this box is the label of the end point object at the start end (i-end) of the line.
- **End point (J):** The data entered in this box is the label of the end point object at the ending end (j-end) of the line.

In addition, for the start and end points of the line object the following information is displayed on the Location tab:

- **Story:** This is the story level associated with the point object at the designated end of the line object.
- **X:** This is the global X coordinate of the point object at the designated end of the line object.
- **Y:** This is the global Y coordinate of the point object at the designated end of the line object.
- **Delta Z:** This is the vertical distance from the story level that the designated point object is associated with to the point object. The value of Delta Z is always between zero and the value of the story height associated with the point object, inclusive. Delta Z is always positive even though it is measured from the story level downward.

Assignments Tab in the Line Information Dialog Box

Following are the items that appear (or in some cases may appear) on the Assignments tab:

- **Section property:** If a frame section property is *not* assigned to the object then this item displays "None." Otherwise it displays the name of the frame section property. If the frame section property assigned is an auto select list then the name of the current analysis section property is displayed along with the name of the auto select list in parenthesis. The frame section property is assigned using the **Assign menu > Frame/Line > Frame Section** command. See the subsection titled "Frame Section Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Releases:** If there are no frame element end releases assigned to the line object then this line says "None." If any releases are assigned to the object then the Releases line is replaced with one or both of the lines listed below. The frame element end releases are assigned using the **Assign menu > Frame/Line > Frame Releases/Partial Fixity** command. See the subsection titled "Frame Releases and Partial Fixity Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
 - ✓ **Releases End-I:** This line lists whatever releases are assigned to the i-end of the frame element. If no releases exist at the i-end then this line is not displayed. The possible releases are P (axial), V2 (shear in the local 2 direction, also known as major shear), V3 (shear in the local 3 direction, also known as minor shear), T (torsion), M2 (moment about the local 2-axis, also known as minor moment), and M3 (moment about the local 3-axis, also known as major moment).

- ✓ **Releases End-j:** This line lists whatever releases are assigned to the j-end of the frame element. If no releases exist at the j-end then this line is not displayed. The possible releases are the same as those listed in the bullet item above for the i-end.
- **Partial fixity springs:** If there are no frame element partial fixity springs assigned to the line object then this line says "None." If any partial fixity springs are assigned to the object then the Partial fixity springs line is replaced with one or more of the lines listed below. The frame element partial fixity springs are assigned using the **Assign menu > Frame/Line > Frame Releases/Partial Fixity** command. See the subsection titled "Frame Releases and Partial Fixity Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
 - ✓ **End-I spring P:** This line lists the spring stiffness for the P (axial) partial fixity spring at the i-end of the line object.
 - ✓ **End-I spring V2:** This line lists the spring stiffness for the V2 (shear in the local 2 direction, also known as major shear) partial fixity spring at the i-end of the line object.
 - ✓ **End-I spring V3:** This line lists the spring stiffness for the V3 (shear in the local 3 direction, also known as minor shear) partial fixity spring at the i-end of the line object.
 - ✓ **End-I spring T:** This line lists the spring stiffness for the T (torsion) partial fixity spring at the i-end of the line object.
 - ✓ **End-I spring M2:** This line lists the spring stiffness for the M2 (moment about the local 2-axis, also known as minor moment) partial fixity spring at the i-end of the line object.

- ✓ **End-I spring M3:** This line lists the spring stiffness for the M3 (moment about the local 2-axis, also known as major moment) partial fixity spring at the i-end of the line object.
- ✓ **End-J spring P:** This line lists the spring stiffness for the P (axial) partial fixity spring at the j-end of the line object.
- ✓ **End-J spring V2:** This line lists the spring stiffness for the V2 (shear in the local 2 direction, also known as major shear) partial fixity spring at the j-end of the line object.
- ✓ **End-J spring V3:** This line lists the spring stiffness for the V3 (shear in the local 3 direction, also known as minor shear) partial fixity spring at the j-end of the line object.
- ✓ **End-J spring T:** This line lists the spring stiffness for the T (torsion) partial fixity spring at the j-end of the line object.
- ✓ **End-J spring M2:** This line lists the spring stiffness for the M2 (moment about the local 2-axis, also known as minor moment) partial fixity spring at the j-end of the line object.
- ✓ **End-J spring M3:** This line lists the spring stiffness for the M3 (moment about the local 2-axis, also known as major moment) partial fixity spring at the j-end of the line object.
- **End length offsets:** If there are no frame element end length offsets (both set to zero) then this line says "None." If they are specified to be automatically determined by ETABS then this line says "Automatic." If any non-zero end length offsets are assigned to the object then this line says "Defined" and the three additional lines listed below also appear. The frame element end length offsets are assigned using the **Assign menu > Frame/Line > Frame Rigid Offsets** command. See the subsection titled "Rigid End Offsets Along the Length of Frame Elements" under the subsection titled "Frame

Rigid Offset Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

- ✓ **End-I length offset:** This is the length of the end offset along the length of the frame element at the i-end of the element.
- ✓ **End-J length offset:** This is the length of the end offset along the length of the frame element at the j-end of the element.
- ✓ **Rigid zone factor:** This is the rigid zone factor, that is, the proportion of the end offset length that is treated as rigid.
- **Joint offsets:** If there are no frame joint offsets assigned to the line object then this line says "None." If any joint offset is assigned to the object then the Joint offsets line is replaced with one or more of the lines listed below. The frame element joint offsets are assigned using the **Assign menu > Frame/Line > Frame Rigid Offsets** command. See the subsection titled "Rigid Frame Joint Offsets" under the subsection titled "Frame Rigid Offset Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
 - ✓ **End-I joint offset X:** This is the joint offset of the i-end of the frame element in the global X direction.
 - ✓ **End-I joint offset Y:** This is the joint offset of the i-end of the frame element in the global Y direction.
 - ✓ **End-I joint offset Z:** This is the joint offset of the i-end of the frame element in the global Z direction.
 - ✓ **End-J joint offset X:** This is the joint offset of the j-end of the frame element in the global X direction.

- ✓ **End-J joint offset Y:** This is the joint offset of the j-end of the frame element in the global Y direction.
- ✓ **End-J joint offset Z:** This is the joint offset of the j-end of the frame element in the global Z direction.
- **Min number stations:** This item is displayed if the output stations along the frame element are specified by a minimum number of stations. It indicates the minimum number of output stations to be used for the frame element. If this item is not displayed then the Max. station spacing item is displayed. The frame element output station parameters are assigned using the **Assign menu > Frame/Line > Frame Output Stations** command. See the subsection titled "Frame Output Station Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Max. station spacing:** This item is displayed if the output stations along the frame element are specified by a maximum station spacing. It indicates the maximum spacing of output stations to be used along the frame element. If this item is not displayed then the Min. number stations item is displayed. The frame element output station parameters are assigned using the **Assign menu > Frame/Line > Frame Output Stations** command. See the subsection titled "Frame Output Station Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Local axis 2 angle:** Refer to the section titled "Default Line Object Local Axes" later in this chapter for a description of the default line object local axes. Note that you can use the **Assign menu > Frame/Line > Local Axes** command to rotate the local 2 and 3 from their default orientation axes (rotation about the local 1-axis). If the local 2 and 3 axes have not been rotated from their default orientation then this item says "Default." Otherwise it displays " X° from Default" where X is the number of degrees the axes have been rotated from their default position. See the subsection titled "Local Axes Assignments to Line Objects" under the section titled "As-

signments to Line Objects" in Chapter 14 for more information.

- **Property modifiers:** If there are no frame property modifiers assigned to the line object then this line says "None." If any frame property modifier is assigned to the object then the Property modifiers line is replaced with one or more of the lines listed below. The frame property modifiers are assigned using the **Assign menu > Frame/Line > Frame Property Modifiers** command. See the subsection titled "Frame Property Modifier Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
 - ✓ **Prop. modifier, Area:** This is the frame property modifier for the axial area.
 - ✓ **Prop. modifier, A2:** This is the frame property modifier for the shear area in the local 2-axis direction.
 - ✓ **Prop. modifier, A3:** This is the frame property modifier for the shear area in the local 3-axis direction.
 - ✓ **Prop. modifier, J:** This is the frame property modifier for the torsional constant.
 - ✓ **Prop. modifier, I2:** This is the frame property modifier for the moment of inertia about the local 2-axis.
 - ✓ **Prop. modifier, I3:** This is the frame property modifier for the moment of inertia about the local 3-axis.

- **Link properties:** If a link element property is *not* assigned to the object then this item displays "None." Otherwise it displays the name of an assigned link element property. Note that you can assign more than one link element property to the same line object. The assignments are additive; they do not replace one another. When more than one link element is assigned to a line object multiple Link properties lines are used, one for each property. The link element property is assigned using the **Assign menu > Frame/Line > Link Properties** command. See the subsection titled "Link Property Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Nonlinear hinges:** If a nonlinear hinge (pushover) property is *not* assigned to the object then this item displays "None." Otherwise it displays the name of an assigned nonlinear hinge and its location along the line object measured from the i-end of the object. Note that you can assign more than one nonlinear hinge to the same line object. The assignments are additive; they do not replace one another. When more than one nonlinear hinge is assigned to a line object multiple Nonlinear Hinge lines are used, one for each hinge. The nonlinear hinge is assigned using the **Assign menu > Frame/Line > Frame Nonlinear Hinges** command. See the subsection titled "Frame Nonlinear Hinge Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Pier:** If a pier label is *not* assigned to the line object then this item displays "No." Otherwise it displays the name of the pier label. The pier label is assigned using the **Assign menu > Frame/Line > Pier Label** command. See the subsection titled "Pier Label Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

- **Spandrel:** If a spandrel label is *not* assigned to the line object then this item displays "No." Otherwise it displays the name of the spandrel label. The spandrel label is assigned using the **Assign menu > Frame/Line > Spandrel Label** command. See the subsection titled "Spandrel Label Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Line Springs:** If no line springs are assigned to the object then this item displays "None." Otherwise this item is replaced with one or more of the three items listed below. Any of the three local directions of the line springs that has an assignment is listed.
 - ✓ **Line Spring Local-1.** If there is a line spring stiffness assigned in the local 1 direction then this item and the spring stiffness are listed.
 - ✓ **Area Spring Local-2.** If there is a line spring stiffness assigned in the local 2 direction then this item and the spring stiffness are listed.
 - ✓ **Area Spring Local-3.** If there is a line spring stiffness assigned in the local 3 direction then this item and the spring stiffness are listed.

The line springs are assigned using the **Assign menu > Frame/Line > Line Springs** command. See the subsection titled "Line Spring Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

- **Line Mass:** If an additional line mass is *not* assigned to the object then this item displays "None." Otherwise it displays the value of the additional line mass. The line mass is assigned using the **Assign menu > Frame/Line > Additional Line Mass** command. See the subsection titled "Additional Line Mass Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

Note that the line mass value is displayed here regardless of the mass source you have specified using the **Define menu > Mass Source** command. If you designate that the mass source is from a specified load combination then the value shown here is not used by the program. See the section titled "Mass Source" in Chapter 11 and the section titled "Mass" in Chapter 27 for more information.

- **Automatic Mesh:** This item indicates whether or not the line object is automatically meshed by ETABS. It displays either Yes or No. See Chapter 30 and the subsection titled "Automatic Frame Mesh/No Mesh Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.
- **Group:** This item displays the groups that the object belongs to. By default in ETABS all objects belong to the default group called ALL. You can not delete the group named ALL and you can not remove an object from the group. Thus even if you have not assigned the line object to any groups you will see ALL for this item indicating that it belongs to the ALL group. If you have assigned the object to one or more groups then additional Group lines are provided, one for each group to which you have assigned the object.

Loads Tab in the Line Information Dialog Box

Following are the items that may appear on the Loads tab:

- **Load:** If no loads are assigned to the line object then this displays "None." Otherwise this item is not displayed.
- **Static Load Case:** This line indicates the name of the static load case to which the load(s) listed on the immediately following line(s) of the Loads tab is (are) applied. For line objects either point, uniform, trapezoidal or temperature loads may be listed. The bulleted items below list the line object loads you may see immediately following the Static Load Case line. One or more of

these load lines may be listed for each static load case depending on the assignments you have made.

Note that the line object loads are assigned using the **Assign menu > Frame/Line Loads** command. Both uniform and trapezoidal loads are assigned using the **Assign menu > Frame/Line Loads > Distributed** command. See the subsection titled "Distributed Load Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

- ✓ **Point F1:** This line indicates the value of the point force load in the local 1-axis direction and its location along the line object measured from the i-end of the object.
- ✓ **Point F2:** This line indicates the value of the point force load in the local 2-axis direction and its location along the line object measured from the i-end of the object.
- ✓ **Point F3:** This line indicates the value of the point force load in the local 3-axis direction and its location along the line object measured from the i-end of the object.
- ✓ **Point FX:** This line indicates the value of the point force load in the global X-axis direction and its location along the line object measured from the i-end of the object.
- ✓ **Point FY:** This line indicates the value of the point force load in the global Y-axis direction and its location along the line object measured from the i-end of the object.
- ✓ **Point FGrav:** This line indicates the value of the point force load in the gravity (negative global Z-axis) direction and its location along the line object measured from the i-end of the object.

- ✓ **Point M1:** This line indicates the value of the point moment load about the local 1-axis and its location along the line object measured from the i-end of the object.
- ✓ **Point M2:** This line indicates the value of the point moment load about the local 2-axis and its location along the line object measured from the i-end of the object.
- ✓ **Point M3:** This line indicates the value of the point moment load about the local 3-axis and its location along the line object measured from the i-end of the object.
- ✓ **Point MX:** This line indicates the value of the point moment load about the global X-axis and its location along the line object measured from the i-end of the object.
- ✓ **Point MY:** This line indicates the value of the point moment load about the global Y-axis and its location along the line object measured from the i-end of the object.
- ✓ **Point MGrav:** This line indicates the value of the point moment load about the gravity (negative global Z-axis) direction and its location along the line object measured from the i-end of the object.
- ✓ **Uniform F1:** This line indicates the value of the uniform force load applied to the line object in the local 1-axis direction along its entire length.
- ✓ **Uniform F2:** This line indicates the value of the uniform force load applied to the line object in the local 2-axis direction along its entire length.
- ✓ **Uniform F3:** This line indicates the value of the uniform force load applied to the line object in the local 3-axis direction along its entire length.

- ✓ **Uniform FX:** This line indicates the value of the uniform force load applied to the line object in the global X-axis direction along its entire length.
- ✓ **Uniform FY:** This line indicates the value of the uniform force load applied to the line object in the global Y-axis direction along its entire length.
- ✓ **Uniform FGrav:** This line indicates the value of the uniform force load applied to the line object in the gravity (negative global Z-axis) direction along its entire length.
- ✓ **Uniform FX Proj:** This line indicates the value of the projected uniform force load applied to the line object in the global X-axis direction.
- ✓ **Uniform FY Proj:** This line indicates the value of the projected uniform force load applied to the line object in the global Y-axis direction.
- ✓ **Uniform FGrav Proj:** This line indicates the value of the projected uniform force load applied to the line object in the Gravity (negative global Z-axis) direction.
- ✓ **Uniform M1:** This line indicates the value of the uniform moment load applied to the line object about the local 1-axis along its entire length.
- ✓ **Uniform M2:** This line indicates the value of the uniform moment load applied to the line object about the local 2-axis along its entire length.
- ✓ **Uniform M3:** This line indicates the value of the uniform moment load applied to the line object about the local 3-axis along its entire length.
- ✓ **Uniform MX:** This line indicates the value of the uniform moment load applied to the line object about the global X-axis direction along its entire length.

- ✓ **Uniform MY:** This line indicates the value of the uniform moment load applied to the line object about the global Y-axis direction along its entire length.
- ✓ **Uniform MGrav:** This line indicates the value of the uniform moment load applied to the line object about the gravity (negative global Z-axis) direction along its entire length.
- ✓ **Start Trap F1:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed force load applied to the line object in the local 1-axis direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap F2:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed force load applied to the line object in the local 2-axis direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap F3:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed force load applied to the line object in the local 3-axis direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap FX:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed force load applied to the line object in the global X-axis direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap FY:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed force load applied to the line object in the global Y-axis direction. It also indicates the location

of the start point measured from the i-end of the line object.

- ✓ **Start Trap FGrav:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed force load applied to the line object in the Gravity (negative global Z-axis) direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap FX Proj:** This line indicates the value at the start point of a linear portion of a projected trapezoidal distributed force load applied to the line object in the global X-axis direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap FY Proj:** This line indicates the value at the start point of a linear portion of a projected trapezoidal distributed force load applied to the line object in the global Y-axis direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap FGrav Proj:** This line indicates the value at the start point of a linear portion of a projected trapezoidal distributed force load applied to the line object in the Gravity (negative global Z-axis) direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **End Trap F1:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed force load applied to the line object in the local 1-axis direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap F2:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed force load applied to the line object in the local 2-axis direction. It also indicates the location of the end point measured from the i-end of the line object.

- ✓ **End Trap F3:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed force load applied to the line object in the local 3-axis direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap FX:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed force load applied to the line object in the global X-axis direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap FY:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed force load applied to the line object in the global Y-axis direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap FGrav:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed force load applied to the line object in the Gravity (negative global Z-axis) direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap FX Proj:** This line indicates the value at the end point of a linear portion of a projected trapezoidal distributed force load applied to the line object in the global X-axis direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap FY Proj:** This line indicates the value at the end point of a linear portion of a projected trapezoidal distributed force load applied to the line object in the global Y-axis direction. It also indicates the location of the end point measured from the i-end of the line object.

- ✓ **End Trap FGrav Proj:** This line indicates the value at the end point of a linear portion of a projected trapezoidal distributed force load applied to the line object in the Gravity (negative global Z-axis) direction. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **Start Trap M1:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed moment load applied to the line object about the local 1-axis. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap M2:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed moment load applied to the line object about the local 2-axis. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap M3:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed moment load applied to the line object about the local 3-axisn. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap MX:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed moment load applied to the line object about the global X-axis. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **Start Trap MY:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed moment load applied to the line object about the global Y-axis. It also indicates the location of the start point measured from the i-end of the line object.

- ✓ **Start Trap MGrav:** This line indicates the value at the start point of a linear portion of a trapezoidal distributed moment load applied to the line object about the Gravity (negative global Z-axis) direction. It also indicates the location of the start point measured from the i-end of the line object.
- ✓ **End Trap M1:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed moment load applied to the line object about the local 1-axis. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap M2:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed moment load applied to the line object about the local 2-axis. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap M3:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed moment load applied to the line object about the local 3-axisn. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap MX:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed moment load applied to the line object about the global X-axis. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap MY:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed moment load applied to the line object about the global Y-axis. It also indicates the location of the end point measured from the i-end of the line object.
- ✓ **End Trap MGrav:** This line indicates the value at the end point of a linear portion of a trapezoidal distributed moment load applied to the line object about the Gravity (negative global Z-axis) direction. It also indicates the location of the end point measured from the i-end of the line object.

- ✓ **Temperature:** This line indicates the value of a temperature change that is assigned to the line object.
- ✓ **Add Point Temp:** This line indicates whether or not you have indicated that ETABS is to consider previously specified point object temperature changes at the ends of the line object. This line either displays "Yes" or "No."

Static load case names are defined using the **Define menu > Static Load Cases** command. See the section titled "Static Load Cases" in Chapter 11 for more information on static load cases.

Point loads are assigned to line objects using the **Assign menu > Frame/Line Loads > Point** command. See the subsection titled "Point Load Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

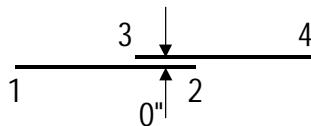
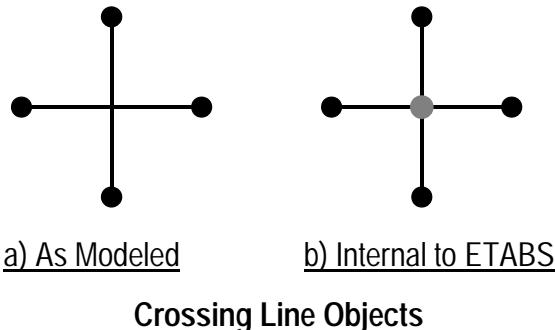
Uniform and trapezoidal loads are assigned to line objects using the **Assign menu > Frame/Line Loads > Distributed** command. See the subsection titled "Distributed Load Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

Temperature loads are assigned to line objects using the **Assign menu > Frame/Line Loads > Temperature** command. See the subsection titled "Temperature Load Assignments to Line Objects" under the section titled "Assignments to Line Objects" in Chapter 14 for more information.

Overlapping Line Objects

Line objects can typically cross over each other but in general it is preferred, and in some cases required, that line objects do not lie one on top of the other. Figure 24-1 shows some examples.

Figure 24-1:
Crossing (intersecting) and overlapping line objects



Overlapping Line Objects



Note:

You can not overlap two line objects that are both assigned a frame section property. They can, however, cross (intersect).

Figure 24-1a shows two crossing line objects as they might appear in your ETABS model. If these line objects are assigned frame section properties then ETABS assumes that the frame sections are connected where they intersect and internally provides a joint there as shown in Figure 24-1b. If the line object is assigned link properties then ETABS assumes that there is no connection. ETABS never breaks a link element up into multiple pieces.

If the line objects shown in Figure 24-1 were assigned both frame section properties and link properties at the same time then ETABS assumes the frame sections to be broken and connected where they cross but the link elements remain unbroken with no connection where they cross.

The bottom portion of Figure 24-1 shows two overlapping line objects. The ends of one of the line objects are labeled 1 and 2 and the ends of the other are labeled 3 and 4. The line objects overlap between points 2 and 3. Following are rules that ETABS applies to these overlapping line objects:

- You can not assign two overlapping objects frame section properties. If both line objects are assigned frame

section properties then when you try to run the analysis ETABS will flag it as an illegal assignment and the analysis stops. The reason it is illegal is because ETABS can not determine what section property to use in the area between points 2 and 3.

- Overlapping line loads and line masses are additive. Suppose both line objects are assigned a uniform line load of 1 kip per foot. Then between points 1 and 3 and between point 2 and 4 the load intensity is 1 kip per foot and between point 2 and 3 the load intensity is 2 kips per foot.
- Link elements can overlap. In this example the link element from point 1 to 2 would not be connected in any way to point 3. Similarly, the link element from point 3 to 4 would not be connected in any way to point 2.
- Suppose the line object from point 1 to 2 is a frame section and the line object from 3 to 4 is assigned link properties. In this case the frame element would be internally broken up in ETABS connecting from point 1 to point 3 and then point 3 to point 2. The link element would not be broken up and would simply span from point 3 to point 4 with no connection to point 2.

Plan Views of Vertical Line Objects



Tip:

You can draw and/or select a column in plan view.

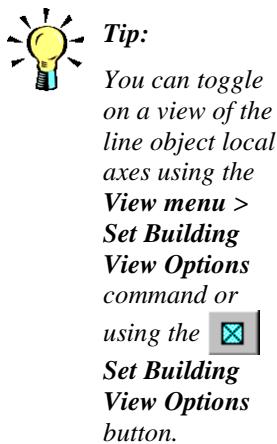
In a plan view of a story level, line objects that are assigned frame section properties (columns) appear as points that represent a plan section (to scale) through the column. While in a plan view you can use the **Draw menu > Draw Line Objects > Create Columns in Region or at Click (plan)** command, or the associated toolbar button on the side toolbar to draw columns. This is a very convenient method for drawing columns. See the section titled "Drawing Line Objects" in Chapter 12 for more information.

Default Line Object Local Axes

The following subsections describe the default local axes orientation for vertical, horizontal and other (neither vertical nor horizontal) line objects. Note that you can use the **Assign menu > Frame/Line > Local Axes** command to rotate the local 2 and 3 axes about the local 1 axes from their default orientation.

Vertical Line Objects

For vertical line objects the local axes have the following default orientation:



- **Local axis 1:** This axis is along the line object. The positive local 1 axis is in the same direction as the positive global Z-axis, upward.
- **Local axis 2:** This axis is perpendicular to the line object. The projection of the positive local 2 axis onto the global X-axis is in the same direction as the positive global X-axis.
- **Local axis 3:** This axis is perpendicular to the line object. The direction of the positive local 3 axis is determined from applying the right-hand rule using the directions of the 1 and 2 axes described above. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

Horizontal Line Objects

For horizontal line objects (parallel to the global X-Y plane) the local axes have the following default orientation:

- **Local axis 1:** This axis is along the line object. The projection of the positive local 1 axis onto the global X-axis is in the same direction as the positive global X-axis. If the line object is parallel to the global Y-axis such that there is no projection onto the global X-axis then the positive direction of the local 1 axis is the same as the direction of the positive global Y-axis.

**Note:**

The local axes of line objects and associated frame or link elements are one and the same.

24

- **Local axis 2:** The local 2 axis is perpendicular to the line object. The positive local 2 axis points in the same direction as the global Z-axis, upward.
- **Local axis 3:** The local 3 axis is perpendicular to the line object and is horizontal. The direction of the positive local 3 axis is determined from applying the right-hand rule using the directions of the 1 and 2 axes described above. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

Other Line Objects

For line objects that are not vertical and are not horizontal the local axes have the following default orientation:

**Tip:**

When local axes are displayed on the screen local axis 1 is red, local axis 2 is white and local axis 3 is blue, always. This is the same order as the colors of the American flag: red, white and blue.

- **Local axis 1:** This axis is along the line object. The positive local 1 axis has an upward sense. The projection of the positive local 1 axis onto the global Z-axis is in the same direction as the positive global Z-axis, upward.
- **Local axis 2:** The local 2 axis is perpendicular to the line object. The local 1-2 plane is vertical. The positive local 2 axis has an upward sense. The projection of the positive local 2 axis onto the global Z-axis is in the same direction as the positive global Z-axis, upward.
- **Local axis 3:** The local 3 axis is perpendicular to the line object and is horizontal. The direction of the positive local 3 axis is determined from applying the right-hand rule using the directions of the 1 and 2 axes described above. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

Polyline

A polyline object is a multi-segmented line object. You can not make any assignments to a polyline object. Polyline objects are only used to define developed elevations. Developed elevations are special user-defined elevation views of the model. Typically they are unfolded views of elevations that would otherwise go around one or more corners of the model. See Chapter 12 for information on defining developed elevations.



Point Objects

General

Point objects are, as the name indicates, points. Point objects are automatically created by ETABS at the corners of all area objects and at the ends of all line objects. In addition, you can use the **Draw menu > Draw Point Objects** command, or the associated toolbar button on the side toolbar to draw additional point objects at any location in your model.

You do not make assignments to point objects as you draw them. Instead you select the point objects after they are created and make assignments to them using the **Assign menu > Joint/Point** command or the **Assign menu > Joint/Point Load** command.

You can right click on a point object to bring up the Point Information dialog box where you can view information about the point including its exact location and assignments. See the section titled “Right Click Information for Point Objects” later in this chapter for more information.

Automatic Point Object Labeling

When you first draw a point object, or one is automatically created at a corner of an area object or at an end of a line object ETABS automatically assigns a label to it. If the point object falls at a story level at the end of a column-type line object then the point object is given the same label as the column, including the C prefix. For example, the point object at the top of column C23 at the 3rd floor level is also labeled C23.

If the point object does *not* fall at a story level at the end of a column-type line object then this label is simply a number with no prefix. For example, a point object located at the corner of a floor diaphragm and not at the top of a column might be labeled 14 (with no prefix).

The advantage to this point labeling scheme is that when you look at your model in plan view you can clearly see where the columns fall just by looking at the point labels.

In general you can not assign your own labels to point objects. Only the automatic ETABS labels are allowed. There is one exception to this. If you import a text file into ETABS then that text file can have point object labels different from the automatic labels described above. Be aware though that in this case, if you move the point object, ETABS will automatically relabel the object.

ETABS uses an intelligent labeling scheme. Typically it gives the same label to point objects located at the same plan location and at the same elevation relative to their respective story level, but at different story levels. Note that point objects are always identified by a story level and a label. This type of labeling scheme makes it much easier for you to locate objects. For example point object 2 at one level will be directly above point object 2 at another story level.

You can use the **Edit menu > Auto Relabel All** command to automatically relabel all area, line and point objects. See the subsection titled "Relabeling Objects" in Chapter 23 for additional information on this.

Assignments Made through the Assign Menu

You can make many assignments to point objects using the Assign menu. The types of point object assignments you can make include:

- Assign a rigid diaphragm constraint.
- Assign a panel zone element.
- Assign point restraints (supports).
- Assign grounded point springs.
- Assign grounded link elements.
- Assign additional point mass.
- Assign point object forces.
- Assign displacements to restrained degrees of freedom of point objects.
- Assign temperature changes to point objects.



Tip:

To make an assignment to a point object first select the object and then click the appropriate command on the Assign menu.

Use either the **Assign menu > Joint/Point** or the **Assign menu > Joint/Point Loads** command to make these assignments. See the section titled "Assignments to Point Objects" in Chapter 14 for more information.

Right Click Information for Point Objects

Right click on a point object to view useful information that identifies the object, describes its location and reveals its assignments. Right clicking on a point object means to position your mouse pointer over the object and click the right (not left as you usually do) button on the mouse. When you do this the Point Information dialog box appears.

**Tip:**

Right click on a point object to view useful information that identifies the object, describes its location and reveals its assignments.

25

**Tip:**

Click on a tab in the Point Information dialog box to see the data associated with the tab.

The units drop-down box in the Point Information dialog box allows you to change the current units without leaving the dialog box. Any change in units that you make while in this dialog box is permanent; that is, the change remains when you exit the dialog box.

The area at the top of the Point Information dialog box that is named Identification provides basic information that identifies the point object. This information includes:

- **Label:** This is the point object label assigned by ETABS. See the previous subsection in this chapter titled "Automatic Point Object Labeling" for more information.
- **Story:** This is the story level associated with the point object.

The Point Information dialog box also has three different tabs located just above the Identification area. Each of these tabs displays different types of information. The tabs are labeled Location, Assignments and Loads. These labels are relatively self-explanatory. The Location tab includes information that locates the point object. The Assignments tab includes information on all point object assignments except for loads. The Loads tab includes information on all load assignments to the point object. The items displayed in each of these tabs are detailed in the subsections below.

Location Tab in the Point Information Dialog Box

The following geometric information about the point object is displayed on the Location tab:

- **X:** This is the global X coordinate of the point object.
- **Y:** This is the global Y coordinate of the point object.

- **Delta Z:** This is the vertical distance from the story level that the point object is associated with to the point object. The value of Delta Z is always between zero and the value of the story height associated with the point object, inclusive. Delta Z is always positive even though it is measured from the story level downward.
- **Connectivity:** If no line or area object connects to the point object then this item says "None." Otherwise this line is blank but it is followed by one or more of the following lines.
 - ✓ **Line:** This item shows the label of a line object that is connected to the selected point object.
 - ✓ **Area:** This item shows the label of an area object that is connected to the selected point object.

Note that as many line and area items are displayed as there are line and area objects connected to the selected point object.

Assignments Tab in the Point Information Dialog Box

Following are the items that appear (or in some cases may appear) on the Assignments tab:

- **Rigid diaphragm:** If a rigid diaphragm is *not* assigned to the point object then this item displays "None." Otherwise it displays the name of the rigid diaphragm assigned to the object. The rigid diaphragm is assigned using the **Assign menu > Joint/Point > Rigid Diaphragm** command. See the subsection titled "Rigid Diaphragm Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.

- **Panel Zones:** If a panel zone element is *not* assigned to the point object then this item displays "None." Otherwise it is blank and some of the items listed below are also displayed. A panel zone element is assigned using the **Assign menu > Joint/Point > Panel Zone** command. See the subsection titled "Panel Zone Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.
 - ✓ **Connectivity:** This item shows the connectivity specified for the panel zone element. It is either Beam-Column, Beam-Brace, or Brace-Column.
 - ✓ **Local axis 2:** This item indicates the orientation of the local 2-axis of the panel zone. It is either From Column or n° from X, where n is a number and X means the global X-axis.
 - ✓ **Spring property:** If the properties of the panel zone are determined from the column or the column plus a specified doubler plate then this item is displayed and it says "From Column." See also the doubler plate item below.
 - ✓ **Doubler Plate:** If the properties of the panel zone are determined from the column or the column plus a specified doubler plate then this item is displayed. If the properties come only from the column then this item displays "None." Otherwise it displays the specified thickness of the doubler plate. See also the spring property item above.
 - ✓ **Major moment spring:** If the properties of the panel zone are determined from specified spring properties then this item is displayed and it shows the specified spring stiffness for major moment. See also the minor moment spring item below.

**Note:**

The *ux*, *uy* and *uz* restraints are translational restraints in the global *X*, *Y* and *Z* directions respectively. The *rx*, *ry* and *rz* restraints are rotational restraints about the global *X*, *Y* and *Z* axes respectively.

- ✓ **Minor moment spring:** If the properties of the panel zone are determined from specified spring properties then this item is displayed and it shows the specified spring stiffness for minor moment. See also the major moment spring item above.
- ✓ **Link property:** If the properties of the panel zone are determined from a specified link property then this item is displayed and it shows the specified property.
- **Restraint:** If a restraint (support) is *not* assigned to the point object then this item displays "None." Otherwise it displays on one line all of the restraints that are assigned to the point object. The six possible restraints are *ux*, *uy*, *uz*, *rx*, *ry* and *rz*. A point object restraint is assigned using the **Assign menu > Joint/Point > Restraints (Supports)** command. See the subsection titled "Restraint (Support) Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.
- **Springs:** If a point spring is *not* assigned to the point object then this item displays "None." Otherwise this line is blank and is followed by one or more of the lines listed below. A point spring is assigned using the **Assign menu > Joint/Point > Point Springs** command. See the subsection titled "Point Spring Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.
 - ✓ **UX:** If an uncoupled point spring stiffness is specified in the global *X* direction then this line is displayed and it shows the specified spring stiffness.
 - ✓ **UY:** If an uncoupled point spring stiffness is specified in the global *Y* direction then this line is displayed and it shows the specified spring stiffness.

- ✓ **UZ:** If an uncoupled point spring stiffness is specified in the global Z direction then this line is displayed and it shows the specified spring stiffness.
- ✓ **RX:** If an uncoupled point spring stiffness is specified for rotation about the global X-axis then this line is displayed and it shows the specified spring stiffness.
- ✓ **RY:** If an uncoupled point spring stiffness is specified for rotation about the global Y-axis then this line is displayed and it shows the specified spring stiffness.
- ✓ **RZ:** If an uncoupled point spring stiffness is specified for rotation about the global Z-axis then this line is displayed and it shows the specified spring stiffness.
- ✓ **UX-UX, etc.:** If you have specified a coupled 6x6 spring then the following twenty-one items from the upper portion of the spring stiffness matrix are displayed each on its own line: UX-UX, UY-UX, UY-UY, UZ-UX, UZ-UY, UZ-UZ, RX-UX, RX-UY, RX-UZ, RX-RX, RY-UX, RY-UY, RY-UZ, RY-RX, RY-RY, RZ-UX, RZ-UY, RZ-UZ, RZ-RX, RZ-RY and RZ-RZ.
- **Link Property:** If a grounded link element property is *not* assigned to the point object then this item displays "None." Otherwise it displays the name of the assigned link property. Note that you can assign more than one link property to the same point object. The assignments are additive; they do not replace one another. When more than one grounded link element is assigned to a point object multiple Link properties lines are used, one for each property. A link element property is assigned using the **Assign menu > Joint/Point > Link Properties** command. See the subsection titled "Link Property Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.

- **Mass:** If an additional point mass is *not* assigned to the object then this item displays "None." Otherwise this line is blank and is followed by one or more of the lines listed below.
 - ✓ **UX:** This is the additional translational point mass specified in the global X direction.
 - ✓ **UY:** This is the additional translational point mass specified in the global Y direction.
 - ✓ **UZ:** This is the additional translational point mass specified in the global Z direction.
 - ✓ **RX:** This is the additional rotational mass moment of inertia specified about the global X-axis.
 - ✓ **RY:** This is the additional rotational mass moment of inertia specified about the global Y-axis.
 - ✓ **RZ:** This is the additional rotational mass moment of inertia specified about the global Z-axis.

Note that the point mass value is displayed here regardless of the mass source you have specified using the **Define menu > Mass Source** command. If you designate that the mass source is from a specified load combination then the value shown here is not used by the program. See the section titled "Mass Source" in Chapter 11 and the section titled "Mass" in Chapter 27 for more information.

The point mass is assigned using the **Assign menu > Joint/Point > Additional Point Mass** command. See the subsection titled "Additional Point Mass Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.

- **Group:** This item displays the groups that the object belongs to. By default in ETABS all objects belong to the default group called ALL. You can not delete the group named ALL and you can not remove an object from the group. Thus even if you have not assigned the point object to any groups you will see ALL for this item indicating that it belongs to the ALL group. If you have assigned the object to one or more groups then additional Group lines are provided, one for each group to which you have assigned the object.

Loads Tab in the Point Information Dialog Box

Following are the items that may appear on the Loads tab:

- **Load:** If no loads are assigned to the point object then this displays "None." Otherwise this item is not displayed.
- **Static Load Case:** This line indicates the name of the static load case to which the load(s) listed on the immediately following line(s) of the Loads tab is (are) applied. For point objects either force, displacement or temperature loads may be listed. The bulleted items below list the point object loads you may see immediately following the Static Load Case line. One or more of these load lines may be listed for each static load case depending on the assignments you have made.
 - ✓ **Force X:** This line indicates the value of the point force load applied to the point object in the global X-axis direction.
 - ✓ **Force Y:** This line indicates the value of the point force load applied to the point object in the global Y-axis direction.
 - ✓ **Force Z:** This line indicates the value of the point force load applied to the point object in the global Z-axis direction.

- ✓ **Moment X:** This line indicates the value of the point moment load applied to the point object about the global X-axis.
- ✓ **Moment Y:** This line indicates the value of the point moment load applied to the point object about the global Y-axis.
- ✓ **Moment Z:** This line indicates the value of the point moment load applied to the point object about the global Z-axis.
- ✓ **Ground Displacement X:** This line indicates the value of the ground displacement applied to the point object in the global X-axis direction.
- ✓ **Ground Displacement Y:** This line indicates the value of the ground displacement applied to the point object in the global Y-axis direction.
- ✓ **Ground Displacement Z:** This line indicates the value of the ground displacement applied to the point object in the global Z-axis direction.
- ✓ **Ground Rotation X:** This line indicates the value of the ground rotation applied to the point object about the global X-axis.
- ✓ **Ground Rotation Y:** This line indicates the value of the ground rotation applied to the point object about the global Y-axis.
- ✓ **Ground Rotation Z:** This line indicates the value of the ground rotation applied to the point object about the global Z-axis.
- ✓ **Temperature:** This line indicates the value of the temperature change assigned to the point object.

Static load case names are defined using the **Define menu > Static Load Cases** command. See the section titled "Static Load Cases" in Chapter 11 for more information on static load cases.

Point loads are assigned to point objects using the **Assign menu > Joint/Point Loads> Force** command. See the subsection titled "Force Loads to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.

Ground displacements are assigned to restrained point objects using the **Assign menu > Joint/Point Loads> Ground Displacement** command. See the subsection titled "Ground Displacement Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.

Temperature loads are assigned to point objects using the **Assign menu > Joint/Point Loads > Temperature** command. See the subsection titled "Temperature Load Assignments to Point Objects" under the section titled "Assignments to Point Objects" in Chapter 14 for more information.

Point Objects Overlapping Other Objects

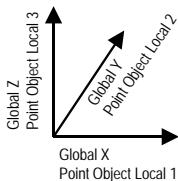
Note:

Two point objects can not be located one on top of the other.

You can not have two point objects on top of each other.

Quite often a point object is located on top of another object. For example it may be on top of a vertical line object that represents a column. If you want to select only the point object you can hold down the Ctrl key on your keyboard as you left or right click on the objects. A dialog box appears where you can choose the object you want to select. Highlight the point object in this dialog box and click the OK button. If you left clicked the object is selected and if you right clicked the object assignments are displayed.

Point Object Local Axes



The local axes for point objects are the same as the global axes. Local axis 1 corresponds to the global X-axis. Local axis 2 corresponds to the global Y-axis. Local axis 3 corresponds to the global Z-axis. You can not change the local axes of a point object.



Groups and Section Cuts

General



Tip:

Groups are a very valuable concept. They are useful for both editing a model and reviewing output. You should not overlook the power of groups in ETABS.

The concept of groups is the backbone of some powerful tools in ETABS. In ETABS a group is a collection of objects that is assigned a name. There are three useful purposes for groups:

- You can select objects by group.
- You can design steel frame elements, including composite beams, by group. In this case the optimum section that works for all frame elements in the group is selected by ETABS.
- You can use groups to help define section cuts in your structure. You can then obtain the forces acting on those section cuts.

Each of these items is described in more detail later in this chapter.

Defining Groups

Defining a group consists of indicating what objects are included in the group and then specifying a name for the group. Use the following steps to define a group.

1. Select the objects that make up the group.
2. Click the **Assign menu > Group Names** command to bring up the Assign Group dialog box.
3. If the name of the group already appears in the list box then simply highlight that name and click the **OK** button. (Note that if you use an existing group name for your new group the selected items replace rather than add to any objects that might have previously been defined for that group.) If the name of the group does not appear in the list box then continue on to Step 4.
4. Type a name for the group into the edit box and click the **Add New Group** button to define the group name. Then click the **OK** button to define the group.

26



Tip:

You can specify that the model display should be by the colors assigned to the groups.

Groups are also assigned colors. You can use the **View menu > Set Display Options** command, or the **Set Display Options** button, , on the main toolbar to indicate that the display should be shown by the colors of the groups. This type of view can be useful for determining what items are included in a particular 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 that it is assigned to. This can sometimes make it difficult to tell which objects are assigned to a particular group. A way to clearly tell which objects are assigned to a group is to view the entire model, click the **Select menu > Select by Groups** command to select the group and then click the **View menu > Show Selection Only** command to see only the objects that are part of the group.

Selecting Groups

Once a group is defined you can select it using the **Select menu > Select by Groups** command. As an example, suppose you have a structure that consists of just beam and column elements. You could select the column elements and define a group called COL using the procedure described in the previous section titled "Defining Groups."



Note:

You can select and deselect by group.

Once the COL group is defined you can select all of the columns at any time by clicking **Select menu > Select by Groups** command, highlighting the group name called COL and clicking the **OK** button. Note that when highlighting the groups to be selected you can hold down the Shift and/or Ctrl keys on the keyboard as you highlight group names to select multiple groups at the same time. See the section titled "Using the mouse" in Chapter 4 for more information.

Suppose that you want to select all of the beams but the only group you have defined is the COL group. First select the entire structure by either clicking **Select menu > Select All** or by clicking the **Select All** button, , located on the side toolbar. Then click **Select menu > Deselect > by Groups**, highlight the group called COL and click the **OK** button. The COL group is deselected and you are left with just the beams selected.

Note that as an alternative method of selecting the beams you could just put them in their own group, you might call it BEAM, and then simply select them by group as described above for the column group, COL.

Designing by Groups

In steel frame design and composite beam design using ETABS you have the option of designing elements by group. When you specify a group for design all elements in the group are given the same section size, if possible.

Steel members must be assigned auto select section lists to be designed as a part of a group. Typically you want each member in the group to be assigned the same auto select section list, although this is not absolutely necessary.

Section Cuts

26

In ETABS you can define section cuts in your building and then get the resultant of the forces acting on the section cut for any load case or load combination. You can define a section cut using the **Define menu > Section Cut** command. To define a section cut you give it a name, indicate a group that defines the section cut, indicate the location that the forces are to be summed about and indicate a local axes orientation. These items are discussed below. Once you have defined one or more section cuts you can display the section cut forces in a tabular form on screen using the **Display menu > Show Section Cut Forces** command.

Defining a Section Cut with a Group

You indicate the location of the section cut by specifying a group that typically consists of one or more area (shell) and/or line (frame and/or link) objects plus the points objects along one side (or end for frame and link members) of the objects. A line connecting the selected points objects describes the section cut and the selected area and line objects indicate which side of the section cut the forces are taken on.



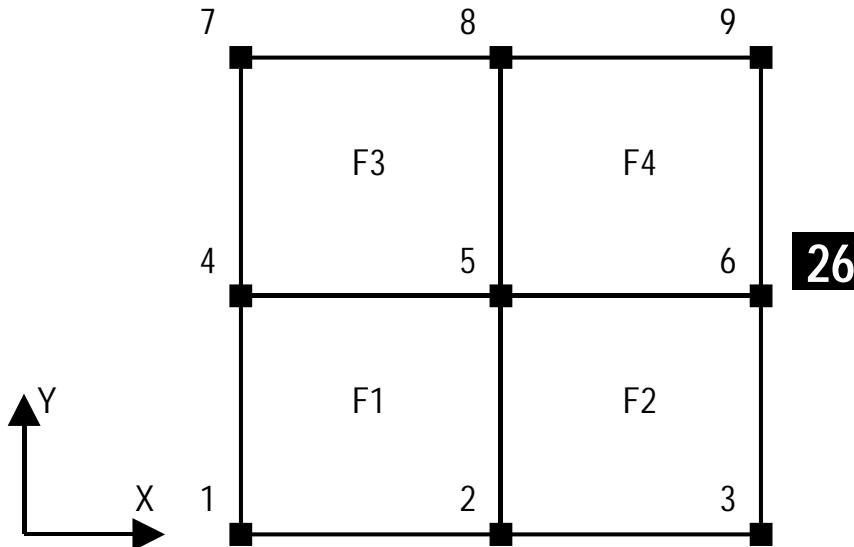
Note:

Section cut forces are reported in the local coordinate system of the section cut.

Figure 26-1 shows a simple example of a floor slab that consists of four area objects labeled F1 through F4 and nine point objects labeled 1 through 9. Suppose that you want to get the forces acting along a section cut parallel to the Y-axis through the center of the slab.

To do this you define a group consisting of point objects 2, 5 and 8 and area objects F1 and F3. Point objects 2, 5 and 8 define the section cut and area objects F1 and F3 define which side of the section cut to consider the forces. Note that if you wanted to consider forces on the other side of the section cut then you should include area objects F2 and F4 in your group instead of F1 and F3.

Figure 26-1:
Example of defining
a section cut with a
group



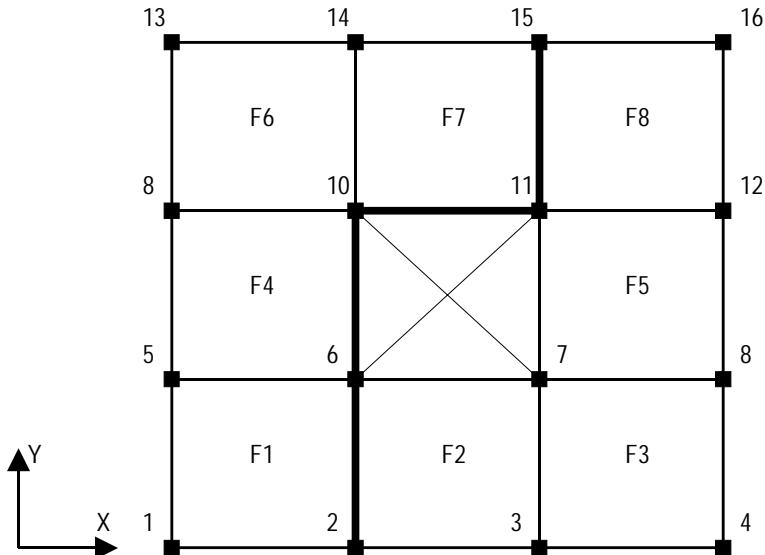
You should not include area objects (or line objects) on both sides of a section cut in the same group. If you do this the forces contributed from the objects on the two sides of the section cut will cancel each other out and you will essentially be left with zero force at the section cut.

A section cut does not necessarily have to be a single straight line. It can be made up of a series of straight line segments in any orientation as long as the break points occur at a point object. For example, refer to Figure 26-2 which shows a floor slab that consists of eight area objects labeled F1 through F8 and sixteen point objects labeled 1 through 16. There is an opening in the center of the floor slab.

In this example the section cut is shown by the heavy line. If you want forces on the negative global X-axis side of this section cut then the group defining the section cut should include point objects 2, 6, 10, 11 and 15 and area objects F1, F4, F6 and F7.

Figure 26-2:
Example of defining
a section cut with a
group

26



Location that Section Cut Forces are Summed About

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. The location that the section cut forces are summed about is reported along with the section cut forces.

In some cases the default location that the section cut forces are summed about may not be convenient for you. If you want to have the section cut forces specified at a location different from the default you can do so by specifying the new location in global coordinates (X, Y and Z) in the section cut definition.

Local Axes for Section Cuts

The local axes of a section cut are determined as follows:

- **Local 1-axis:** By default the positive local 1-axis is in the same direction as the positive global X-axis. You can rotate the local 1-axis in a plane parallel to the global X-Y plane by specifying an angle of rotation in degrees. The angle is measured from the positive global X-axis to the positive local 1-axis. When looking down on the

Note:

You can rotate the local 1 and 2 (horizontal) axes of a section cut about the local 3 (vertical) axis.

structure in the negative global Z direction a positive angle is counterclockwise.

- **Local 2-axis:** The direction of the positive local 2-axis is determined by applying the right-hand rule to the directions of the local 1 and 3 axes as described here.
- **Local 3-axis:** This axis is always vertical. The positive local 3-axis is in the same direction as the positive global Z-axis.

Note that by default the positive local 1, 2 and 3 axes of the section cut correspond to the global X, Y and Z axis respectively.

How ETABS Calculates Section Cut Forces

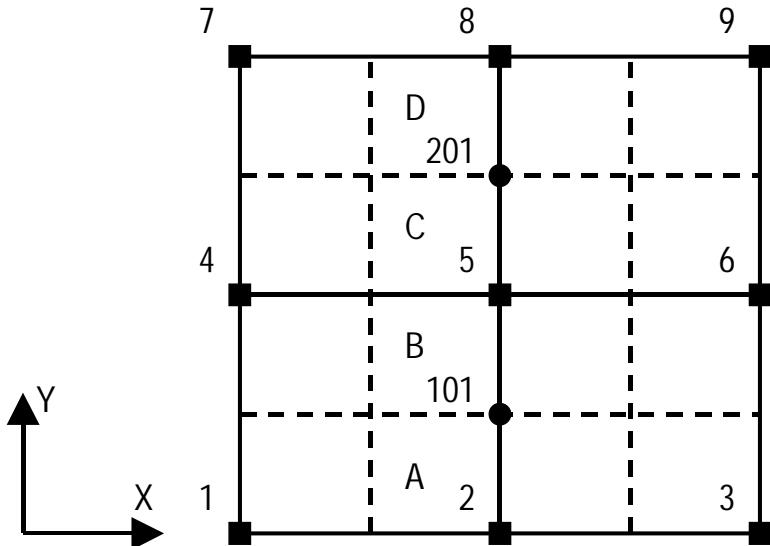
This section describes how ETABS calculates section cut forces. Essentially ETABS sums joint forces from the frame, shell and link members included in the group that defines the section cut. The joints that are considered are those at the same location as the point objects that are included in the group.

To sum the forces, each considered joint force and moment is transformed to the location that the forces are summed about and then they are transformed to the local coordinate system specified for the section cut forces. When all of the considered forces have been transformed to the appropriate location and coordinate system they are simply summed up and the total forces are reported.

The above explanation is complete if there is no ETABS internal meshing in the area objects that are included in the group defining the section cut. As an example, consider Figure 26-1, and assume that the group defining the section cut includes area objects F1 and F3 and point objects 2, 5 and 8. Further assume that all of the point objects are equally spaced in each direction. Finally assume that ETABS does not internally mesh area objects F1 and F3.

To determine the default location that the section cut forces are summed about ETABS averages the coordinates of point objects 2, 5 and 8. In this simple example those average coordinates fall at the same location as point object 5.

Figure 26-3:
First example of how ETABS calculates section cut forces when objects are internally meshed by the program



Next ETABS takes the joint forces from area object F1 at point 2 and transforms the forces to point 5 and also transforms them to the local coordinate system of the section cut. Then ETABS takes the joint forces from area object F1 at point 5 and performs the same double transformation. Similarly, the joint forces from area object F3 at points 5 and 8 are doubly transformed. Finally ETABS adds up the four sets of transformed forces and reports the totals.

Now lets suppose that area objects F1 and F3 are internally meshed by ETABS. In this case, the model still appears to you as shown in Figure 26-1. The group defining the section cut is still the same as previously described, that is, area objects F1 and F3 and point objects 2, 5 and 8. Internally to ETABS the model appears as shown in Figure 26-3 which shows the internal meshing of the area objects with dashed lines.

In Figure 26-3 for identification purposes some of the area objects created by the internal meshing are identified as A, B, C and D. The points labeled 101 and 201 are internal points created by the internal meshing of the area objects.

In this case ETABS determines the section cut forces as follows:

- To determine the default location that the section cut forces are summed about ETABS averages the coordinates of point objects 2, 5 and 8. Again, in this simple example those average coordinates fall at the same location as point object 5. Note that the internal joint 101 and 201 are not considered in determining this location.
- ETABS takes the joint forces from area object A at point 2 and transforms the forces to point 5 and also transforms them to the local coordinate system of the section cut.
- ETABS takes the joint forces from area object A at point 101 and transforms the forces to point 5 and also transforms them to the local coordinate system of the section cut.
- Similarly, the joint forces from area object B at points 101 and 5 are doubly transformed.
- Similarly, the joint forces from area object C at points 5 and 201 are doubly transformed.
- Similarly, the joint forces from area object D at points 201 and 8 are doubly transformed.
- Finally ETABS adds up the eight sets of transformed forces and reports the totals.

ETABS includes the forces from an internal joint in the section cut forces if the line defining the section cut passes through that joint. Recall that the line defining the section cut is simply a line that connects the point objects in the group defining the section cut.

Note:

The term "doubly transformed" refers to transforming the forces to the point that the forces are summed about and then transforming them to the local axes of the section cut. Thus two separate transformations are taking place.

As a second example consider the section cut illustrated in Figure 26-2. The group defining the section cut includes area objects F1, F4, F6 and F7 and point objects 2, 6, 10, 11 and 15. The coordinates of the default location that the forces are summed about, call it point X (not shown in the figure), are equal to the average of the coordinates of points 2, 6, 10, 11 and 15. Point X falls within the opening in the center of the slab. Assuming that

the floor is not further internally meshed by ETABS the section cut force is determined as follows:

- ETABS takes the joint forces from area object F1 at points 2 and 6 and transforms the forces to point X and also transforms them to the local coordinate system of the section cut.
- ETABS takes the joint forces from area object F4 at points 6 and 10 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object F6 at point 10 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object F7 at points 10, 11 and 15 and doubly transforms the forces to point X.
- Finally ETABS adds up the eight sets of transformed forces and reports the totals.

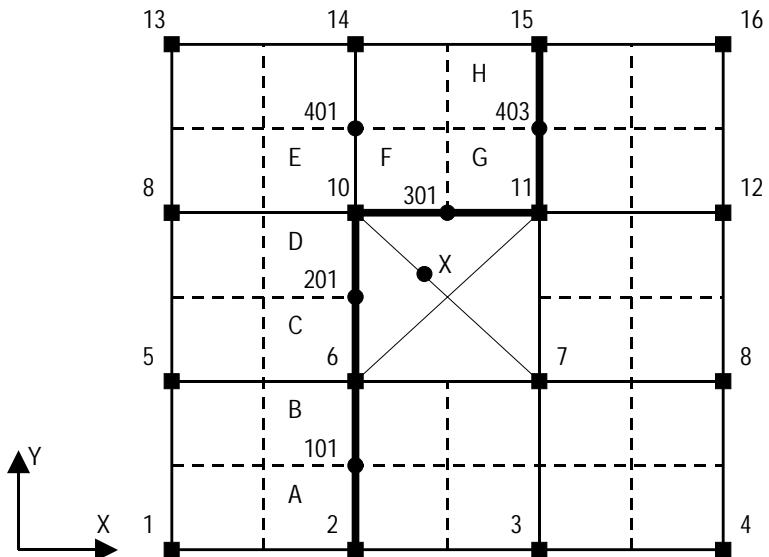
Now suppose that the floor system shown in Figure 26-2 is internally meshed By ETABS. The group defining the section cut is still the same as previously described, that is, area objects F1, F4, F6 and F7 and point objects 2, 6, 10, 11 and 15. Internally to ETABS the model appears as shown in Figure 26-4 which shows the internal meshing of the area objects with dashed lines.

In Figure 26-4 for identification purposes some of the area objects created by the internal meshing are identified as A, B, C, D, E, F, G and H. The points labeled 101, 201, 301, 401 and 403 are internal points created by the internal meshing of the area objects.

In this case ETABS determines the section cut forces as follows:

- The coordinates of the default location that the forces are summed about (point X in the figure), are equal to the average of the coordinates of points 2, 6, 10, 11 and 15.

Figure 26-4:
Second example of
how ETABS calcu-
lates section cut
forces when objects
are internally
meshed by the pro-
gram



- ETABS takes the joint forces from area object A at points 2 and 101 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object B at points 101 and 6 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object C at points 6 and 201 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object D at points 201 and 10 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object E at point 10 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object F at points 10 and 301 and doubly transforms the forces to point X.

- ETABS takes the joint forces from area object G at points 301, 11 and 403 and doubly transforms the forces to point X.
- ETABS takes the joint forces from area object H at points 403 and 15 and doubly transforms the forces to point X.
- Finally ETABS adds up the sixteen sets of transformed forces and reports the totals.

Note that the joint forces at the point labeled 401 in Figure 26-4 are not included in the section cut forces because point 401 does not fall on the heavy line that defines the section cut.

Load Cases, Load Combinations and Mass

This chapter discusses ETABS load cases, load combinations and mass.

Load Cases

There are four types of load cases available in ETABS. They are listed here and discussed below.

- Static
- Response spectrum
- Time history
- Static nonlinear

You assign a unique label to each load case as part of its definition. These labels can be used to create load combinations and to control displayed and printed output.

Static Load Case

There are many different types of loads that can be applied in a static load case. They include:

Note:

*Do not confuse
the load types
listed here with
the design types
that are speci-
fied for each
load case.*

- Concentrated forces and moments acting on point objects.
- Concentrated or distributed forces and moments acting on frame elements.
- Distributed forces acting on shell elements.
- Self-weight acting on all element types.
- Thermal loads acting on frame and shell elements.
- Displacements of the restrained (support or spring) degrees of freedom of point objects.

Each static load case may consist of an arbitrary combination of the above load types. Usually it is convenient to restrict each load case to a single type of loading and use load combinations to create more complicated combinations. Any number of static load cases may be defined in an ETABS analysis.

Some typical static load cases used for building structures might include:

- Dead load
- Superimposed dead load
- Live load
- Reduced live load
- Snow Load
- Wind Load
- Earthquake load

Use the **Define menu > Static Load Cases** command to specify static load cases in ETABS. To specify a static load case you give the load case a name, a design type and a self-weight multiplier.

Design Type

Note:

Design types are specified for each load case. They are used by the ETABS design postprocessors to create appropriate design load combinations.

In ETABS a design type is specified for each load case. The design type is used by the design post processors to create factored load combinations for use in design. It is also used to determine whether or not a live load is reducible. The design types available in ETABS are:

- Dead
- Super dead (superimposed dead load)
- Live
- Reduce live (reducible live load). See the subsection below titled "Live Load Reduction" for additional information.
- Quake (see Chapter 28 for additional information).
- Wind (see Chapter 29 for additional information).
- Snow
- Other

When you specify a static load case which has a design type of Quake you can define an automatic code-specific static lateral earthquake load that is associated with the load case. Similarly, when you specify a static load case which has a design type of Wind you can define an automatic code-specific static lateral wind load that is associated with the load case. Automatic earthquake loads are discussed in Chapter 28. Automatic wind loads are discussed in Chapter 29.

Self-Weight Multiplier

The self-weight of the structure is determined from the weight per unit volume that is specified in the material properties for the structural elements of the model. You can review and/or change material properties using the **Define menu > Material Properties** command. The actual self-weight of an element is determined by multiplying the weight per unit volume of each element times its actual volume.

Typically you assign a self-weight multiplier of 1 to one load case only and a self-weight multiplier of 0 to all other load cases. You may want to assign self-weight to its very own load case, perhaps named SELF, or you may want to include the self-weight in the same load case as your other dead load. Either is acceptable.

Important: If you assign a self-weight multiplier to more than one load case then you run the risk of double-counting (or more) your self-weight when you combine load cases to create load combinations.

The self-weight multiplier of 1 means that the self-weight of each element times a factor of 1.0 is automatically included in the load case. You could, if desired, use a self-weight multiplier other than 1 to represent the self-weight but this is not typically what you should do.

The self-weight acts in the gravity direction, that is the global -Z direction, only.

Response Spectrum Load Case

Note:

There is no limit on the number of load cases or load combinations you can create in ETABS.

A response spectrum load case defines the response spectrum analysis to be performed. When your analysis includes response spectrum load cases you must be sure you have requested that ETABS calculate the vibration modes of the model by either eigenvector analysis or ritz-vector analysis. We recommend that you use ritz-vector analysis when analyzing response spectrum load cases. Click the **Analyze menu > Set Analysis Options** command, check the Dynamic Analysis check box and click the **Set Dynamic Parameters** button to specify the number of

modes and type of modal analysis. See Chapter 15 for more information.

Any number of response spectrum load cases may be used in an ETABS analysis. For design ETABS assumes that all response spectrum load cases are earthquake-type loads.

To define a response spectrum load case first use the **Define menu > Response Spectrum Functions** command to define a response spectrum. Next use the **Define menu > Response Spectrum Cases** command to define the actual response spectrum load case. See the sections titled "Response Spectrum Functions" and "Response Spectrum Cases" in Chapter 11 for more information.

Time History Load Case

A time history load case defines the time history analysis to be performed. When your analysis includes time history load cases you must be sure you have requested that ETABS calculate the vibration modes of the model by either eigenvector analysis or ritz-vector analysis. We recommend that you use ritz-vector analysis when analyzing response spectrum load cases.

Any number of time history load cases may be used in an ETABS analysis. ETABS does not by default assume that a time history load case is an earthquake-type load for design.

To define a time history load case first use the **Define menu > Time History Functions** command to define an input time history. Next use the **Define menu > Time History Cases** command to define the actual time history load case. See the sections titled "Time History Functions" and "Time History Cases" in Chapter 11 for more information.

Static Nonlinear Load Case

A static nonlinear load case defines the static nonlinear analysis to be performed. Any number of static nonlinear load cases may be used in an ETABS analysis. ETABS does not by default assume that a static nonlinear load case is an earthquake-type load for design.

To define a static nonlinear load case use the **Define menu > Static Nonlinear/Pushover Cases** command.

Note that you must have the nonlinear version of ETABS to perform static nonlinear analysis. Full documentation of static nonlinear analysis is beyond the scope of this manual.

Live Load Reduction

General

You can specify some preferences for how ETABS handles live load reduction using the **Options menu > Preferences > Live Load Reduction** command. These preferences apply to the entire model. See the subsection titled "Live Load reduction" under the section titled "Preferences" in Chapter 18 for more information.

Load Combinations

General

A load combination, called a Combo for short, can consist of any of the following:

- A combination of the results from one or more load cases.
- A combination of the results from one or more previously defined Combos.

**Tip:**

Forces and displacements associated with a mode shape can be obtained by including the mode in a Combo.

- A combination of results from one or more Modes.
- A combination of the results from load cases and/or previously defined Combos and/or Modes.

You define Combos using the **Define menu > Load Combinations** command. You can define Combos before or after you run an analysis.

Combo results include all displacements and forces at the points (joints) and internal forces or stresses in the elements. You may specify any number of Combos. You assign a unique label to each Combo.

Output Values Produced by Load Combinations

Each load case, mode or Combo that is included in a Combo supplies either one or two values to the Combo for each response quantity:

- Static load cases, static nonlinear load cases, modes and additive Combos of Loads provide a single value. It is sometimes easier to think of this single value as two values, a maximum and a minimum, where the two values just happen to be the same. This thought process is used in the Combo examples discussed below.
- All other load cases and Combos provide two values: a maximum and minimum. For some types of Combos, both values are used. For other types of Combos, only the value with the larger absolute value is used.

For response-spectrum analyses, the maximum value is the result of the analysis, and the minimum value is just the negative of the maximum.

For time-history analyses, the values used are the maximum and minimum values attained at any time during the analysis.

For nonlinear static analyses, the values used are the maximum and minimum values attained at any time during the analysis.

Each load case, mode or Combo that is included in a Combo is multiplied by a specified scale factor before being included in the Combo.

Types of Load Combinations

Four types of Combos are available. For each individual response quantity (force, stress, or displacement component) the maximum and minimum Combo values are calculated as follows:

- **Additive type:** The Combo maximum is an algebraic linear combination of the maximum values for each of the included analysis cases. Similarly, Combo minimum is an algebraic linear combination of the minimum values for each of the included analysis cases.
- **Absolute type:** The Combo maximum is the sum of the larger absolute values for each of the included analysis cases. The Combo minimum is the negative of the Combo maximum.
- **SRSS type:** The Combo maximum is the square root of the sum of the squares of the larger absolute values for each of the included analysis cases. The Combo minimum is the negative of the Combo maximum.
- **Envelope type:** The Combo maximum is the maximum of all of the maximum values for each of the included analysis cases. Similarly, the Combo minimum is the minimum of all of the minimum values for each of the included analysis cases. See the bullet item titled "Diagram Fill" in the subsection titled "Output Colors" in Chapter 18 for additional information.

Only additive Combos of single-valued analysis cases produce a single-valued result, i.e., the maximum and minimum values are equal. Thus only Combos of static load cases, and other single-valued Combos are single-valued. All other Combos will generally have different maximum and minimum values.



Tip:

When you define a Combo that includes other Combos, make sure that the combo is not recursive. For example, assume you define two combos called C1 and C2. Make sure that C1 does not contain C2 while at the same time C2 contains C1.

These would be recursive Combos that ETABS can not handle.

Examples

For example, suppose that the values, after scaling, for the displacement at a particular point are 3.5 for a static load case labeled LL and 2.0 for Response spectrum load case labeled QUAKE. Suppose that these two load cases have been included in an additive-type Combo called ADDCOMB and an envelope-type Combo called ENVCOMB. The results for the displacement at the point are computed as follows:

27

- ADDCOMB: The maximum is $3.5 + 2.0 = 5.5$, and the minimum is $3.5 - 2.0 = 1.5$
- ENVCOMB: The maximum is $\max(3.5, 2.0) = 3.5$, and the minimum is $\min(3.5, -2.0) = -2.0$

As another example, suppose that static load cases GRAV, WINDX and WINDY are gravity load and two perpendicular, transverse wind loads, respectively; and that EQ is a response-spectrum load case. The following four Combos could be defined:

- WIND: An SRSS-type Combo of the two wind load cases, WINDX and WINDY. The maximum and minimum results produced for each response quantity in this Combo are equal and opposite.
- GRAVEQ: An additive-type Combo of the gravity load, GRAV, and the response-spectrum results, EQ. The Combo automatically accounts for the positive and negative senses of the earthquake load.
- GRAVWIN: An additive-type Combo of the gravity load, GRAV, and the wind load given by Combo WIND, which already accounts for the positive and negative senses of the load.
- SEVERE: An envelope-type Combo that produces the worst case of the two additive Combos, GRAVEQ and GRAVWIN.



Note:

The examples in the remainder of this section illustrate some of the powerful features available for load combinations used in ETABS.

Suppose that the values of axial force in a frame element, after scaling, are 10, 5, 3, and 7 for load cases GRAV, WINDX, WINDY, and EQ, respectively. The following results for axial force are obtained for the Combos above:

- WIND: maximum $\sqrt{5^2 + 3^2} = 5.8$, minimum = -5.8
- GRAVEQ: maximum = $10 + 7 = 17$, minimum = $10 - 7 = 3$
- GRAVWIN: maximum = $10 + 5.8 = 15.8$, minimum = $10 - 5.8 = 4.2$
- SEVERE: maximum = $\max(17, 15.8) = 17$, minimum = $\min(3, 4.2) = 3$

As you can see, using Combos of Combos gives you considerable power and flexibility in how you can combine the results of the various analysis cases.

Design Load Combinations

When Combos are used for design, they may be treated somewhat differently than has been described here for analysis output purposes:

- For time history analysis you can either design based on the maximum and minimum envelope values obtained from the time history analysis or you can design at each and every time step. Designing based on envelope values can be conservative in some cases because you have lost correspondence between the various force components, however, it is relatively quick.

The advantage of designing at each and every time step is that you get the correct correspondence between the various force components, however, it can be very time consuming. If you wish to design based on every time step you may want to do preliminary designs based on envelope values and then do a final design based on each time step of the time history.

- For nonlinear static analysis all design is based on considering every step separately. This maintains the correspondence between forces. Typically there are not very many steps in a static nonlinear analysis so the time penalty that results from considering each step is small.

Restrictions for Load Case and Load Combination Labels

ETABS has some restrictions on the labeling of load cases and load combinations. No load case or load combination can have the same label. A load case must have a label that is unique from all other load cases and all load combinations. Similarly, a load combination must have a label that is unique from all other load combinations and all load cases. Finally, the word Mode is a reserved word that can not be used for either a load case label or a load combination label.

Mass

In ETABS a modal (eigenvector or ritz-vector) analysis is based in part on the mass of the building. The weight of the building used for creating automatic static lateral seismic loads is based on the building mass. Also in a static nonlinear pushover analysis the conversion of a force-displacement plot to a spectral acceleration versus spectral displacement plot is based in part on the mass of the building. For these types of analyses you must have mass defined for the building.



Tip:

You can define the building mass based on a specified load combination if you wish.

You specify the mass source that you want to use through the **Define menu > Mass Source** command. There are two options available for defining the source of the mass of a building. You can use either one option or the other but you can not use both simultaneously. The two options are:

- **Define the mass element masses and additional masses:** One of the items specified in the material properties is a mass per unit volume. Each structural element has a material property associated with it. Thus the building mass associated with the element mass is determined by multiplying the volume of each structural element times its specified mass per unit volume.

Additional mass can be specified on the Assign menu for area, line and point objects. Any additional mass assigned to these objects is added to the element mass to give the total mass of the building. The additional mass might be used to account for partitions, cladding, etc.

The total mass is applied to each joint in the structure on a tributary area basis in all three translational directions, although, if you indicated that only lateral mass is to be included (considered), then the mass is only active in the two horizontal translational directions even though it is assigned in all three translational directions.

ETABS defaults to defining the mass from element masses plus additional masses.

- **Define the mass from a specified load combination:** You can specify a load combination that essentially defines the mass of the structure. The mass is equal to the weight defined by the load combination divided by the gravitational multiplier, g. Only the global Z-direction loads are considered when calculating the mass.

This mass is applied to each joint in the structure on a tributary area basis in all three translational directions, although, if you indicated that only lateral mass is to be included (considered), then the mass is only active in the two horizontal translational directions even though it is assigned in all three translational directions.

Net downward loads acting on a joint are considered a positive mass. If the net load acting up on a joint is upward then the mass at that joint is set to zero. You can not have a negative mass in ETABS.

Note that it is possible to assign additional mass to area, line and point objects. This additional mass is only considered by ETABS if the "Define the mass from specified material masses" mass source option is used. The additional mass is not considered by ETABS if you specify that the mass source is a load combination.

Additional masses can be input as negative. The additional masses are added to the masses determined from the material property masses. If, as a result of including negative additional mass the net mass at a joint is negative then ETABS sets the mass at that joint to zero.



Automatic Seismic Loads

General

This chapter documents the automatic seismic lateral static load cases that can be generated by ETABS. Automatic seismic loads can be generated in the global X or global Y direction for the following codes:



Note:

The automatic seismic loads generated by ETABS are static loads.

- 1994 UBC (American)
- 1997 UBC (American)
- 1997 UBC Isolated Building (American)
- 1996 BOCA (American)
- 1995 NBCC (Canadian)
- IBC 2000 (American)
- 1997 NEHRP (American)

Each of these is discussed later in this chapter. In addition user-defined seismic loads can be automatically generated in the global X and Y directions.

Defining Automatic Seismic Load Cases

The automatic seismic static load cases are defined using the **Define menu > Static Load Cases** command. This command brings up the Define Static Load Case Names dialog box. In this dialog box you specify a name for a load case, a design type, a self-weight multiplier and in some an Auto Lateral Load. See the subsections titled "Design Type" and "Self-Weight Multiplier" in Chapter 27 for additional information on those items.

When you specify the design type for a static load as Quake the Auto Lateral Load box becomes active and you can choose from any of the codes mentioned above. If you do not want the Quake load to be an automatic lateral load then select None in the Auto Lateral Load box.

If you select a code in the Auto Lateral Load box, then when you click the **Add New Load** or **Modify Load** buttons a dialog box pops up that allows you to specify the appropriate parameters for that particular code. If you want to modify the parameters for an existing automatic lateral load then highlight the load in the Define Static Load Case Names dialog box and click the **Modify Load** button.

Each automatic static lateral load that you define must be in a separate load case. You can not have two automatic static lateral loads in the same load case. You can, however, add additional user-defined loads to a load case that includes an automatic static lateral load.

You must define a separate automatic static load case for each direction and, in the case of seismic loading, each eccentricity that you want to consider. For example, if you want to define automatic seismic lateral loads based on the 1997 UBC for X-direction load with no eccentricity, X-direction load with +5% eccentricity, X-direction load with -5% eccentricity, Y-direction load with no eccentricity, Y-direction load with +5% eccentricity



Tip:

Note that the actual forces associated with an automatic static lateral load are not calculated until you run the analysis.

and Y-direction load with -5% eccentricity, then you need to define six separate load cases.

Note that the actual forces associated with an automatic static lateral load are not calculated until you run the analysis. Thus you can not view the resultant automatic lateral loads until after you have run an analysis.

Automatic Seismic Load Cases

The dialog boxes defining the automatic seismic loads are broken into two halves. The left half of the dialog box defines items that are dependent on the direction of the loading. The right half of the dialog box defines factors and coefficients that are not, at least in ETABS, dependent on the direction of loading.

Some of the direction-dependent data is common to all of the codes. This includes the direction and eccentricity data and the story range data. This data is described once below. Other direction-dependent data including building period information and other factors and coefficients and the non-direction-dependent factors and coefficients are described separately for each code later in this chapter.

The weight of the building used in the calculation of automatic seismic loads is based on the specified mass of the building.

Distribution of Automatic Seismic Loads at a Story Level

The method that ETABS uses to calculate the seismic base shear and the associated story lateral forces is documented separately for each code later in this chapter. Once ETABS has calculated a story level force for an automatic seismic load that force is apportioned to each joint at the story level elevation in proportion to its mass.

Direction and Eccentricity Data

In the direction and eccentricity data you can choose one of six options for the direction of load and eccentricity associated with the load case. The options are:

- X-direction load with no eccentricity
- X-direction load with + eccentricity
- X-direction load with - eccentricity
- Y-direction load with no eccentricity
- Y-direction load with + eccentricity
- Y-direction load with - eccentricity

28

You can choose any one of these options, but not more than one, for a particular load case.



Tip:

In some cases you may want to create special point objects at the edges of your rigid diaphragm, even if there is no structure in the model at the diaphragm edges. These point objects can then be assigned to the rigid diaphragm and then ETABS will correctly calculate the diaphragm width and the eccentric moment.

If you choose one of the options with + or - eccentricity then you specify a percent eccentricity that is applicable to all rigid diaphragms. The default percentage is 5%. The eccentricity options only have meaning if your model has rigid diaphragms. ETABS ignores eccentricities where rigid diaphragms are not present.

Where rigid diaphragms are present ETABS calculates a maximum width of the diaphragm perpendicular to the direction of the seismic loading. This width is calculated by finding the maximum and minimum X or Y coordinates (depending on direction of load considered) of the points that are part of the rigid diaphragm constraint and determining the distance between these maximum and minimum values. See the subsection titled "Rigid Diaphragms" in Chapter 23 for more information about the points that are part of the rigid diaphragm constraint when rigid diaphragms are defined using an assignment to an area object. See the subsection titled "Rigid Diaphragms" in Chapter 25 for more information about rigid diaphragms defined as an assignment to one or more point objects.

Once the appropriate diaphragm width is determined ETABS then applies a moment that is equal to the specified percent eccentricity times the maximum width of the diaphragm perpendicular to the direction of the seismic loading times the total lateral force applied to the diaphragm. This moment is applied about the rigid diaphragm center of mass to account for the eccentricity.

When defining eccentricities you can click the **Override** button to override the eccentricity for any rigid diaphragm at any level. Thus you could conceivably have different percent eccentricities specified at different story levels. **Note that when you override the eccentricities you input an actual distance from the center of mass of the rigid diaphragm, not a percentage.**

When you have overridden the eccentricities ETABS calculates the eccentric moment as the specified eccentricity distance times the total lateral force applied to the diaphragm. This moment is again applied about the rigid diaphragm center of mass to account for the eccentricity.

Story Range Data

In the story range data you specify a top story and a bottom story. This specifies the elevation range over which the automatic static lateral loads are calculated.

In most instances you would specify the top story as the uppermost level in the building, typically the roof. In some cases you may want to specify a lower level as the top story for automatic seismic loads. For example if you have included penthouses in your building model you may want to have the automatic lateral load calculation done based on the roof level (not penthouse roof level) being the top story, and then add in additional user-defined load to the load case to account for the penthouses.

The bottom level would typically be the base level, but this may not always be the case. For example, if your building has several below-grade levels, and you are assuming that the seismic loads are transferred to the ground at ground level, then you may specify the bottom story to be above the base of the building.

Note that no seismic loads are calculated for the bottom story. They are calculated for the first story above the bottom story and for all stories up to and including the top story.

By default the bottom story is the base of the building and the top story is the uppermost level of the building.

1994 UBC Seismic Loads

Options for 1994 UBC Building Period

Three options are provided for the building period used in calculating the 1994 UBC automatic seismic loads. They are:

Method A: Calculate the period based on the Method A period discussed in Section 1628.2.2 of the 1994 UBC. The period is calculated using Equation 28-1 (1994 UBC Equation 28-3). The value used for C_t is user input and h_n is determined by ETABS from the input story level heights.

$$T_A = C_t (h_n)^{3/4} \quad \text{Eqn. 28-1}$$

Note that the item C_t is always input in English units as specified in the code. A typical range of values for C_t is 0.020 to 0.035. The height h_n is measured from the elevation of the (top of the) specified bottom story level to the (top of the) specified top story level.

Program Calculated: ETABS starts with the period of the mode calculated to have the largest participation factor in the direction that loads are being calculated (X or Y). Call this period T_{ETABS} . ETABS also calculates a period based on the Method A period discussed in Section 1628.2.2 of the 1994 UBC. The period is calculated using Equation 28-1 (1994 UBC Equation 28-3). The value used for C_t is user input and h_n is determined by ETABS from the input story level heights. Call this period T_A . The building period, T , that ETABS chooses depends on the seismic zone factor, Z .

If $Z \geq 0.35$ (Zone 4) then:

If $T_{ETABS} \leq 1.30T_A$ then $T = T_{ETABS}$.

If $T_{ETABS} > 1.30T_A$ then $T = T_A$.

If $Z < 0.35$ (Zone 1, 2 or 3) then:

If $T_{ETABS} \leq 1.40T_A$ then $T = T_{ETABS}$.

If $T_{ETABS} > 1.40T_A$ then $T = T_A$.

Note:

Always input C_t in English units regardless of the current units for your model.

User Defined: In this case you input a building period. ETABS uses this period in the calculations. It does not compare it against the Method A period. It is assumed that you have already done this comparison before specifying the period.

Other Input Factors and Coefficients

The R_w factor is direction dependent. It is specified in 1994 UBC Table 16-N. A typical range of values for R_w is 4 to 12.

The seismic zone factor, Z, can be input per the code which restricts it to one of the following values: 0.075, 0.15, 0.2, 0.3, 0.4 as specified in 1994 UBC Table 16-I. Alternatively the Z factor can be user-defined which allows any value to be input.

The site coefficient for soil characteristics, S, can be 1, 1.2, 1.5 or 2. These correspond to soil types S_1 , S_2 , S_3 and S_4 in Table 16-J of the 1994 UBC. No other values can be input.

The seismic importance factor, I can be input as any value. See 1994 UBC Table 16-K. A typical range of values for I is 1.00 to 1.25.

Algorithm for 1994 UBC Seismic Loads

The algorithm for determining 1994 UBC seismic loads is based on Chapter 16, Section 1628 of the 1994 UBC. ETABS calculates a period as described in the previous section titled "Options for 1994 UBC Building Period."

A numerical coefficient, C, is calculated using Equation 28-2 (1994 UBC Equation 28-2).

$$C = \frac{1.25S}{T^{2/3}} \quad \text{Eqn. 28-2}$$

where,

S = Site coefficient for soil characteristics.

T = Building period.

If the value of C exceeds 2.75 then C is set equal to 2.75 for use in Equation 28-3. If the value of C/R_w is less than 0.075 then for use in Equation 28-3 it is set equal to 0.075.

The base shear, V, is calculated from Equation 28-3 (1994 UBC Equation 28-1).

$$V = \frac{ZIC}{R_w} W \quad \text{Eqn. 28-3}$$

where,

Z = Seismic zone factor.

I = Importance factor.

C = Numerical coefficient calculated in Equation 28-2.

R_w = Numerical factor specified in UBC Table 16-N.

W = Weight of the building (based on specified mass).

Note that the weight, W, that ETABS uses in Equation 28-3 is derived from the building mass.

The total base shear, V, is broken into a concentrated force applied to the top of the building and forces applied at each story level in accordance with Equation 28-4 (1994 UBC Equation 28-6):

$$V = F_t + \sum_{\text{story}=1}^n F_{\text{story}} \quad \text{Eqn. 28-4}$$

where,

V = Building base shear.

F_t = Concentrated force at the top of the building.

F_{story} = Portion of base shear applied to a story level.

n = Number of story levels in the building.

The concentrated force at the top of the building, F_t , is calculated as shown in Equation 28-5 (1994 UBC Equation 28-7):

$$\begin{aligned} \text{If } T \leq 0.7 \text{ sec, then } F_t &= 0 \\ \text{If } T > 0.7 \text{ sec, then } F_t &= 0.07TV \leq 0.25V \end{aligned} \quad \text{Eqn. 28-5}$$

where,

T = Building period.

V = Building base shear.

The remaining portion of the base shear, ($V - F_t$), is distributed over the height of the building in accordance with Equation 28-6 (1994 UBC Equation 28-8):

$$F_{\text{story}} = \frac{(V - F_t) w_{\text{story}} h_{\text{story}}}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}} \quad \text{Eqn. 28-6}$$

where,

F_{story} = Portion of base shear applied to a story level.

V = Building base shear.

F_t = Concentrated force at the top of the building.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

n = Number of story levels in the building.

1997 UBC Seismic Loads

Options for 1997 UBC Building Period

Three options are provided for the building period used in calculating the 1997 UBC automatic seismic loads. They are identical to the options previously described for the 1994 UBC in the subsection titled "Options for 1994 UBC Building Period."

28

Other Input Factors and Coefficients

The overstrength factor, R , and the force factor, Ω , are direction dependent. Both are specified in 1997 UBC Table 16-N. A typical range of values for R is 2.8 to 8.5. A typical range of values for Ω is 2.2 to 2.8.

The seismic coefficients C_a and C_v can either be determined per the code or they can be user-defined. If C_a and C_v are user-defined then you simply specify values for them. A typical range of values for C_a is 0.06 to 0.40 and larger if the near source factor N_a exceeds 1.0. A typical range of values for C_v is 0.06 to 0.96 and larger if the near source factor N_v exceeds 1.0.

If C_a and C_v are determined per the code then you specify a soil profile type and a seismic zone factor. Based on the input soil profile type and a seismic zone factor ETABS determines C_a from 1997 UBC Table 16-Q and C_v from 1997 UBC Table 16-R.

The soil profile type can be S_A , S_B , S_C , S_D or S_E . These correspond to soil types S_A , S_B , S_C , S_D and S_E in Table 16-J of the 1997 UBC. No other values can be input. Note that soil profile type S_F is not allowed for the automatic 1997 UBC seismic loads.

The seismic zone factor, Z , is restricted to one of the following values: 0.075, 0.15, 0.2, 0.3, 0.4 as specified in 1997 UBC Table 16-I.

Note that in 1997 UBC Table 16-Q the C_a value for $Z=0.4$ has an additional factor, N_a . Similarly, in 1997 UBC Table 16-R the C_v value for $Z=0.4$ has an additional factor, N_v . The values for the near source factors, N_a and N_v , can either be determined per the

code or they can be user-defined. If N_a and N_v are user-defined then you simply specify values for them. If they are determined per the code then you specify a seismic source type and a distance to the closest known seismic source. Based on the input seismic source type and distance to the source ETABS determines N_a from 1997 UBC Table 16-S and N_v from 1997 UBC Table 16-T. ETABS uses linear interpolation for specified distances between those included in 1997 UBC Tables 16-S and 16-T.

Note:

Always input the distance to the closest known seismic source in kilometers (km) regardless of the current units for your model.

The seismic source type can be A, B or C. These correspond to seismic source types A, B and C in Table 16-U of the 1997 UBC. No other values can be input.

The distance to the closest known seismic source should be input in kilometers (km).

The seismic importance factor, I can be input as any value. See 1997 UBC Table 16-K. Note that the value from Table 16-K to be input for automatic seismic loads is I, not I_p or I_w . A typical range of values for I is 1.00 to 1.25.

Algorithm for 1997 UBC Seismic Loads

The algorithm for determining 1997 UBC seismic loads is based on Chapter 16, Section 1630.2 of the 1997 UBC. ETABS calculates a period as described in the previous section titled "Options for 1997 UBC Building Period."

Initially the total design base shear, V, is calculated using Equation 28-7 (1997 UBC Equation 30-4). This base shear value is then checked against the limits specified in Equations 28-8, 28-9 and 28-10 and modified as necessary to obtain the final base shear.

$$V = \frac{C_v I}{RT} W \quad \text{Eqn. 28-7}$$

where,

C_v = 1997 UBC seismic coefficient, C_v .

I = Importance factor.

R = Overstrength factor specified in UBC Table 16-N.

T = Building period.

W = Weight of the building (based on specified mass).

The total design base shear, V, need not exceed that specified in Equation 28-8 (1997 UBC Equation 30-5). If the base shear calculated per Equation 28-7 exceeds that calculated per Equation 28-8 then ETABS sets the base shear equal to that calculated per Equation 28-8.

$$V = \frac{2.5C_a I}{R} W \quad \text{Eqn. 28-8}$$

where,

C_a = 1997 UBC seismic coefficient, C_a .

and all other terms are as described for Equation 28-7.

The total design base shear, V, can not be less than that specified in Equation 28-9 (1997 UBC Equation 30-6). If the base shear calculated per Equation 28-9 exceeds that calculated per Equation 28-7 then ETABS sets the base shear equal to that calculated per Equation 28-9.

$$V = 0.11C_a I W \quad \text{Eqn. 28-9}$$

where all terms are as previously described for Equations 28-7 and 28-8.

Finally, if the building is in seismic zone 4, the total design base shear, V, can not be less than that specified in Equation 28-10 (1997 UBC Equation 30-7). If the building is in seismic zone 4 and the base shear calculated per Equation 28-10 exceeds that calculated per Equations 28-7 and 28-9 then ETABS sets the base shear equal to that calculated per Equation 28-10.

$$V = \frac{0.8Z N_v I}{R} W \quad \text{Eqn. 28-10}$$

where,

Z = Seismic zone factor (0.40).

N_v = Near source factor, N_v .

I = Importance factor.

R = Overstrength factor specified in UBC Table 16-N.

W = Weight of the building (based on specified mass).

Note that ETABS only checks Equation 28-10 if the seismic coefficients, C_a and C_v , are determined per the code and the seismic zone factor Z is specified as 0.40. If the C_a and C_v coefficients are user specified then ETABS never checks Equation 28-10.

Note that the weight, W , that ETABS uses in Equations 28-7 through 28-10 is derived from the building mass.

The total base shear, V , is broken into a concentrated force applied to the top of the building and forces applied at each story level in accordance with Equation 28-11 (1997 UBC Equation 30-13):

$$V = F_t + \sum_{\text{story}=1}^n F_{\text{story}} \quad \text{Eqn. 28-11}$$

where,

V = Building base shear.

F_t = Concentrated force at the top of the building.

F_{story} = Portion of base shear applied to a story level.

n = Number of story levels in the building.

The concentrated force at the top of the building, F_t , is calculated as shown in Equation 28-12 (1997 UBC Equation 30-14):

If $T \leq 0.7$ sec, then $F_t = 0$

Eqn. 28-12

If $T > 0.7$ sec, then $F_t = 0.07TV \leq 0.25V$

where,

T = Building period.

V = Building base shear.

The remaining portion of the base shear, ($V - F_t$), is distributed over the height of the building in accordance with Equation 28-13 (1997 UBC Equation 30-15):

$$F_{\text{story}} = \frac{(V - F_t) w_{\text{story}} h_{\text{story}}}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}} \quad \text{Eqn. 28-13}$$

where,

F_{story} = Portion of base shear applied to a story level.

V = Building base shear.

F_t = Concentrated force at the top of the building.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

n = Number of story levels in the building.

1997 UBC Isolated Building Seismic Loads

Other Input Factors and Coefficients

For 1997 UBC isolated building seismic loads the bottom story should be input as the story at the top of the isolators.

The overstrength factor, R_i , is direction dependent. It relates to the structure above the isolation interface. It is specified in 1997 UBC Table A-16-E which is in Appendix Chapter 16, Division IV. A typical range of values for R_i is 1.6 to 2.0.

The coefficient for damping, B_d , is direction dependent. It should be specified based on an assumed effective damping using 1997 UBC Table A-16-C which is in Appendix Chapter 16, Division IV. A typical range of values for B_d is 0.8 to 2.0.

The maximum effective stiffness and minimum effective stiffness items refer to the maximum and minimum effective stiffness of the isolation system (not individual isolators) at the design displacement level (not the maximum displacement level). They correspond to the terms $K_{D\max}$ and $K_{D\min}$, respectively, in Appendix Chapter 16, Division IV.

The seismic coefficient C_{vd} can either be determined per the code or it can be user-defined. If C_{vd} is user-defined then you simply specify a value for it. A typical range of values for C_{vd} is 0.06 to 0.96 and larger if the near source factor N_v exceeds 1.0.

If C_{vd} is determined per the code then you specify a soil profile type and a seismic zone factor. Based on the input soil profile type and a seismic zone factor ETABS determines C_{vd} from 1997 UBC Table 16-R which is in Chapter 16, not Appendix Chapter 16, Division IV.

Note that in 1997 UBC Table 16-R the C_v value for $Z=0.4$ has an additional factor, N_v . The value for this near source factor, N_v , can either be determined per the code or it can be user-defined. If N_v is user-defined then you simply specify a value for it. If it is determined per the code then you specify a seismic source type and a distance to the closest known seismic source. Based on the input seismic source type and distance to the source ETABS determines N_v from 1997 UBC Table 16-T. ETABS uses linear interpolation for specified distances between those included in 1997 UBC Table 16-T.

Algorithm for 1997 UBC Isolated Building Seismic Loads

The algorithm for determining 1997 UBC seismic loads for isolated buildings is based on Appendix Chapter 16, Division IV, Sections 1658.3 and 1658.4 of the 1997 UBC.

The effective period at the design displacement, T_D , is determined from Equation 28-14 (1997 UBC Equation 58-2).

$$T_D = 2\pi \sqrt{\frac{W}{k_{D\min} g}} \quad \text{Eqn. 28-14}$$

where,

W = Weight of the building (based on specified mass).

$k_{D\min}$ = Minimum effective stiffness of the isolation system at the design displacement.

g = Gravity constant, (e.g., 386.4 in/sec², 9.81 m/sec², etc.).

The design displacement at the center of rigidity of the isolation system, D_D , is determined from Equation 28-15 (1997 UBC Equation 58-1).

$$D_D = \frac{\left(\frac{g}{4\pi^2}\right) C_{vd} T_D}{B_d} \quad \text{Eqn. 28-15}$$

where,

g = Gravity constant, (e.g., 386.4 in/sec², 9.81 m/sec², etc.).

C_{vd} = Seismic coefficient, C_{vd} .

T_D = Effective period at the design displacement.

B_d = Coefficient for damping.

Note:

The limits on V_s specified in 1997 UBC section 1658.4.3 are not considered by ETABS.

The base shear, V_s , is calculated from Equation 28-16 (1997 UBC Equation 58-8).

$$V_s = \frac{k_{D\max} D_D}{R_i} \quad \text{Eqn. 28-16}$$

Note that Equation 28-16 gives a force level that is applicable for the structure above the isolation system. If you want a force level that is applicable to the isolation system per 1997 UBC Equation 58-7 then you should create a different load combination with a scale factor of R_i for the seismic load.

Also note that the limits on V_s specified in 1997 UBC section 1658.4.3 are not considered by ETABS.

The total base shear, V_s , is distributed over the height of the building in accordance with Equation 28-17 (1997 UBC Equation 58-9):

$$F_{\text{story}} = \frac{V_s w_{\text{story}} h_{\text{story}}}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}} \quad \text{Eqn. 28-17}$$

where,

F_{story} = Portion of base shear applied to a story level.

V_s = Building base shear per Equation 28-16.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

n = Number of story levels in the building.

1996 BOCA Seismic Loads

Options for 1996 BOCA Building Period

Three options are provided for the building period used in calculating the 1996 BOCA automatic seismic loads. They are:

Note:

Always input C_T in English units regardless of the current units for your model.

Approximate: Calculate the approximate period, T_a , based on the approximate formula discussed in Section 1610.4.1.2.1 of the 1996 BOCA. The period is calculated using Equation 28-18. The value used for C_T is user input and h_n is determined by ETABS from the input story level heights.

$$T_a = C_T (h_n)^{3/4} \quad \text{Eqn. 28-18}$$

Note that the item C_T is always input in English units as specified in the code. A typical range of values for C_T is 0.020 to 0.035. The height h_n is measured from the elevation of the (top of the) specified bottom story level to the (top of the) specified top story level.

Program Calculated: ETABS starts with the period of the mode calculated to have the largest participation factor in the direction that loads are being calculated (X or Y). Call this period T_{ETABS} . ETABS also calculates a period based on the approximate formula discussed in Section 1610.4.1.2.1 of the 1996 BOCA. The period is calculated using Equation 28-1. The value used for C_T is user input and h_n is determined by ETABS from the input story level heights. Call this period T_a .

ETABS also determines a value for the coefficient for the upper limit on the calculated period, C_a , using Table 1610.4.1.2 in the 1996 BOCA. Note that the value used for C_a depends on the specified value for the effective peak velocity-related coefficient, A_v . ETABS determines C_a using linear interpolation if the specified value of A_v is not in Table 1610.4.1.2. If A_v exceeds 0.40 then C_a is taken as 1.2. If A_v is less than 0.05 then C_a is taken as 1.7.

The building period, T , that ETABS chooses is determined as follows:

If $T_{ETABS} > C_a T_a$ then $T = C_a T_a$.

If $T_{ETABS} \leq C_a T_a$ then $T = T_{ETABS}$.

User Defined: In this case you input a building period. ETABS uses this period in the calculations. It does not compare it against the coefficient for the upper limit on the calculated period times the approximate period ($C_a T_a$). It is assumed that you have already done this comparison before specifying the period.

Other Input Factors and Coefficients

The response modification factor, R , is direction dependent. It is specified in 1996 BOCA Table 1610.3.3. A typical range of values for R is 3 to 8.

Any value can be input for the effective peak acceleration coefficient, A_a . Refer to BOCA section 1610.1.3. A typical range of values for A_a is 0.05 to 0.40.

Any value can be input for the effective peak velocity-related coefficient, A_v . Refer to BOCA section 1610.1.3. A typical range of values for A_v is 0.05 to 0.40.

The soil profile type can be S₁, S₂, S₃ or S₄. These correspond to soil types S₁, S₂, S₃ and S₄ in Table 1610.3.1 of the 1996 BOCA. No other values can be input.

Algorithm for 1996 BOCA Seismic Loads

The algorithm for determining 1996 BOCA seismic loads is based on Section 1610.4.1 of the 1996 BOCA. ETABS calculates a period as described in the previous section titled "Options for 1996 BOCA Building Period."

Initially the seismic coefficient, C_s, is calculated from Equation 28-19. The value of this coefficient is then checked against the limit specified in Equation 28-20 and modified as necessary to obtain the seismic coefficient.

$$C_s = \frac{1.2A_v S}{R T^{2/3}} \quad \text{Eqn. 28-19}$$

where,

A_v = The effective peak velocity-related coefficient.

S = The site coefficient based on the input soil profile type.

R = Response modification factor.

T = Building period.

The seismic coefficient, C_s, need not exceed that specified in Equation 28-20. If the seismic coefficient calculated per Equation 28-19 exceeds that calculated per Equation 28-20 then ETABS sets the seismic coefficient equal to that calculated per Equation 28-20.

$$C_s = \frac{2.5A_a}{R} \quad \text{Eqn. 28-20}$$

where,

A_a = The effective peak acceleration coefficient.

R = Response modification factor.

The base shear is calculated using Equation 28-21.

$$V = C_s W \quad \text{Eqn. 28-21}$$

where,

C_s = Seismic coefficient calculated from Equation 28-19 or 28-20 as appropriate.

W = Weight of the building (based on specified mass).

The base shear, V , is distributed over the height of the building in accordance with Equation 28-22:

$$F_{\text{story}} = \frac{V w_{\text{story}} h_{\text{story}}^k}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}^k} \quad \text{Eqn. 28-22}$$

where,

F_{story} = Portion of base shear applied to a story level.

V = Building base shear.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

k = Exponent applied to building height. The value of k depends on the value of the building period, T , used for determining the base shear. If $T \leq 0.5$ seconds then $k = 1$. If $T > 2.5$ seconds then $k = 2$. If $0.5 \text{ seconds} < T < 2.5$ seconds then k is linearly interpolated between 1 and 2.

n = Number of story levels in the building.

1995 NBCC Seismic Loads

Options for 1995 NBCC Building Period

Five options are provided for the building period used in calculating the 1995 NBCC automatic seismic loads. They are:

Code - Moment Frame: Calculate the period as $0.1 * N$, where N is the number of stories in the building based on the specified top and bottom story levels.

Code - Other: Calculate the period, T, using Equation 28-23:

$$T = \frac{0.09h_n}{\sqrt{D_s}} \quad \text{Eqn. 28-23}$$

where,

h_n = Height of the building measured from the elevation of the (top of the) specified bottom story level to the (top of the) specified top story level measured in meters.

D_s = Length of wall or braced frame which constitutes the main lateral-force-resisting system measured in meters.

Program Calculated - Moment Frame: ETABS uses the period of the mode calculated to have the largest participation factor in the direction that loads are being calculated (X or Y). In addition, ETABS runs a parallel calculation using a period equal to $0.1 * N$, where N is the number of stories in the building based on the specified top and bottom story levels.

ETABS calculates the equivalent lateral force at the base of the structure, V_e , using both periods. Call these values $V_{e\text{-mode}}$ and $V_{e-0.1*N}$. ETABS determines the value of V_e to use as follows:

If $V_{e\text{-mode}} \geq 0.8 V_{e-0.1*N}$ then $V_e = V_{e\text{-mode}}$.

If $V_{e\text{-mode}} < 0.8 V_{e-0.1*N}$ then $V_e = 0.8 V_{e-0.1*N}$.

Program Calculated - Other: ETABS uses the period of the mode calculated to have the largest participation factor in the direction that loads are being calculated (X or Y). In addition, ETABS runs a parallel calculation using a period calculated using Equation 28-23.

ETABS calculates the equivalent lateral force at the base of the structure, V_e , using both periods. Call these values $V_{e\text{-mode}}$ and $V_{e\text{-Eqn. 28-23}}$. ETABS determines the value of V_e to use as follows:

If $V_{e\text{-mode}} \geq 0.8 V_{e\text{-Eqn. 28-23}}$ then $V_e = V_{e\text{-mode}}$.

If $V_{e\text{-mode}} < 0.8 V_{e\text{-Eqn. 28-23}}$ then $V_e = 0.8 V_{e\text{-Eqn. 28-23}}$.

User Defined: In this case you input a building period. ETABS uses this period in the calculations. It does not calculate other values of V_e using this method for comparison against the V_e calculated using your specified period. It is assumed that you have already done this comparison before specifying the period.

Other Input Factors and Coefficients

The force modification factor, R, is direction dependent. It is specified in 1995 NBCC Table 4.1.9.1.B. A typical range of values for R is 1.5 to 4.0.

The acceleration-related seismic zone, Z_a , can be input as 0, 1, 2, 3, 4, 5 or 6. No other input values are allowed.

The velocity-related seismic zone, Z_v , can be input as 0, 1, 2, 3, 4, 5 or 6. No other input values are allowed.

The zonal velocity ratio, v, can either be based on Z_v , or a user-specified value can be input. If it is based on Z_v then ETABS assumes v is equal to 0.00, 0.05, 0.10, 0.15, 0.20, 0.30 or 0.40 for Z_v equal to 0, 1, 2, 3, 4, 5 or 6, respectively.

The importance factor, I, can be input as any value. It is specified in 1995 NBCC Sentence 4.1.9.1(10). A typical range of values for I is 1.0 to 1.5.

The foundation factor, F, can be input as any value. It is specified in 1995 NBCC Table 4.1.9.1.C. A typical range of values for F is 1.0 to 2.0.

Algorithm for 1995 NBCC Seismic Loads

The algorithm for determining 1995 NBCC seismic loads is based on Subsection 4.1.9 of the 1995 NBCC. ETABS calculates a period as described in the previous section titled "Options for 1995 NBCC Building Period."

First ETABS checks if $Z_v = 0$ and $Z_a > 0$. If so, then ETABS sets $Z_v = 1$ and $v = 0.05$ for the calculation of the base shear.

ETABS calculates the seismic response factor, S, based on 1995 NBCC Table 4.1.9.1.A.

ETABS determines the product of the foundation factor, F, and the seismic response factor, S. Call this product FS. If necessary this product is modified as follows:

If $FS > 3$ and $Z_a \leq Z_v$ then $FS = 3$.

If $FS > 4.2$ and $Z_a > Z_v$ then $FS = 4.2$.

ETABS determines the equivalent lateral force representing elastic response per Equation 28-24:

$$V_e = v * FS * I * W \quad \text{Eqn. 28-24}$$

Note that in cases where the building period is program calculated the value of V_e is calculated twice and then one of the calculated values is chosen. See the previous section titled "Options for 1995 NBCC Building Period" for more information.

The minimum lateral seismic force, V, is calculated using Equation 28-25.

$$V = \frac{0.6V_e}{R} \quad \text{Eqn. 28-25}$$

The total base shear, V, is broken into a concentrated force applied to the top of the building and forces applied at each story level in accordance with Equation 28-26:

$$V = F_t + \sum_{\text{story}=1}^n F_{\text{story}} \quad \text{Eqn. 28-26}$$

where,

V = Building base shear.

F_t = Concentrated force at the top of the building.

F_{story} = Portion of base shear applied to a story level.

n = Number of story levels in the building.

The concentrated force at the top of the building, F_t , is calculated as shown in Equation 28-27:

If $T \leq 0.7$ sec, then $F_t = 0$

If $T > 0.7$ sec, then $F_t = 0.07TV \leq 0.25V$

Eqn. 28-27

where,

T = Building period.

V = Building base shear.

The remaining portion of the base shear, ($V - F_t$), is distributed over the height of the building in accordance with Equation 28-28:

$$F_{\text{story}} = \frac{(V - F_t) w_{\text{story}} h_{\text{story}}}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}} \quad \text{Eqn. 28-28}$$

where,

F_{story} = Portion of base shear applied to a story level.

V = Building base shear.

F_t = Concentrated force at the top of the building.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

n = Number of story levels in the building.

Note that the torsional moment discussed in 1995 NBCC Sentence 4.1.9.1(28) are not automatically included by ETABS. However you can override the eccentricities at each diaphragm to specify these torsional moments.

IBC 2000 Seismic Loads

Options for IBC 2000 Building Period

Three options are provided for the building period used in calculating the IBC 2000 automatic seismic loads. They are:

Note:

Always input C_T in English units regardless of the current units for your model.

Approximate Period: Calculate the period based on Equation 28-29 (IBC 2000 Equation 1613.4.9.2-1). The value used for C_T is user input and h_n is determined by ETABS from the input story level heights.

$$T_A = C_T (h_n)^{3/4}$$

Eqn. 28-29

Note that the item C_T is always input in English units as specified in the code. A typical range of values for C_T is 0.020 to 0.035. The height h_n is measured from the elevation of the (top of the) specified bottom story level to the (top of the) specified top story level.

Program Calculated: ETABS starts with the period of the mode calculated to have the largest participation factor in the direction that loads are being calculated (X or Y). Call this period T_{ETABS} . ETABS also calculates a period based on the Equation 28-29 (IBC 2000 Equation 1613.4.9.2-1). The value used for C_T is user input and h_n is determined by ETABS from the input story level heights. Call this period T_A .

ETABS also calculates a coefficient for the upper limit on the calculated period, C_u , based on IBC 2000 Table 1613.4.9.2. Note that ETABS uses linear interpolation to calculate values of C_u where the value of S_{D1} is not specifically specified in Table 1613.4.9.2.

The building period, T , that ETABS chooses is determined as follows:

If $T_{ETABS} \leq C_u T_A$ then $T = T_{ETABS}$.

If $T_{ETABS} > C_u T_A$ then $T = C_u T_A$.

User Defined: In this case you input a building period. ETABS uses this period in the calculations. It does not compare it against $C_u T_A$. It is assumed that you have already done this comparison before specifying the period.

Other Input Factors and Coefficients

The response modification factor, R, and the system overstrength factor, Ω , are direction dependent. Both are specified in IBC 2000 Table 1613.4.1. A typical range of values for R is 2 to 8. A typical range of values for Ω is 2 to 3.

The seismic group can be input as I, II or III. No other values are allowed. See IBC 2000 Table 1613.1.4 for information about the seismic group. ETABS determines the occupancy importance factor, I, from the input seismic group and IBC 2000 Table 1613.1.4.

The seismic coefficients can either be input per the code or they can be user-defined. If the seismic coefficients are per code then you specify a site class, S_s and S_1 . If seismic coefficients are user defined then you specify S_s , S_1 , F_a and F_v .

The site class can be either A, B, C, D or E. Note that site class F is not allowed for ETABS automatic IBC 2000 lateral seismic loads. See IBC 2000 Table 1613.1.2.1.1 for site class definitions.

Note:

In ETABS input S_s and S_1 in g rather than percent g as it is shown on the code maps.

S_s is the mapped spectral acceleration for short periods as determined in IBC 2000 Section 1613.2.1. A typical range of values for S_s is 0 to 3. Note that the seismic maps show S_s in %g with a typical range of 0% to 300%. The input in ETABS is in g. Thus the map values should be divided by 100 when they are input into ETABS. For example, if the map value is 125%g it should be input into ETABS as 1.25g.

S_1 is the mapped spectral acceleration for a one-second period as determined in IBC 2000 Section 1613.2.1. A typical range of values for S_1 is 0 to 2. Note that the seismic maps show S_1 in %g with a typical range of 0% to 200%. The input in ETABS is in g. Thus the map values should be divided by 100 when they are in-

put into ETABS. For example, if the map value is 125%g it should be input into ETABS as 1.25g.

F_a is a site coefficient. If the site coefficients are determined per code then ETABS automatically determines F_a from the site class and S_s based on IBC 2000 Table 1613.2.1.2-1. If site coefficients are user-defined then the F_a is directly input by the user. A typical range of values for F_a is 0.8 to 2.5.

F_v is a site coefficient. If the site coefficients are determined per code then ETABS automatically determines F_v from the site class and S_1 based on IBC 2000 Table 1613.2.1.2-2. If site coefficients are user-defined then the F_v is directly input by the user. A typical range of values for F_v is 0.8 to 3.5.

Algorithm for IBC 2000 Seismic Loads

The algorithm for determining IBC 2000 seismic loads is based on IBC 2000 Section 1613.4.9. ETABS calculates a period as described in the previous section titled "Options for IBC 2000 Building Period."

ETABS begins by calculating the design spectral response acceleration at short periods, S_{DS} , using Equation 28-30. Equation 28-30 is derived by combining IBC 2000 Equations 1613.2.1.2-1 and 1613.2.1.3-1.

$$S_{DS} = \frac{2}{3} F_a S_s \quad \text{Eqn. 28-30}$$

Next ETABS calculates the design spectral response acceleration at a one-second period, S_{D1} , using Equation 28-31. Equation 28-31 is derived by combining IBC 2000 Equations 1613.2.1.2-2 and 1613.2.1.3-2.

$$S_{D1} = \frac{2}{3} F_v S_1 \quad \text{Eqn. 28-31}$$

ETABS determines a seismic design category (A, B, C, D, E or F with A being the least severe and F being the most severe) based on IBC 2000 Section 1613.3.1. A seismic design category is determined based on S_{DS} using IBC 2000 Table 1613.3.1-1. A seismic design category is also determined based on S_{D1} using

IBC 2000 Table 1613.3.1-2. The more severe of the two seismic categories is chosen by ETABS as the seismic design category for the building.

Initially a seismic response coefficient, C_s , is calculated using Equation 28-32 (IBC 2000 Equation 1613.4.9.1-2). This base shear value is then checked against the limits specified in Equations 28-33, 28-34 and 28-35 and modified as necessary to obtain the final base shear.

$$C_s = \frac{S_{DS}}{\frac{R}{I}} \quad \text{Eqn. 28-32}$$

where,

S_{DS} = The design spectral response acceleration at short periods.

R = Response modification factor specified in IBC 2000 Table 1613.4.1.

I = The occupancy importance factor determined in accordance with IBC 2000 Table 1613.1.4.

The seismic response coefficient, C_s , need not exceed that specified in Equation 28-33 (IBC 2000 Equation 1613.4.9.1-3). If the seismic response coefficient calculated per Equation 28-32 exceeds that calculated per Equation 28-33 then ETABS sets the seismic response coefficient, C_s , equal to that calculated per Equation 28-33.

$$C_s = \frac{S_{DI}}{\left(\frac{R}{I}\right)T} \quad \text{Eqn. 28-33}$$

where,

S_{DI} = The design spectral response acceleration at a one second period.

T = The building period used for calculating the base shear.

and all other terms are as described for Equation 28-32.

The seismic response coefficient, C_s , can not be less than that specified in Equation 28-34 (IBC 2000 Equation 1613.4.9.1-4). If the seismic response coefficient calculated per Equation 28-34 exceeds that calculated per Equation 28-32 then ETABS sets the seismic response coefficient equal to that calculated per Equation 28-34.

$$C_s = 0.044 S_{DS} I \quad \text{Eqn. 28-34}$$

where all terms are as previously described for Equation 28-32.

Finally, if the building is in seismic design category E or F, the seismic response coefficient, C_s , can not be less than that specified in Equation 28-35 (IBC 2000 Equation 1613.4.9.1-5). If the building is in seismic design category E or F and the seismic response coefficient calculated per Equation 28-35 exceeds that calculated per Equations 28-32 and 28-34 then ETABS sets the seismic response coefficient equal to that calculated per Equation 28-35.

$$C_s = \frac{0.5S_1}{\frac{R}{I}} \quad \text{Eqn. 28-35}$$

where,

S_1 = The mapped spectral acceleration for a one second period.

and all other terms are as previously described for Equation 28-32.

The base shear, V , is calculated using Equation 28-36 (IBC 2000 Equation 1613.4.9.1-1):

$$V = C_s W \quad \text{Eqn. 28-36}$$

C_s = Seismic response coefficient as determined from one of Equations 28-32 through 28-35 as appropriate.

W = Weight of the building (based on specified mass).

The base shear, V, is distributed over the height of the building in accordance with Equation 28-37. Equation 28-37 is derived by combining IBC 2000 Equations 1613.4.9.3-1 and 1613.4.9.3-2.

$$F_{\text{story}} = \frac{V w_{\text{story}} h_{\text{story}}^k}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}^k} \quad \text{Eqn. 28-37}$$

where,

F_{story} = Portion of base shear applied to a story level.

V = Building base shear.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

k = Exponent applied to building height. The value of k depends on the value of the building period, T, used for determining the base shear. If $T \leq 0.5$ seconds then $k = 1$. If $T > 2.5$ seconds then $k = 2$. If $0.5 < T < 2.5$ seconds then k is linearly interpolated between 1 and 2.

n = Number of story levels in the building.

1997 NEHRP Seismic Loads

Options for 1997 NEHRP Building Period

Three options are provided for the building period used in calculating the 1997 NEHRP automatic seismic loads. They are:

Note:

Always input C_T in English units regardless of the current units for your model.

Approximate Period: Calculate the period based on Equation 28-38 (1997 NEHRP Equation 5.3.3.1-1). The value used for C_T is user input and h_n is determined by ETABS from the input story level heights.

$$T_A = C_T (h_n)^{3/4}$$

Eqn. 28-38

Note that the item C_T is always input in English units as specified in the code. A typical range of values for C_T is 0.020 to 0.035. The height h_n is measured from the elevation of the (top of the) specified bottom story level to the (top of the) specified top story level.

Program Calculated: ETABS starts with the period of the mode calculated to have the largest participation factor in the direction that loads are being calculated (X or Y). Call this period T_{ETABS} . ETABS also calculates a period based on the Equation 28-38 (1997 NEHRP Equation 5.3.3.1-1). The value used for C_T is user input and h_n is determined by ETABS from the input story level heights. Call this period T_A .

ETABS also calculates a coefficient for the upper limit on the calculated period, C_u , based on 1997 NEHRP Table 5.3.3. Note that ETABS uses linear interpolation to calculate values of C_u where the value of S_{DI} is not specifically specified in Table 5.3.3.

The building period, T , that ETABS chooses is determined as follows:

If $T_{ETABS} \leq C_u T_A$ then $T = T_{ETABS}$.

If $T_{ETABS} > C_u T_A$ then $T = C_u T_A$.

User Defined: In this case you input a building period. ETABS uses this period in the calculations. It does not compare it against $C_u T_A$. It is assumed that you have already done this comparison before specifying the period.

Other Input Factors and Coefficients

The response modification coefficient, R , and the system overstrength factor, Ω , are direction dependent. Both are specified in 1997 NEHRP Table 5.2.2. A typical range of values for R is 2 to 8. A typical range of values for Ω is 2 to 3.

The seismic group can be input as I, II or III. No other values are allowed. See 1997 NEHRP Table 1.4 for information about the seismic group. ETABS determines the occupancy importance

**Note:**

*In ETABS input
 S_s and S_1 in g
 rather than
 percent g as it
 is shown on the
 code maps.*

factor, I, from the input seismic group and 1997 NEHRP Table 1.4.

The seismic coefficients can either be input per the code or they can be user-defined. If the seismic coefficients are per code then you specify a site class, S_s and S_1 . If seismic coefficients are user defined then you specify S_s , S_1 , F_a and F_v .

The site class can be either A, B, C, D or E. Note that site class F is not allowed for ETABS automatic 1997 NEHRP lateral seismic loads. See 1997 NEHRP Section 4.1.2.1 for site class definitions.

S_s is the mapped maximum considered spectral acceleration for short periods as determined in 1997 NEHRP Section 4.1.2. A typical range of values for S_s is 0 to 3. Note that the seismic maps show S_s in %g with a typical range of 0% to 300%. The input in ETABS is in g. Thus the map values should be divided by 100 when they are input into ETABS. For example, if the map value is 125%g it should be input into ETABS as 1.25g.

S_1 is the mapped maximum considered spectral acceleration for a one second period as determined in 1997 NEHRP Section 4.1.2. A typical range of values for S_1 is 0 to 2. Note that the seismic maps show S_1 in %g with a typical range of 0% to 200%. The input in ETABS is in g. Thus the map values should be divided by 100 when they are input into ETABS. For example, if the map value is 125%g it should be input into ETABS as 1.25g.

F_a is a site coefficient. If the site coefficients are determined per code then ETABS automatically determines F_a from the site class and S_s based on 1997 NEHRP Table 4.1.2.4a. If site coefficients are user-defined the F_a is directly input by the user. A typical range of values for F_a is 0.8 to 2.5.

F_v is a site coefficient. If the site coefficients are determined per code then ETABS automatically determines F_v from the site class and S_1 based on 1997 NEHRP Table 4.1.2.4b. If site coefficients are user-defined the F_v is directly input by the user. A typical range of values for F_v is 0.8 to 3.5.

Algorithm for 1997 NEHRP Seismic Loads

The algorithm for determining 1997 NEHRP seismic loads is based on 1997 NEHRP Section 5.3. ETABS calculates a period as described in the previous section titled "Options for 1997 NEHRP Building Period."

ETABS begins by calculating the design spectral response acceleration at short periods, S_{DS} , using Equation 28-39. Equation 28-39 is derived by combining 1997 NEHRP Equations 4.1.2.4-1 and 4.1.2.5-1.

$$S_{DS} = \frac{2}{3} F_a S_s \quad \text{Eqn. 28-39}$$

Next ETABS calculates the design spectral response acceleration at a one-second period, S_{D1} , using Equation 28-40. Equation 28-40 is derived by combining 1997 NEHRP Equations 4.1.2.4-2 and 4.1.2.5-2.

$$S_{D1} = \frac{2}{3} F_v S_1 \quad \text{Eqn. 28-40}$$

ETABS determines a seismic design category (A, B, C, D, E or F with A being the least severe and F being the most severe) based on 1997 NEHRP Section 4.2.1. A seismic design category is determined based on S_{DS} using 1997 NEHRP Table 4.2.1a. A seismic design category is also determined based on S_{D1} using 1997 NEHRP Table 4.2.1b. The more severe of the two seismic categories is chosen by ETABS as the seismic design category for the building.

Initially a seismic response coefficient, C_s , is calculated using Equation 28-41 (1997 NEHRP Equation 5.3.2.1-1). This base shear value is then checked against the limits specified in Equations 28-42, 28-43 and 28-44 and modified as necessary to obtain the final base shear.

$$C_s = \frac{S_{DS}}{\frac{R}{I}} \quad \text{Eqn. 28-41}$$

where,

S_{DS} = The design spectral response acceleration at short periods.

R = Response modification factor specified in 1997 NEHRP Table 5.2.2.

I = The occupancy importance factor determined in accordance with 1997 NEHRP Table 1.4.

The seismic response coefficient, C_s , need not exceed that specified in Equation 28-32 (1997 NEHRP Equation 5.3.2.1-2). If the seismic response coefficient calculated per Equation 28-41 exceeds that calculated per Equation 28-42 then ETABS sets the seismic response coefficient, C_s , equal to that calculated per Equation 28-42.

$$C_s = \frac{S_{D1}}{\left(\frac{R}{I}\right)T} \quad \text{Eqn. 28-42}$$

where,

S_{D1} = The design spectral response acceleration at a one second period.

T = The building period used for calculating the base shear.

and all other terms are as described for Equation 28-41.

The seismic response coefficient, C_s , can not be less than that specified in Equation 28-43 (1997 NEHRP Equation 5.3.2.1-3). If the seismic response coefficient calculated per Equation 28-43 exceeds that calculated per Equation 28-41 then ETABS sets the seismic response coefficient equal to that calculated per Equation 28-43.

$$C_s = 0.1 S_{D1} I \quad \text{Eqn. 28-43}$$

where all terms are as previously described for Equations 28-41 and 28-42.

Finally, if the building is in seismic design category E or F, the seismic response coefficient, C_s , can not be less than that specified in Equation 28-44 (1997 NEHRP Equation 5.3.2.1-4). If the building is in seismic design category E or F and the seismic response coefficient calculated per Equation 28-44 exceeds that calculated per Equations 28-41 and 28-43 then ETABS sets the seismic response coefficient equal to that calculated per Equation 28-44.

$$C_s = \frac{0.5S_1}{\frac{R}{I}} \quad \text{Eqn. 28-44}$$

where,

S_1 = The mapped spectral acceleration for a one second period.

and all other terms are as previously described for Equation 28-41.

The base shear, V , is calculated using Equation 28-45 (1997 NEHRP Equation 5.3.2):

$$V = C_s W \quad \text{Eqn. 28-45}$$

C_s = Seismic response coefficient as determined from one of Equations 28-41 through 28-44 as appropriate.

W = Weight of the building (based on specified mass).

The base shear, V , is distributed over the height of the building in accordance with Equation 28-46. Equation 28-46 is derived by combining 1997 NEHRP Equations 5.3.4-1 and 5.3.4-2.

$$F_{\text{story}} = \frac{V w_{\text{story}} h_{\text{story}}^k}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}^k} \quad \text{Eqn. 28-46}$$

where,

- F_{story} = Portion of base shear applied to a story level.
- V = Building base shear.
- w_{story} = Weight of story level (based on specified mass).
- h_{story} = Story height, distance from base of building to story level.
- k = Exponent applied to building height. The value of k depends on the value of the building period, T , used for determining the base shear. If $T \leq 0.5$ seconds then $k = 1$. If $T > 2.5$ seconds then $k = 2$. If $0.5 < T < 2.5$ seconds then k is linearly interpolated between 1 and 2.
- n = Number of story levels in the building.

User-Defined Seismic Loads

Input Factors and Coefficients

The base shear coefficient, C , is direction dependent. This coefficient multiplied times the building weight gives the lateral seismic base shear in the direction specified.

The building height exponent, k , is used as an exponent on the building height when determining the distribution of the base shear over the height of the building. See Equation 28-48.

Algorithm for User-Defined Seismic Loads

The base shear, V , is calculated using Equation 28-47:

$$V = C W \quad \text{Eqn. 28-47}$$

where,

C = User-defined base shear coefficient.

W = Weight of the building (based on specified mass).

The base shear, V, is distributed over the height of the building in accordance with Equation 28-48:

$$F_{\text{story}} = \frac{V w_{\text{story}} h_{\text{story}}^k}{\sum_{\text{story}=1}^n w_{\text{story}} h_{\text{story}}^k} \quad \text{Eqn. 28-48}$$

where,

F_{story} = Portion of base shear applied to a story level.

V = Building base shear.

w_{story} = Weight of story level (based on specified mass).

h_{story} = Story height, distance from base of building to story level.

k = A user-defined exponent.

n = Number of story levels in the building.



Automatic Wind Loads

General



Note:

The automatic wind loads generated by ETABS are static loads.

This chapter documents the automatic wind lateral static load cases that can be generated by ETABS. Automatic wind loads can be generated in any arbitrary horizontal direction for the following codes:

- 1994 UBC (American)
- 1997 UBC (American)
- 1995 NBCC (Canadian)
- 1996 BOCA (American)
- ASCE 7-95 (American)

Each of these is discussed later in this chapter. In addition user-defined wind loads can be automatically generated in any arbitrary horizontal direction.

Defining Automatic Wind Load Cases

The automatic wind static load cases are defined using the **Define menu > Static Load Cases** command. This command brings up the Define Static Load Case Names dialog box. In this dialog box you specify a name for a load case, a design type, a self-weight multiplier and in some an Auto Lateral Load. See the subsections titled "Design Type" and "Self-Weight Multiplier" in Chapter 27 for additional information on those items.

When you specify the design type for a static load as Wind the Auto Lateral Load box becomes active and you can choose from any of the codes mentioned above. If you do not want the Wind load to be an automatic lateral load then select None in the Auto Lateral Load box.

If you select a code in the Auto Lateral Load box then when you click the **Add New Load** or **Modify Load** buttons a dialog box pops up that allows you to specify the appropriate parameters for that particular code. If you want to modify the parameters for an existing automatic lateral load then highlight the load in the Define Static Load Case Names dialog box and click the **Modify Load** button.

Each automatic static lateral load that you define must be in a separate load case. You can not have two automatic static lateral loads in the same load case. You can, however, add additional user-defined loads to a load case that includes an automatic static lateral load. You must define a separate automatic static load case for each direction of wind load.

Note that the actual forces associated with an automatic static lateral load are not calculated until you run the analysis. Thus you can not view the resultant automatic lateral loads until after you have run an analysis.



Tip:

Note that the actual forces associated with an automatic static lateral load are not calculated until you run the analysis.

Automatic Wind Load Cases

In ETABS automatically calculated wind loads are only applied to rigid diaphragms. A separate load is created for each rigid diaphragm present at a story level. The wind loads calculated at any story level are based on the story level elevation, the story height above and below that level, the assumed exposure width for the rigid diaphragm(s) at that story level and various code-dependent wind coefficients.

If you have a model where you are using floor elements to model the actual in-plane stiffness of the diaphragm and you want to create some automatic wind load cases then you will need to define one or more dummy rigid diaphragms at each story level. You can assign a dummy rigid diaphragm to just one point object at a story level. The point object that is assigned a dummy rigid diaphragm becomes a location where the wind load is applied.

The dialog boxes defining the automatic wind loads are broken into four areas. The first area allows you to define the wind direction and the second area is for defining the exposure height. The third area is for defining wind coefficients and the fourth area is for specifying the exposure width and wind load application point associated with a rigid diaphragm. The data in the direction, exposure height and the exposure width s is common to all of the codes and is described once below. The data in the wind coefficients area is code-dependent and is described separately for each code later in this chapter.

You can also input user-defined wind loads. The format for this is described at the end of this chapter.

Wind Direction

When specifying the wind direction you indicate the direction of the wind by an angle measured in degrees. An angle of 0 degrees means the wind is blowing in the positive global X-direction, that is it is blowing from the negative global X-direction to the positive global X-direction. An angle of 90 degrees means the wind is blowing in the positive global Y-direction. An angle of 180 degrees means the wind is blowing in the negative global X-

direction. An angle of 270 degrees means the wind is blowing in the negative global Y-direction.

You can input any angle for the wind direction. The angle is always measured counterclockwise from the positive global X-axis. A positive angle appears counterclockwise as you look down on the model in the negative global Z-direction.

Wind Exposure Height

The exposure height is based on the story level heights specified for the building. You can view and/or modify these heights using the **Edit menu > Edit Story Data > Edit** command. Three additional items that you input in the dialog box defining the automatic wind loads to complete the definition of the exposure height are a top story, bottom story level and, if desired, a parapet height.

The top story indicates the highest story level to be assumed exposed to wind loading for the purposes of calculating the automatic wind load. In most instances the top story should be the uppermost story level in the building and this is the default value.

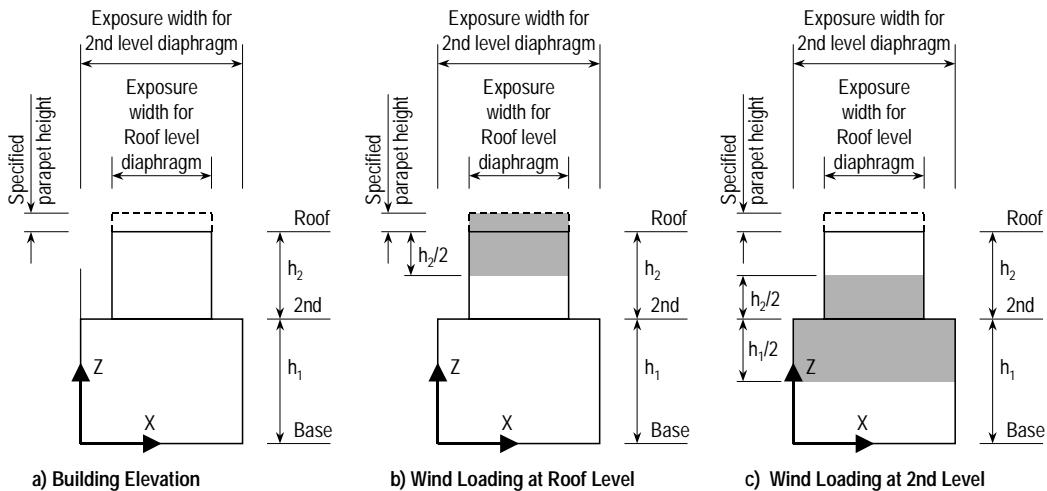
In some instances, for example where penthouses are included in the ETABS model and there is a parapet at the main roof level, it may be more convenient to indicate that the top story level for automatic wind loading is the main roof level. You can then add user-defined loads to the load case to account for the wind loads acting on the penthouse.

Note:

You can specify a parapet height that is used in determining automatic wind loads.

The bottom story level indicates the lowest story level that is exposed to wind loading. When the wind load is calculated the bottom story receives wind load from half of the story height above. It is assumed that all stories above the bottom story are loaded by the wind.

By default the bottom story is assumed to be the base level of the building. In some cases you may want to specify that a higher level is the bottom story for wind loading. One example of this might be if your building has several below-grade levels which should not receive any wind loading.



(Above)

Figure 29-1:
Example extent of
wind loading

The parapet height item allows you to specify the height that a parapet extends above a specified top story level. The wind load that acts on the parapet extension is assumed to act at the specified top story level.

Consider the example shown in Figure 29-1a. The figure shows an elevation of a two-story building with rigid diaphragms at each story level. Assume wind load is to be automatically calculated for the Y-direction. Thus the wind load is acting on the face of the building shown in the elevation.

The shaded area in Figure 29-1b illustrates the extent of the wind load that ETABS assumes is applied to the Roof level. The shaded area in Figure 29-1c illustrates the extent of the wind load that ETABS assumes is applied to the 2nd level.

Wind Exposure Width and Wind Load Application Point

By default, the wind exposure width for a rigid diaphragm is equal to the ETABS calculated width of the diaphragm in a direction perpendicular to the direction of the wind load. You can modify the assumed exposure width if desired when defining the code-specific automatic wind load parameters.

ETABS calculates the maximum width of the rigid diaphragm perpendicular to the direction of the wind loading using the following three-step process.

- Transform the coordinates of all of the point objects that are part of the rigid diaphragm constraint into a system of coordinates that is parallel and perpendicular to the specified direction of wind loading.
- Find the point objects that have the maximum and minimum coordinates perpendicular to the direction of the wind load. For example, if the wind load is in the global X direction then you want to find the point objects with the maximum and minimum global Y coordinates.
- Subtract the minimum perpendicular coordinate from the maximum perpendicular coordinate to obtain the diaphragm width perpendicular to the wind load.

By default, the point where the wind load is applied to a rigid diaphragm is the ETABS-calculated geometric center of the diaphragm. You can modify the assumed wind load application point if desired when defining the code-specific automatic wind load parameters.

1994 UBC Wind Loads

Input Wind Coefficients

Three wind coefficients are input for 1994 UBC wind loads. They are the basic wind speed in miles per hour (mph), the exposure type and the wind importance factor, I_w .

The basic wind speed is described in 1994 UBC Sections 1614 and 1616. A typical range of values for the basic wind speed is 70 to 130 mph.

The exposure types are described in 1994 UBC Sections 1614 and 1617. The exposure type can be B, C or D. No other values are allowed.

The wind importance factor can be found in 1994 UBC Table 16-K. You should input the wind importance factor, I_w , not one of the seismic importance factors, I or I_p . A typical range of values for I_w is 1.00 to 1.15.

Algorithm for 1994 UBC Wind Loads

ETABS automatic wind loads for the 1994 UBC are based on Sections 1614 through 1619 of the 1994 UBC.

The wind loads applied in ETABS are a modified version of the Method 2 (Projected Area Method) as described in Section 1619.3 of the 1994 UBC. ETABS applies horizontal wind loads on the vertical projected area as described in Section 1619.3. ETABS has two modifications to the requirements of Section 1619.3. The first modification is that ETABS does not automatically apply vertical wind loads over the projected horizontal area. If you want to include these vertical wind loads in the load case then you must manually include them yourself.

The other modification is that ETABS applies the method to structures of any height. It does not limit it to structures less than 200 feet high as discussed in 1994 UBC Section 1619.3.

The shape of the horizontal projected area is determined based on the story heights and the input exposure widths for each rigid diaphragm. ETABS uses Equation 29-1 (1994 UBC Equation 18-1) to determine the wind pressure, P , at any point on the surface of the horizontal projected area.

$$P = C_e C_q q_s I_w \quad \text{Eqn. 29-1}$$

where,

C_e = Combined height, exposure and gust factor coefficient as given in 1994 UBC Table 16-G.

C_q = Pressure coefficient for the structure as given in 1994 UBC Table 16-H.

q_s = Wind stagnation pressure at the standard height of 33 feet as given in 1994 UBC Table 16-F.

I_w = Importance factor as input by the user.

ETABS determines the C_e coefficient from 1994 UBC Table 16-G using the input exposure type and the input bottom story. For use in 1994 UBC Table 16-G the elevation of the input bottom story is assumed to be zero (0). ETABS uses linear interpolation to determine the value of the C_e coefficient at heights above 15 feet that are not listed in 1994 UBC Table 16-G.

ETABS determines the C_q coefficient from 1994 UBC Table 16-H. ETABS uses the values for Primary Frames and Systems using Method 2 (the projected area method). Thus for buildings 40 feet or less in height ETABS uses $C_q = 1.3$ and for buildings more than 40 feet in height ETABS uses $C_q = 1.4$.

ETABS determines q_s from Equation 29-2.

$$q_s = 0.00256 V^2 \geq 10 \text{ psf} \quad \text{Eqn. 29-2}$$

where,

q_s = Wind stagnation pressure at the standard height of 33 feet, psf.

V = Basic wind speed as input by the user, mph.

Note the units that are specified for q_s and V . Also note that Equation 29-2 is consistent with 1994 UBC Table 16-F.

ETABS distributes the pressures, P , on the surface of the horizontal projected area to each rigid diaphragm on a tributary area basis as shown in Figure 29-1.

1997 UBC Wind Loads

Input Wind Coefficients

Three wind coefficients are input for 1997 UBC wind loads. They are the basic wind speed in miles per hour (mph), the exposure type and the wind importance factor, I_w .

The basic wind speed is described in 1997 UBC Sections 1616 and 1618. A typical range of values for the basic wind speed is 70 to 130 mph.

The exposure types are described in 1997 UBC Sections 1616 and 1619. The exposure type can be B, C or D. No other values are allowed.

The wind importance factor can be found in 1997 UBC Table 16-K. You should input the wind importance factor, I_w , not one of the seismic importance factors, I or I_p . A typical range of values for I_w is 1.00 to 1.15.

Algorithm for 1997 UBC Wind Loads

ETABS automatic wind loads for the 1997 UBC are based on Sections 1616 through 1621 of the 1997 UBC.

The wind loads applied in ETABS are a modified version of the Method 2 (Projected Area Method) as described in Section 1621.3 of the 1997 UBC. ETABS applies horizontal wind loads on the vertical projected area as described in Section 1621.3. ETABS has two modifications to the requirements of Section 1621.3. The first modification is that ETABS does not automatically apply vertical wind loads over the projected horizontal area. If you want to include these vertical wind loads in the load case then you must manually include them yourself.

The other modification is that ETABS applies the method to structures of any height. It does not limit it to structures less than 200 feet high as discussed in 1997 UBC Section 1621.3.

The shape of the horizontal projected area is determined based on the story heights and the input exposure widths for each rigid diaphragm. ETABS uses Equation 29-3 (1997 UBC Equation 20-1) to determine the wind pressure, P , at any point on the surface of the horizontal projected area.

$$P = C_e C_q q_s I_w \quad \text{Eqn. 29-3}$$

where,

C_e = Combined height, exposure and gust factor coefficient as given in 1997 UBC Table 16-G.

C_q = Pressure coefficient for the structure as given in 1997 UBC Table 16-H.

q_s = Wind stagnation pressure at the standard height of 33 feet as given in 1997 UBC Table 16-F.

I_w = Importance factor as input by the user.

ETABS determines the C_e coefficient from 1997 UBC Table 16-G using the input exposure type and the input bottom story. For use in 1997 UBC Table 16-G the elevation of the input bottom story is assumed to be zero (0). ETABS uses linear interpolation to determine the value of the C_e coefficient at heights above 15 feet that are not listed in 1997 UBC Table 16-G.

ETABS determines the C_q coefficient from 1997 UBC Table 16-H. ETABS uses the values for Primary Frames and Systems using Method 2 (the projected area method). Thus for buildings 40 feet or less in height ETABS uses $C_q = 1.3$ and for buildings more than 40 feet in height ETABS uses $C_q = 1.4$.

ETABS determines q_s from Equation 29-4.

$$q_s = 0.00256 V^2 \geq 10 \text{ psf} \quad \text{Eqn. 29-4}$$

where,

q_s = Wind stagnation pressure at the standard height of 33 feet, psf.

V = Basic wind speed as input by the user, mph.

Note the units specified for q_s and V . Equation 29-4 is consistent with 1997 UBC Table 16-F.

ETABS distributes the pressures, P , on the surface of the horizontal projected area to each rigid diaphragm on a tributary area basis as shown in Figure 29-1.

1996 BOCA Wind Loads

Input Wind Coefficients for 1996 BOCA

Five wind coefficients are input for 1996 BOCA wind loads. They are the basic wind speed in miles per hour (mph), the exposure category and the wind importance factor, I, the gust response factor, G_h and the wall pressure coefficient for the leeward wall, $C_{p\text{-leeward}}$.

29

The basic wind speed is described in 1996 BOCA Section 1609.3. A typical range of values for the basic wind speed is 70 to 130 mph.

The exposure categories are described in 1996 BOCA Section 1609.4. The exposure category can be A, B, C or D. No other values are allowed.

The wind importance factor, I, is described in 1996 BOCA Section 1609.5. A typical range of values for I is 0.90 to 1.23.

The gust response factor, G_h , is discussed in 1996 BOCA Section 1609.7 and in Table 1609.7(5). You can either specify that the gust response factor is to be calculated based on the height (distance) of the specified top story above the specified bottom story and the exposure category per the code using Table 1609.7(5) or you can input your own value. Note that in 1996 BOCA Section 1609.7 the following statement is made about G_h .

The gust response factor for buildings which have a height to least dimension ratio greater than 5 or a fundamental frequency less than one cycle per second (period greater than 1 second) shall be calculated by an approved rational analysis that incorporates the dynamic properties of the main wind force-resisting system.

When you select the Per Code option for the gust response factor ETABS does *not* check the height to least dimension ratio or the building period and it does *not* determine the gust factor using an approved rational analysis incorporating the dynamic properties of the main wind force-resisting system. It is assumed that you

**Note:**

In ETABS input
 $C_{p\text{-leeward}}$ as a
positive num-
ber.

29

will do this yourself, if necessary, and provide a user-defined value for G_h . A typical range of values for G_h is 1.00 to 2.36.

The wall pressure coefficient for the leeward wall, $C_{p\text{-leeward}}$, is determined from 1996 BOCA Table 1609.7(1). **Although C_p for leeward walls is shown as a negative value in 1996 BOCA Table 1609.7(1) in ETABS it should be input as a positive value.** The default value for this is 0.5. You may want to change it from this default value depending on the horizontal dimensions of your building parallel and perpendicular to the direction of the wind. Typical values for $C_{p\text{-leeward}}$ are 0.5, 0.3 and 0.2.

Algorithm for 1996 BOCA Wind Loads

ETABS automatic wind loads for the 1996 BOCA are based on Section 1609 of the 1996 BOCA.

The wind loads applied in ETABS are a modified version of those described in 1996 BOCA Section 1609.7. ETABS applies windward and leeward horizontal wind loads on the vertical projected area of the building as determined from the story heights and the input rigid diaphragm exposure widths. ETABS does not automatically apply vertical wind loads over the projected horizontal area of roof surfaces. If you want to include these vertical wind loads in the load case then you must manually include them yourself.

ETABS uses Equation 29-5 to determine the wind pressure, P , at any point on the surface of the horizontal projected area.

$$P = P_v I [0.8 K_z G_h + K_h G_h C_{p\text{-leeward}}] \quad \text{Eqn. 29-5}$$

where,

P_v = Basic velocity pressure given in 1996 BOCA Table 1609.7(3).

I = Importance factor as input by the user.

K_z = Velocity pressure exposure coefficient at the height of interest as given in 1996 BOCA Table 1609.7(4).

G_h = Gust response factor either as given in 1996 BOCA Table 1609.7(5) or user specified.

K_h = Velocity pressure exposure coefficient, evaluated at the specified top story level, as given in 1996 BOCA Table 1609.7(4).

$C_{p\text{-leeward}}$ = Wall pressure coefficient on the leeward wall as input by the user.

The 0.8 factor in Equation 29-5 represents the wall pressure coefficient for the windward wall.

29

ETABS determines P_v coefficient from Equation 29-6.

$$P_v = 0.00256 V^2 \geq 10 \text{ psf} \quad \text{Eqn. 29-6}$$

where,

P_v = Basic velocity pressure, psf.

V = Basic wind speed as input by the user, mph.

Note the units specified for P_v and V . Equation 29-6 is consistent with 1996 BOCA Table 1609.7(3).

ETABS determines the K_z coefficient from 1996 BOCA Table 1609.7(4) using the input exposure category and the input bottom story. For use in 1996 BOCA Table 1609.7(4) the elevation of the input bottom story is assumed to be zero (0). ETABS uses linear interpolation to determine the value of the K_z coefficient at heights above 15 feet that are not listed in 1996 BOCA Table 1609.7(4).

For discussion of the gust response factor, G_h , refer to the previous section titled "Input Wind Coefficients for 1996 BOCA."

ETABS determines the K_h coefficient from 1996 BOCA Table 1609.7(4) using the input exposure category and the height of the input top story above the input bottom story. ETABS uses linear interpolation to determine the value of the K_h coefficient at heights above 15 feet that are not listed in 1996 BOCA Table 1609.7(4).

For discussion of the wall pressure coefficient on the leeward wall, $C_{p\text{-leeward}}$, refer to the previous section titled "Input Wind Coefficients for 1996 BOCA."

ETABS distributes the pressures, P , on the surface of the horizontal projected area to each rigid diaphragm on a tributary area basis as shown in Figure 29-1.

1995 NBCC Wind Loads

29

Input Wind Coefficients

Two wind coefficients are input for 1995 NBCC wind loads. They are the velocity pressure, q , in kPa and the gust effect factor, C_g .

The velocity pressure, q , can be obtained from 1995 NBCC Appendix C. A typical range of values for the velocity pressure is 0.20 to 0.90 kPa. Any positive value or zero is allowed.

The gust effect factor, C_g , is discussed in 1995 NBCC Sentence 4.1.8.1(6). The default value is 2.0. Any positive value is allowed.

Algorithm for 1995 NBCC Wind Loads

ETABS automatic wind loads for the 1995 NBCC are based on Section 4.1.8.1 of the 1995 NBCC.

ETABS applies windward and leeward horizontal wind loads on the vertical projected area of the building as determined from the story heights and the input rigid diaphragm exposure widths. ETABS does not automatically apply vertical wind loads over the projected horizontal area of roof surfaces. If you want to include these vertical wind loads in the load case then you must manually include them yourself.

ETABS uses Equation 29-7 to determine the wind pressure, p , at any point on the surface of the horizontal projected area.

$$p = q C_g [0.8 C_{e\text{-windward}} + 0.5 C_{e\text{-leeward}}] \quad \text{Eqn. 29-7}$$

where,

q = Velocity pressure as input by the user.

C_g = Gust effect factor as input by the user.

$C_{e\text{-windward}}$ = Exposure factor for the windward wall.

$C_{e\text{-leeward}}$ = Exposure factor for the leeward wall.

The 0.8 factor in Equation 29-7 represents the external pressure coefficient for the windward wall. The 0.5 factor in Equation 29-7 represents the external pressure coefficient for the leeward wall.

ETABS determines $C_{e\text{-windward}}$ from Equation 29-8.

$$C_{e\text{-windward}} = \left(\frac{h}{10} \right)^{1/5} \geq 0.9 \quad \text{Eqn. 29-8}$$

where,

h = Distance from the input bottom story level to the elevation considered, meters.

ETABS determines $C_{e\text{-leeward}}$ from Equation 29-9.

$$C_{e\text{-leeward}} = \left(\frac{h_{\text{middle}}}{10} \right)^{1/5} \geq 0.9 \quad \text{Eqn. 29-9}$$

where,

h_{middle} = one-half of the distance from the input bottom story level to the input top story level, meters.

ETABS distributes the pressures, p , on the surface of the horizontal projected area to each rigid diaphragm on a tributary area basis as shown in Figure 29-1.

ASCE 7-95 Wind Loads

Input Wind Coefficients for ASCE 7-95

Five wind coefficients are input for ASCE 7-95 wind loads. They are the basic wind speed in miles per hour (mph), the exposure category, the wind importance factor, I, the wall pressure coefficient for the leeward wall, $C_{p\text{-leeward}}$ and the topographic factor, K_{zt} .

The basic wind speed is described in ASCE 7-95 Section 6.5.2. A typical range of values for the basic wind speed is 85 to 150 mph.

The exposure categories are described in ASCE 7-95 Section 6.5.3. The exposure category can be A, B, C or D. No other values are allowed.

The wind importance factor, I, is described in ASCE 7-95 Table 6-2. Note that the building and structure classification categories in are defined in ASCE 7-95 Table 1-1. A typical range of values for I is 0.87 to 1.15.

Note:

In ETABS input
 $C_{p\text{-leeward}}$ as a
 positive number.

The wall pressure coefficient for the leeward wall, $C_{p\text{-leeward}}$, is determined from ASCE 7-95 Figure 6-3. **Although C_p for leeward walls is shown as a negative value in ASCE 7-95 Figure 6-3 in ETABS it should be input as a positive value.** The default value for this is 0.5. You may want to change it from this default value depending on the horizontal dimensions of your building parallel and perpendicular to the direction of the wind. Typical values for $C_{p\text{-leeward}}$ are 0.5, 0.3 and 0.2.

The topographic factor K_{zt} is discussed in ASCE 7-95 Section 6.5.5. The default value for K_{zt} is 1.0. K_{zt} can not be less than 1.0.

Algorithm for ASCE 7-95 Wind Loads

ETABS automatic wind loads for the ASCE 7-95 are based on Sections 6.4 through 6.6 of ASCE 7-95.

The wind loads applied in ETABS are a modified version of those described in ASCE 7-95 Sections 6.4 through 6.6. ETABS

applies windward and leeward horizontal wind loads on the vertical projected area of the building as determined from the story heights and the input rigid diaphragm exposure widths. ETABS does not automatically apply vertical wind loads over the projected horizontal area of roof surfaces. If you want to include these vertical wind loads in the load case then you must manually include them yourself.

ETABS uses Equation 29-10 (ASCE 7-95 Equation 6-1) to determine the velocity pressure, q_z , at any height z on the surface of the horizontal projected area in pounds per square foot (psf).

$$q_z = 0.00256 K_z K_{zt} V^2 I$$

Eqn. 29-10

where,

K_z = The velocity pressure exposure coefficient. See Equations 29-11a and 29-11b.

K_{zt} = Topographic factor as input by the user.

V = Basic wind speed in miles per hour (mph) as input by the user.

I = Importance factor as input by the user.

The velocity pressure exposure coefficient, K_z , is obtained using Equations 29-11a and 29-11b (Equations C3a and C3b in ASCE 7-95 Commentary Section 6.5.1).

$$K_z = 2.01 \left(\frac{z}{z_g} \right)^{2/\alpha} \quad \text{for } 15 \text{ feet} \leq z \leq z_g \quad \text{Eqn. 29-11a}$$

$$K_z = 2.01 \left(\frac{15}{z_g} \right)^{2/\alpha} \quad \text{for } z < 15 \text{ feet} \quad \text{Eqn. 29-11b}$$

where,

z = Distance (height) from input bottom story to point considered.

Table 29-1:
 α and z_g factors for
use in Equations 29-
11a and 29-11b

Exposure Category	α	z_g (feet)
A	5.0	1500
B	7.0	1200
C	9.5	900
D	11.5	700

ETABS uses Equation 29-12 to determine the wind pressure, p , at any point on the surface of the horizontal projected area. Equation 29-12 is based on ASCE 7-95 Table 6-1. In particular it is based on the row titled "Main wind force-resisting systems" under the heading titled "Buildings of all heights."

$$p = 0.8 q G + q_h G C_{p\text{-leeward}} \quad \text{Eqn. 29-12}$$

where,

q = velocity pressure, q_z , at any height z on the surface of the horizontal projected area calculated using Equation 29-10.

G = Gust effect factor. As described in ASCE 7-95 Section 6.6.1, this is taken by ETABS as 0.80 for exposure categories A and B and 0.85 for exposure categories C and D.

q_h = Velocity pressure at the top story height on the surface of the horizontal projected area calculated using Equation 29-10.

$C_{p\text{-leeward}}$ = Wall pressure coefficient on the leeward wall as input by the user.

The 0.8 factor in Equation 29-12 represents the wall pressure coefficient for the windward wall as specified in ASCE 7-95 Figure 6-3.

For discussion of the wall pressure coefficient on the leeward wall, $C_{p\text{-leeward}}$, refer to the previous section titled "Input Wind Coefficients for ASCE 7-95."

ETABS distributes the pressures, P , on the surface of the horizontal projected area to each rigid diaphragm on a tributary area basis as shown in Figure 29-1.

User-Defined Wind Loads

For user defined loads you define the direction of the wind loading, the wind load force applied to each rigid diaphragm at each story level and the location of the wind load force.

Based on this information ETABS automatically creates a point object at the location of the applied load and applies the wind load to the point. Note that the point object loads are always specified in global coordinates. Thus, if the wind load direction is not parallel to one of the global axes then ETABS automatically breaks the wind load up into its proper components in the global X and Y directions.

Automatic Meshing of Area and Line Objects

General



Note:

See the section titled “The ETABS Analysis Model” in Chapter 5 for more information on the analysis model.

This chapter discusses how ETABS automatically meshes (divides) floor-type (horizontal) area objects and line objects. ETABS performs two separate types of automatic meshing. One type that is fully automatic meshes objects into the analysis model only (not the object-based model) on an as-needed basis with no input from you. The other type, that is automatic but must be manually activated by you, meshes objects into your object-based model.

When objects are fully automatically meshed directly into the analysis model the meshing takes place when you start the analysis. In this case the meshing all takes place inside the program and is essentially invisible to the user. The only objects that ETABS can fully automatically mesh into the analysis model are line objects with frame section properties and floor-type area objects with membrane properties only (not plate bending or shell behavior).



Important Notes about Automatic Meshing of Area and Line Objects

When an analysis is started ETABS automatically meshes (divides) all line objects with frame section properties into the analysis model. ETABS also meshes all floor-type (horizontal) area objects that either have deck section properties or have slab section properties with membrane behavior only into the analysis model. You do not have to do anything to make this happen. It simply occurs when you run the analysis. Any other type of area object must be manually meshed by you in your object-based ETABS model, prior to running the analysis, using the methods described in Chapter 31.

The automatic meshing of area and line objects by ETABS into the analysis model has no affect on your object-based ETABS model. In this case, the automatic meshing only affects the analysis model, which is internal to ETABS. After running the analysis your object-based model still has the same number of objects in it as it did before the analysis was run.

It is possible to automatically mesh the objects in your object-based model exactly as they would be meshed into the analysis model. To mesh line objects in this way, select all of the line objects with frame section properties, click the **Edit menu > Divide Lines** command, and select the Break at Intersections with Selected Lines and Points option. To mesh area objects in this way select all floor-type area objects, click the **Edit menu > Mesh Areas** command, and select the Auto Mesh Floor Areas option.

Note that while the automatic meshing of floor-type area objects into the analysis model is limited to those with membrane behavior only, you can automatically mesh any floor-type area object in your object-based ETABS model using the Auto Mesh Floor Areas option of the **Edit menu > Mesh Areas** command.

In some cases (probably unusual cases) you may not want ETABS to automatically mesh some line and/or area objects into the analysis model. In these cases you can select the line objects that you do not want automatically meshed and use the **Assign menu > Frame/Line > Automatic Frame Mesh/No Mesh** to tell ETABS not to automatically mesh them. Similarly, you can select the area objects you don't want automatically meshed and use the **Assign menu > Shell/Area > Automatic Membrane Floor Mesh/No Mesh** to tell ETABS not to automatically mesh them.

Even if your object-based ETABS model is not automatically meshed you can still at any time view the automatic mesh for the area objects that will be used in the analysis model. To do this click the **View menu > Set Building View Options** command (or click the **Set Building View Options** button, , on the main (top) toolbar), check the Auto Floor Mesh item in the Other Special Items area of the Set Building View Options dialog box and click the **OK** button.

**Tip:**

The Auto Mesh Floor Areas option of the Edit menu > Mesh Areas command allows you to mesh any floor-type area object in your object-based model (not the analysis model) as described in this chapter.

For example, suppose you model a slab over metal deck with just one area object in your object-based ETABS model. Then when ETABS internally creates the analysis model it automatically meshes that one object in the object-based model into many elements in the analysis model. This all happens internal to the program. After the analysis is run your object-based model still has only one area object representing the slab over deck, not the many objects that ETABS used internally to run the analysis.

ETABS can automatically mesh *any* line object and *any* floor-type area object into your *object-based ETABS model*. If you select all objects in your object-based model and use the Break at Intersections with Selected Lines and Points option of the **Edit menu > Divide Lines** command, then ETABS automatically divides all line objects in your object-based model. If you select all floor-type area objects and use the Auto Mesh Floor Areas option of the **Edit menu > Mesh Areas** command, then ETABS automatically meshes all of the floor-type area objects in your object-based model as described in this chapter. The meshing takes place immediately and the selected area objects in your object-based model are broken into smaller objects. You can use the **Edit menu > Undo** command or the associated **Undo** toolbar button to reverse the meshing if necessary.

Automatic Meshing of Line Objects

This section discusses how ETABS fully automatically meshes (divides) line objects into the analysis model. Only line objects with frame section assignments are automatically meshed into the analysis model by ETABS.

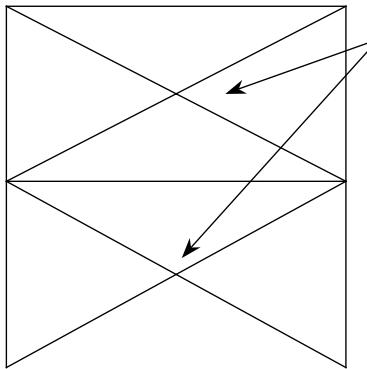
**Note:**

Frame elements are meshed at locations where other frame elements attach to or cross them and at locations where point objects lie on them.

ETABS divides the frame elements at locations where they intersect (cross) other frame elements and at locations where point objects lie on them. These point objects may be at the ends of other frame members, ends of link members, corners of area objects with structural (wall, slab or deck) properties, or they may simply be free-floating point objects.

Line objects assigned link properties are never automatically meshed into the analysis model by ETABS. If a line object is assigned both a frame section property and a link property then in the analysis model ETABS treats it as two elements and meshes

Figure 30-1:
Example of automatic meshing of frame elements



Elevation

In the ETABS analysis model the braces are connected where they cross unless you indicate that they are not connected by selecting the braces and clicking the **Assign menu > Frame/Line > No Automatic Meshing** command

30

(divides) the element with the frame section assignment. ETABS does not mesh the element with the link property.

Consider the braced frame shown in Figure 30-1. When the analysis is run ETABS automatically meshes (divides) the braces at the point where they cross in the analysis model. Thus in your object-based ETABS model each set of braces at a story level is modeled with two objects, but in the analysis model that is internally created by ETABS each set of braces at a story level is modeled with four elements since the braces are broken at their intersection point.

When ETABS divides frame sections in this way it assumes full continuity between the divided pieces. No end releases are introduced. If you want end releases you should manually divide the objects yourself and assign the releases. Use the **Edit menu > Divide Lines** command to manually mesh the braces. See the section titled "Dividing Lines" in Chapter 9 for more information.

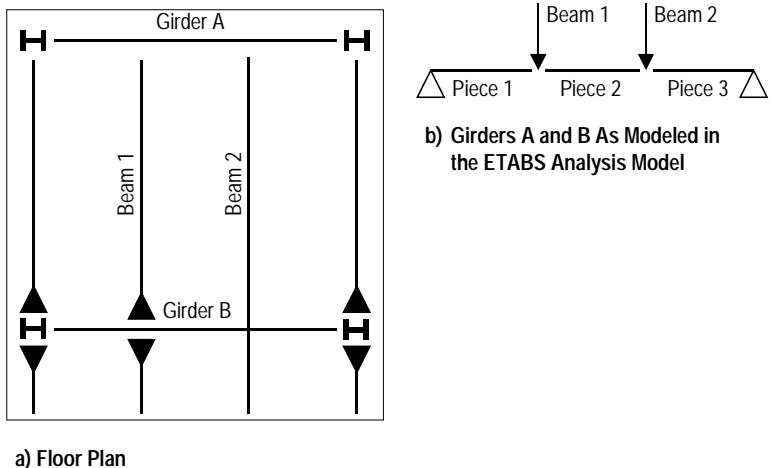
If you do *not* want the braces modeled as divided and attached to each other at their intersection point in the analysis model, then select the braces and click the **Assign menu > Frame/Line > Automatic Frame Mesh/No Mesh** command. See the subsection titled "Automatic Frame Mesh/No Mesh Assignments to Line Objects" in Chapter 14 for more information.



Tip:

If you do not want a frame element to be automatically meshed in the analysis model then select it and click the **Assign menu > Frame/Line > Automatic Frame Mesh/No Mesh** command.

Figure 30-2:
Example showing
how beams are
automatically di-
vided (meshed)
where they support
other beams for the
ETABS analysis
model



a) Floor Plan

b) Girders A and B As Modeled in the ETABS Analysis Model

As a second example consider the floor system shown in Figure 30-2a. The beams labeled Beam 1 and Beam 2 are both cantilever beams with full continuity through Girder B. You can model these beams in your object-based ETABS model either with two line objects as shown for Beam 1 or with one line object as shown for Beam 2. ETABS treats both of these beams exactly the same way in the analysis model. In the analysis model Beam 2 is broken where it intersects Girder B so that there is a connection to Girder B. In general, we recommend that you model cantilevers as shown for Beam 1.

Note that if you use the **Assign menu > Frame/Line > Automatic Frame Mesh/No Mesh** command to specify that Beam 2 in Figure 30-2a is not to be automatically meshed then there will be no connection between Girder B and Beam 2.

Figure 30-2b shows that in the analysis model both Girders A and B are broken into three pieces (elements) even though they remain a single object in your object-based ETABS model. They are broken at the locations where Beams 1 and 2 attach to (or intersect) them.

Automatic Meshing of Area Objects

General

All floor-type (horizontal) area objects that have either deck section properties or slab section properties with membrane behavior only are fully automatically meshed into the analysis model by ETABS. This automatic meshing only affects the analysis model. It does not change your object-based ETABS model in any way.

No other types of area objects are fully automatically meshed into the analysis model by ETABS. You must manually mesh all other types of area objects in your object-based model by selecting them and using the manual meshing options available from the **Edit menu > Mesh Areas** command. See Chapter 31 for documentation of these options.

You can use the Auto Mesh Floor Areas option of the **Edit menu > Mesh Areas** command to automatically mesh any floor-type area object *into your object-based model* as described in this chapter. Note that this command is not limited to area objects that have either deck section properties or slab section properties with membrane behavior only. It applies to any floor-type area object.

For floors that ETABS automatically meshes either into the analysis model or into the object-based model *we strongly recommend* that at a minimum you include all the beams that connect to columns in your object-based ETABS model. If you do not actually have beams connecting to the columns then *we strongly recommend* that you include Null-type line objects connecting the columns. These line objects do not necessarily need to have any assignments. Having the columns connected by line objects in this manner makes the automatic meshing of the floor much easier for ETABS and more predictable for you.

How ETABS Automatically Meshes Floors



Note:

For the purposes of automatically meshing a floor ETABS treats a wall like two columns and a beam where the columns are located at the ends of the wall and the beam connects the columns.

When ETABS automatically meshes a floor-type area object it breaks the object up into four-sided (quadrilateral) elements. Typically, each side of each element of the mesh has a beam or wall running along it. This beam may be real or it may be an imaginary beam internal to ETABS. These imaginary internal beams are used by ETABS for the purposes of meshing the floor and distributing any load on the floor. See Chapter 32 for discussion of transfer (distribution) of loads in ETABS.

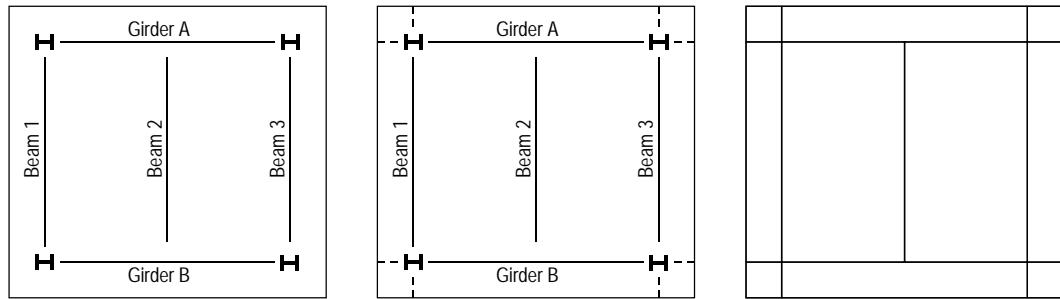
When meshing a floor, ETABS treats walls as if they were two columns (one located at each end of the wall) connected by a beam.

The automatic floor mesh is created by ETABS using the following sequence of steps:

1. Each column is assumed to have four beams connecting to it. If four beams do not connect to a column then ETABS considers other line objects (without frame section properties) that connect to the columns as imaginary beams. If this still does not provide enough beams connecting to the columns then ETABS internally creates its own imaginary beams to satisfy the requirement. Note that if necessary imaginary beams are extended from columns near the edge of the floor to the edge of the floor. An example presented later in Figure 30-3 discusses this.
2. The floor is broken up at all walls and all real and imaginary beams to create a mesh of four-sided elements.

Viewing the Automatic Floor Mesh

If you use the Auto Mesh Floor Areas option of the **Edit menu > Mesh Areas** command to mesh the floor in your object-based model then the floor mesh is there for you to see any time your floor-type area objects are visible.



a) Floor Plan

b) ETABS Imaginary Beams Shown Dashed

c) ETABS Automatic Floor Meshing

(Above)

Figure 30-3:
Example of ETABS automatically generated mesh for floor-type area objects

30

If you instead have a membrane floor and decided to allow ETABS to fully automatically mesh it into the analysis model, then you will not typically see the floor mesh that ETABS uses. In this case, if you want to see the floor mesh, then click the **View menu > Set Building View Options** command (or click the **Set Building View Options** button, , on the main (top) toolbar) to open the Set Building View Options dialog box. In this dialog box check the Auto Floor Mesh item in the Other Special Items area and then click the **OK** button.

In some cases you may not want ETABS to automatically mesh a membrane floor object into the analysis model. In such cases you can click the **Assign menu > Shell/Area > Automatic Membrane Floor Mesh/No Mesh** command and specify that ETABS is not to automatically mesh the area object when it creates the analysis model.

Examples of Automatic Floor Meshing

Consider the example floor plan shown in Figure 30-3a. The figure shows a single area object that models a floor. Horizontal line objects model the beams and vertical line objects model the columns. Figure 30-3b shows how ETABS extends imaginary beams from the columns to the edge of the floor. Notice that the imaginary beams are extended parallel to the beams (or line objects) already framing into the columns.

Figure 30-4:
Second example of
ETABS automatically generated mesh
for floor-type area
objects

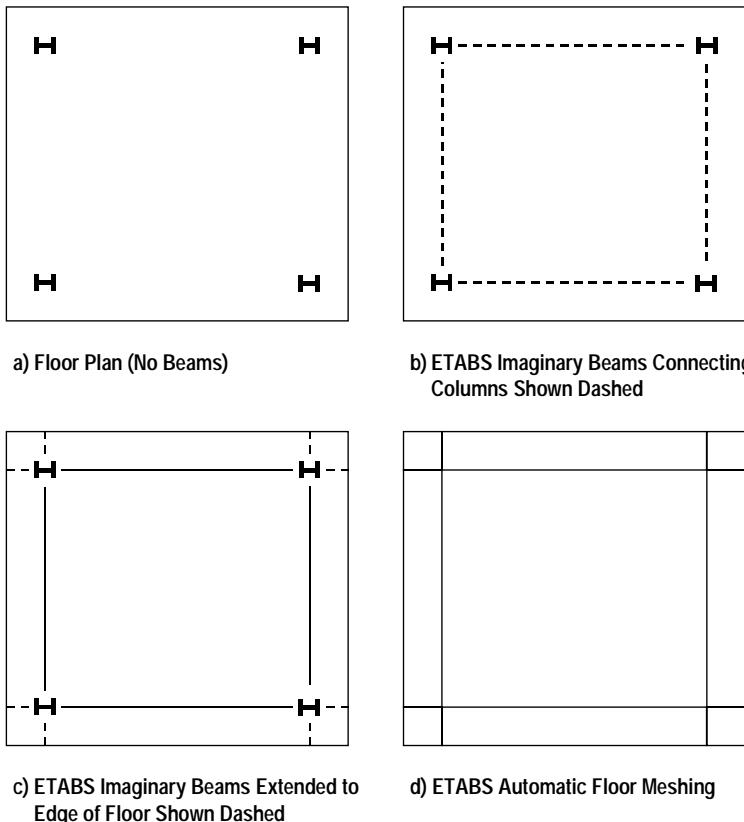


Figure 30-3c shows the floor mesh used in the analysis model that results from meshing the floor at all real and imaginary beams. Also note that the girders labeled Girder A and Girder B are divided (internally to ETABS) into two elements at the point where Beam 2 attaches to them. Note that if Beam 3 in Figure 30-3 is replaced with a wall the process described is exactly the same because for the purposes of automatically meshing a floor ETABS treats a wall like two columns located at the ends of the wall and a beam connecting the columns.

As a second example consider the floor plan shown in Figure 30-4a. In this case there is a single area object modeling a floor and four vertical line objects modeling the columns. No beams (or other null-type line objects connecting the columns) are included in the model. Figure 30-4b shows how ETABS might create imaginary beams connecting the columns. (The next example will discuss how this is done). Figure 30-4c shows how addi-

**Tip:**

There are two ways that you can have some control over automatic meshing. The first is to define beams (or null-type line objects connecting the columns) since the floor mesh breaks at these lines. A second way you can control automatic meshing is to indicate that certain objects are not to be automatically meshed using commands available on the Assign menu.

30

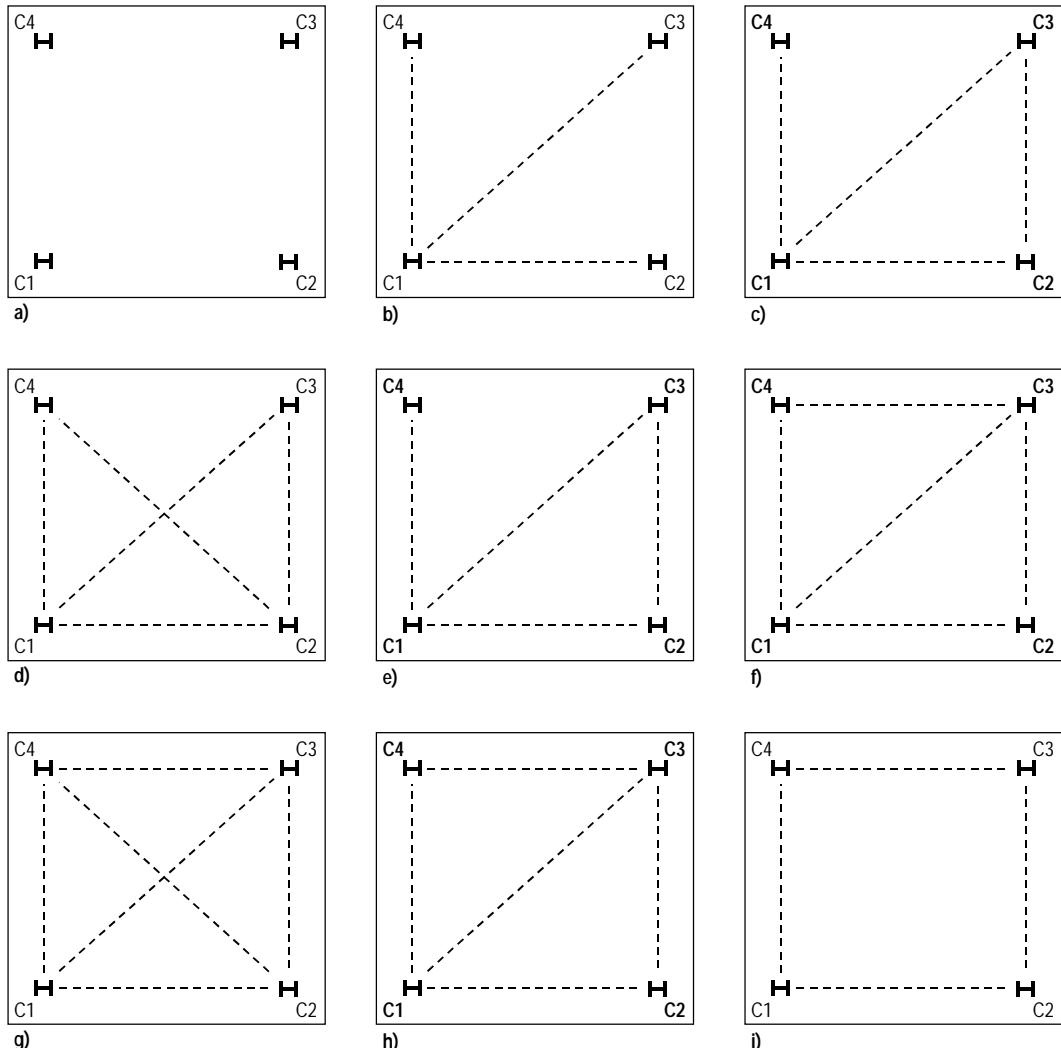
tional imaginary beams are extended from the columns parallel to the imaginary beams already defined in Figure 30-4b to the edge of the floor. Finally, Figure 30-4d shows the floor mesh that results from meshing the floor at all real and imaginary beams. The resulting mesh is similar to that obtained in Figure 30-3c.

Important Recommendation: In general, for floors that are automatically meshed by ETABS we recommend that you model beams (or at least null-type line objects) connecting columns as shown in Figure 30-3a rather than no beams (or line objects) connecting the columns as shown in Figure 30-4a. If you are modeling a flat slab we recommend that you include null-type line objects connecting the columns in your model. Including the lines connecting the columns in your model makes the automatic meshing for the analysis model cleaner, faster, and perhaps most important, more predictable for you.

Let's examine how ETABS might create the distribution of imaginary beams shown in Figure 30-4b. Refer to Figures 30-5a through 30-5i. They show the sequence of events taking place internally in ETABS that leads up to Figure 30-4b. Figure 30-5a shows the four columns and labels them C1, C2, C3 and C4 for reference. Figure 30-5b shows how imaginary beams are connected from column C1 to all other columns. The imaginary beams are shown dashed. Figure 30-5c shows an imaginary beam connecting columns C2 and C3. Figure 30-5d shows an imaginary beam connecting columns C2 and C4.

Note that in Figure 30-5d the imaginary beams from C1 to C3 and C2 to C4 cross. When beams cross like this one of them is eliminated. Typically the longer beam is eliminated and the shorter one is kept. In a situation like this example where both beams are the same length the one created last is eliminated. Thus the beam from C2 to C4 is eliminated and the beam from C1 to C3 is kept. This is illustrated in Figure 30-5e.

Figure 30-5f shows an imaginary beam added from column C3 to C4. Thus column C3 is connected to all columns. Moving on to column C4, Figure 30-5g shows that column C4 is connected to C2 with an imaginary beam. Again this imaginary beam crosses the one from column C1 to C3. Since both of the cross-



30

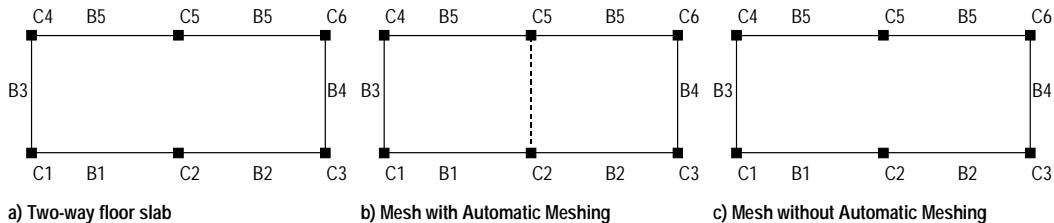
(Above)

Figure 30-5:

Illustration of how ETABS creates the distribution of imaginary beams shown in Figure 30-4b

ing beams are the same length we eliminate that last created one, that is the one from C4 to C2. This is shown in Figure 30-5h.

At this point we have gone through all of the columns and now have a series of triangles. ETABS now combines the triangles to make quadrilaterals that are as close to squares and rectangles as possible. This is shown in Figure 30-5i which is the same as Figure 30-4b.



(Above)

Figure 30-6:
Example of no automatic meshing

30

If you try extending this process to a slightly more complicated problem, for example, add another bay or two, or, offset one of the columns, you will see that the problem quickly becomes quite complex and that it is difficult to predict exactly how ETABS will end up developing the membrane floor mesh. Also you may not even be sure at which column ETABS started the process. These are the reasons that we generally recommend that you model beams and/or null-type line objects between your columns when ETABS is going to automatically mesh a membrane floor. The beams (or line objects) make your automatic membrane floor mesh much more predictable.

As a final example consider the floor shown in Figure 30-6a which is a two-way slab supported by 6 columns. Assume that beams are specified around the perimeter of the floor but that no beam is specified between columns C2 and C5.

Figure 30-6b shows how the slab is meshed when ETABS automatically meshes it. Note that an imaginary beam (shown dashed) is created between column C2 and C5 and that the area object is divided along this imaginary beam.

Now suppose that you select the area object and click the **Assign menu > Shell/Area > Automatic Membrane Floor Mesh/No Mesh** command to indicate that you do not want the floor to be automatically meshed by ETABS. Figure 30-6c shows how the slab is meshed (or perhaps more accurately, not meshed) in this case. Note that an imaginary beam is *not* created between column C2 and C5 and thus the area object is *not* divided.

As a last comment about the example in Figure 30-6 suppose you select the area object, click the **Edit menu > Mesh Areas** command, select the Auto Mesh Floor Areas option and click the **OK** button. In this case the area object is meshed *in your object-based ETABS model* as shown in Figure 30-6b.



Tip:

Including beams and/or null-type line objects between all columns in your model makes automatic floor meshing more predictable.

Manual Meshing of Area Objects

General



Tip:

You should wait until just before you run your first analysis to manually mesh your area objects. This way you can take advantage of having to work with fewer area objects as you make assignments to your model.

This chapter discusses the manual meshing methods available in ETABS for area objects.

In general you must manually mesh the area objects in your object-based ETABS model into a finite element mesh before you run an analysis. There are two exceptions to this. They are floor-type (horizontal) area objects with deck section properties and floor-type (horizontal) area objects with slab section properties that have membrane behavior only (not plate bending or shell behavior). ETABS automatically meshes the area objects into the analysis model for these two exceptions. The objects remain unmeshed in your object-based ETABS model. See Chapter 30 for discussion of this automatic meshing.

We recommend that you work with larger area objects as you create your model and wait until just before you run the analysis to mesh the area objects (assuming they are not membrane floors that ETABS automatically meshes into the analysis model). As-

suming that you do not initially draw your area objects in a finite element mesh (we do not recommend that you do this) there are an assortment of tools available in ETABS to assist you with manual meshing of area objects. These tools are available through the **Edit menu > Mesh Areas** command. There are three basic types of manual meshing tools. They are:

- **Automatic floor meshing:** This special manual meshing tool applies only to floor-type (horizontal) area objects. For selected floor-type area objects the Auto Mesh Floor Areas option meshes the object in your object-based ETABS model as described in Chapter 30. This meshing command is applicable to any floor-type area object, not just those with membrane-behavior-only properties.
- **Cookie cut meshing tools:** These meshing tools are typically used to mesh single area objects, with any number of sides, that may encompass a large portion of your model into a series of smaller objects. The idea is to use these options to reduce the larger objects down to three and four-sided area objects (preferably four-sided). These tools are discussed in the section titled "Cookie Cut Meshing Tools" later in this chapter.
- **Meshing tools for quadrilaterals (quads) and triangles:** These meshing tools are used to mesh three and four-sided area objects into smaller elements. These tools are discussed in the section titled "Meshing Tools for Quadrilaterals and Triangles" later in this chapter.



Tip:

Typically to get good plate bending behavior you want to have four or more elements between each support point.

Typically we envision that you will use the automatic floor meshing options or one of the cookie cutter options to mesh larger area objects into smaller three and four-sided objects. Then we envision you using the meshing tools for quads and triangles on an as-needed basis to further refine your mesh.

Important Tip: If you are meshing area objects that have out-of-plane bending properties and you want to adequately capture the out-of-plane bending behavior, then you should have four or more meshed objects between each support point. This means that you would like to have at least sixteen elements in a single square or rectangular bay.

Cookie Cut Meshing Tools

There are two cookie cut meshing options available. The term cookie cut is used because these tools allow you to define what is effectively a cookie cutter that meshes your area object.

Both of the cookie cut meshing options mesh a large area object that encloses other objects (either line objects or point objects, depending on the option chosen). Typically, but not necessarily, you mesh one area object at a time using these options.

The following two subsections describe the cookie cut meshing options.

Cookie Cut at Selected Line Objects

This subsection discusses the Cookie Cut at Selected Line Objects option for the **Edit menu > Mesh Areas** command. This option works for area objects with any number of edges, not just quadrilaterals and triangles.

The idea with this option is to mesh a large area object into smaller objects by selecting the large object along with other line objects that define how the selected area object is to be meshed. The selected line objects essentially define a cookie cutter that ETABS uses to mesh the selected area object.

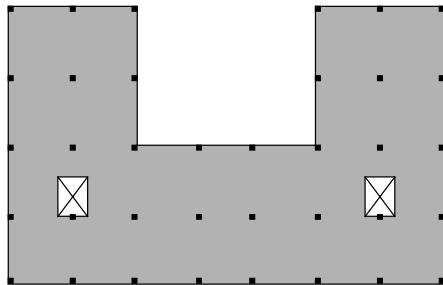
Consider the floor slab shown in Figure 31-1a. To mesh this slab using the Cookie Cut at Selected Line Objects option you should draw line objects to help define the desired mesh. These line objects are illustrated in Figure 31-1b. Note the following about these line objects.

- You can use any line object to aid in the meshing. The line objects shown in this example are Null-type line objects with no assignments of any type. Such line objects are useful as construction lines for meshing. (Incidentally, construction lines like these can also be used for snapping to while drawing or editing objects, and for guides while extending or trimming lines).

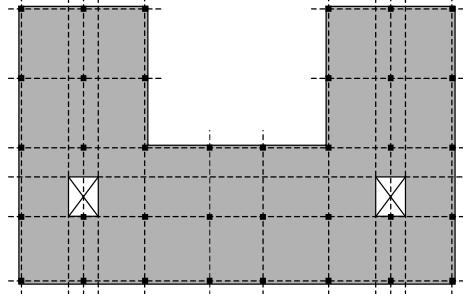
Note:

When cookie cutter meshing based on selected line objects you can select any type of line object as a meshing line. This includes column, beam, brace and null-type line objects.

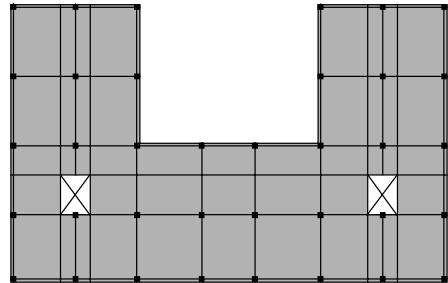




a) Floor slab



b) Mesh lines



c) Completed mesh

- It works best if you extend your line objects used for meshing a little beyond the edges of the area object you are meshing. This is illustrated by the mesh lines shown dashed in Figure 31-1b.

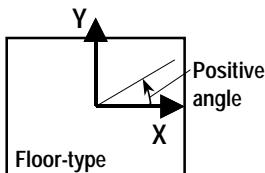
In this example you should select the large area object that defines the slab and all of the mesh lines (shown dashed in Figure 31-1b) before executing the mesh command. The final result of the meshing is shown in Figure 31-1c.

Cookie Cut at Selected Points at X Degrees

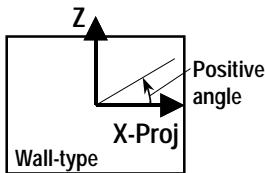
This subsection discusses the Cookie Cut at Selected Points at X Degrees option for the **Edit menu > Mesh Areas** command. This command works for area objects with any number of edges.

The idea with this option is to mesh a large area object into smaller objects by selecting the large object along with other point objects. You then specify an angle. ETABS internally constructs two perpendicular lines through each selected point. One line is at the specified angle and the other line is at the specified angle plus 90 degrees. These internally constructed lines essentially define a cookie cutter that ETABS uses to mesh the selected area object.

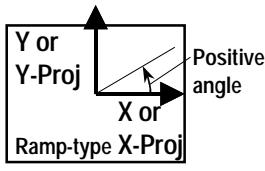
The specified angle is defined as follows for the various types of area objects:



- **Floor-type (horizontal) area objects:** The angle is measured from the positive global X-axis. Positive angles appear counterclockwise when you are looking down on them from above. See the sketch to the left.



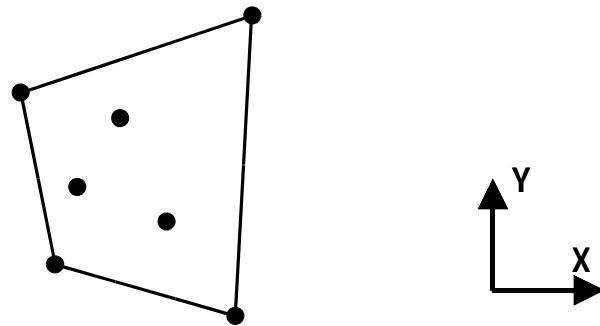
- **Wall-type (vertical) area objects:** The angle is measured from the horizontal projection of the positive global X-axis onto the plane of the wall. If the wall is parallel to the global Y-axis then the angle is measured from the positive global Y-axis. Positive angles appear counterclockwise when you are looking straight at the wall and the positive projection of the X-axis (or the positive Y-axis if the wall is parallel to the global Y-axis) points to your right. See the sketch to the left.



- **Ramp-type (not horizontal and not vertical) area objects:** The angle is measured from the vertical projection of the positive global X-axis onto the plane of the ramp. Positive angles appear counterclockwise when you are looking down on them from above. See the sketch to the left. See the sketch to the left.

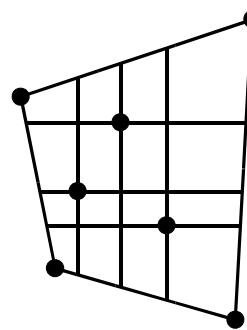
Consider the floor-type (horizontal) area object shown in Figure 31-2a. Also shown in the figure are three selected point objects used to mesh the slab employing the Cookie Cut at Selected Points at X Degrees option.

Figure 31-2:
Example of meshing
using the Cookie Cut
at Selected Points at
X Degrees option

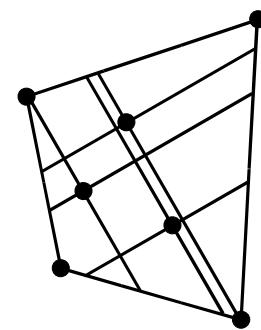


a) Area object and selected points

b) Orientation of the global axes



c) Resulting mesh if specified angle is 0 degrees



d) Resulting mesh if specified angle is 30 degrees

Figure 31-2c shows the resulting mesh when the specified angle is 0 degrees. Note that the meshing lines are parallel to the X-axis and 90 degrees from the X-axis.

Figure 31-2d shows the resulting mesh when the specified angle is 30 degrees. Note that the meshing lines are 30 degrees from the X-axis and 120 degrees from the X-axis.

Meshing Tools for Quadrilaterals and Triangles

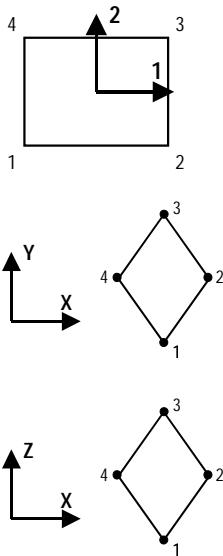
Background Information

The following two subsections provide useful background information about three and four-sided area objects that can help you when meshing them.

Four-sided Area Objects

Four-sided area objects (quadrilaterals, or quads for short) have their corner points labeled 1, 2, 3 and 4. You can right click on any four-sided area object and look in the Area Information dialog box to see this. To use the meshing tools for individual four-sided area objects most effectively it is useful to understand how ETABS decides which corner points of an area object are points 1, 2, 3 and 4.

The following rules are used to define the corner points of the four-sided area objects:



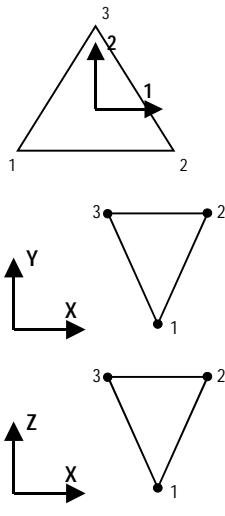
- The corner points labeled 1, 2, 3 and 4 always appear in a counterclockwise order when you look at a four-sided area object with its local 3 axis pointing toward you. See the sketch to the left, which per the right hand rule has the local 3 axis pointing toward you.
- For floor-type (horizontal) area objects the 3 and 4 points are the ones with the largest global Y coordinates. The sketch to the left shows how ETABS handles the special case where points 2 and 4 have the same global Y coordinate.
- For wall-type (vertical) and ramp-type (not vertical or horizontal) area objects the 3 and 4 points are the ones with the largest global Z coordinates. The sketch to the left shows how ETABS handles the special case where points 2 and 4 have the same global Z coordinate.

If at any time you are unsure which corner point is which in a four-sided area object you can always right click on the area object to find out.

Three-sided Area Objects

Three-sided area objects have their corner points labeled 1, 2, and 3. You can right click on any three-sided area object and look in the Area Information dialog box to see this. To use the meshing tools for individual three-sided area objects most effectively it is useful to understand how ETABS decides which corner points of an area object are points 1, 2 and 3.

The following rules are used to define the corner points of the three-sided area objects:



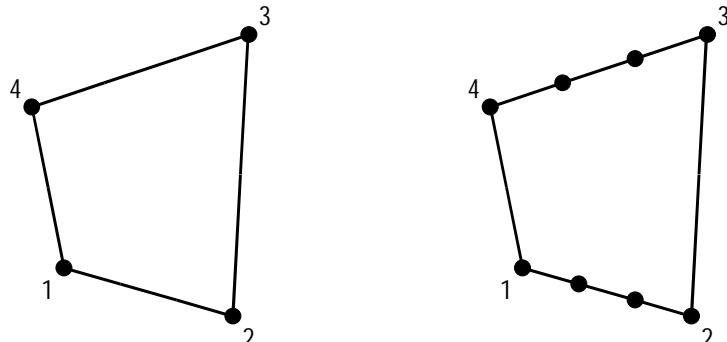
- The corner points labeled 1, 2, and 3 always appear in a counterclockwise order when you look at a three-sided area object with its local 3 axis pointing toward you. See the sketch to the left, which per the right hand rule has the local 3 axis pointing toward you.
- For floor-type (horizontal) area objects the 3 point is the one with the largest global Y coordinate. The sketch to the left shows how ETABS handles the special case where points 2 and 3 have the same global Y coordinate.
- For wall-type (vertical) and ramp-type (not vertical or horizontal) area objects the 3 point is the one with the largest global Z coordinate. The sketch to the left shows how ETABS handles the special case where points 2 and 3 have the same global Z coordinate.

Note the corner points for three-sided area object are defined in a manner consistent with those of four-sided area objects. If at any time you are unsure which corner point is which in a three-sided area object you can always right click on the area object to find out.

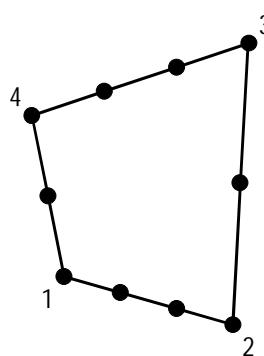
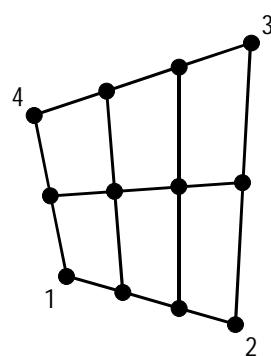
Mesh Quadrilaterals and Triangles into N by M Areas

This subsection discusses the Mesh Quads/Triangles into N by M Areas option for the **Edit menu > Mesh Areas** command. This command works for three and four-sided area objects. It does not work for area objects with more than four sides.

Figure 31-3:
Example of meshing
a four-sided area
object into n by m (3
by 2 in this case)
area objects



a) Quadrilateral Element

b) Divide edges 1-2 and 3-4
into n equal piecesc) Divide edges 2-3 and 4-1
into m equal pieces

d) Complete meshing

31

**Tip:**

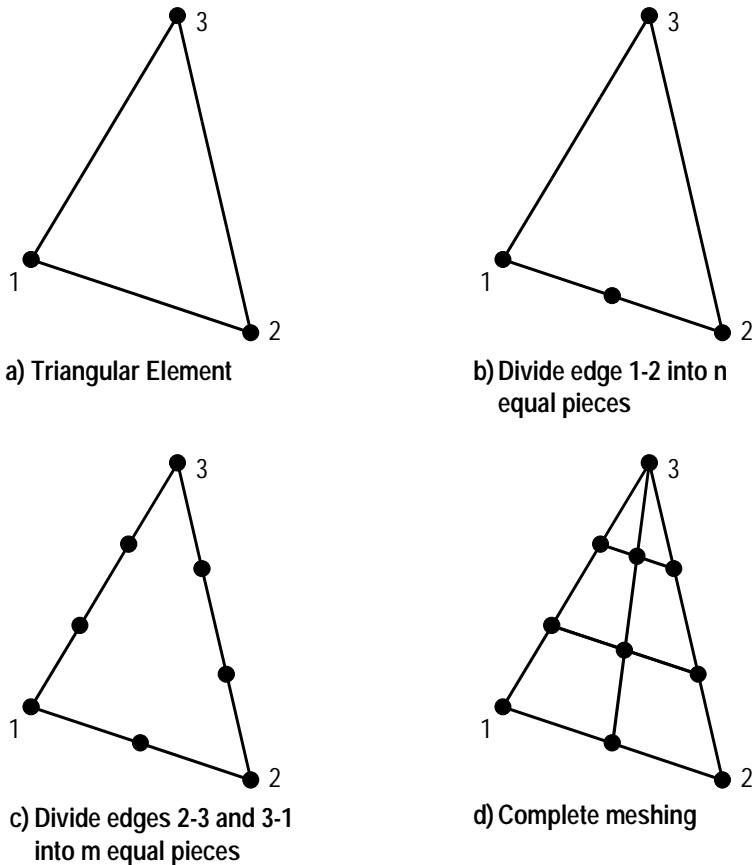
Sometimes it may be quicker and easier to use a trial and error process to determine which sides correspond to n and m when meshing an area object into n by m area objects.

Consider the four-sided area object shown in Figure 31-3a. The corner point numbers in this area object are determined as described in the previous subsection titled "Background Information."

Assume for this example that you specify the object is to be broken into 3 by 2 objects, that is, $n = 3$ and $m = 2$. Then ETABS does the following:

- Break the 1-2 and the 3-4 edges up into n equal pieces. (Note that the 1-2 edge extends from corner point 1 to corner point 2, and so on). In this example, n is equal to 3. The 1-2 and 3-4 edges are shown divided in Figure 31-3b.

Figure 31-4:
*Example of meshing
 a three-sided area
 object into n by m
 (2 by 3 in this case)
 area objects*



- Break the 2-3 and the 4-1 edges up into m equal pieces . In this example, m is equal to 2. The 2-3 and 4-1 edges are shown divided in Figure 31-3c.
- Mesh the area object by connecting the resulting mesh points on opposite sides of the object. That is, connect the mesh points on edge 1-2 to those on edge 3-4. Similarly, connect the mesh points on edge 2-3 to those on edge 4-1. This is illustrated in Figure 31-3d.

Now consider the three-sided area object shown in Figure 31-4a. The corner point numbers in this area object are determined as described in the previous subsection titled "Background Information."

Assume for this example that you specify the object is to be broken into 2 by 3 objects, that is, $n = 2$ and $m = 3$. (Note that this is different from the four-sided example that was broken into 3 by 2 objects). Then ETABS does the following:

- Break the 1-2 edge up into n equal pieces . In this example n is equal to 2. The 1-2 edge is shown divided in Figure 31-4b.
- Break the 2-3 and the 3-1 edges up into m equal pieces. In this example m is equal to 3. The 2-3 and 3-1 edges are shown divided in Figure 31-4c.
- Mesh the area object by connecting the resulting mesh points on the 1-2 edge to point 3 and by connecting the resulting mesh points on sides 2-3 and 3-1 together. This is illustrated in Figure 31-4d.

Note:

The 1-2 edge of an area object extends from corner point 1 to corner point 2, the 2-3 edge extends from corner point 2 to corner point 3, and so on.

**Note:**

You can mesh based on any one or any combination of the three sub-options of the Mesh Quads/Triangles At option.

Sometimes you may find it easier to guess which sides of the area object correspond to n and which correspond to m . If you guess wrong then simply undo the mesh using the **Edit menu > Undo** command or using the **Undo** button, , located on the main (top) toolbar. Then mesh again with the values for n and m reversed. In some cases you may find this trial and error process faster than trying to figure out beforehand exactly what ETABS will do.

Mesh Quadrilaterals and Triangles at Intersections and Selected Points on Edges

This subsection discusses the Mesh Quads/Triangles At option for the **Edit menu > Mesh Areas** command. This command works for three and four-sided area objects. It does not work for area objects with more than four sides.

Three Mesh Quads/Triangles At sub-options are available. They can be used either separately or in combination. The three sub-options are:

- Mesh quadrilaterals and triangles at intersections with visible grid lines.

**Tip:**

It is not necessary that opposite sides of four-sided area objects end up with the same number of meshing points defined. Similarly, it is not necessary that sides 2-3 and 3-1 of three-sided area objects end up with the same number of meshing points defined. ETABS internally adds additional meshing points as necessary such that opposite sides have equal numbers of meshing points.

31

- Mesh quadrilaterals and triangles at selected point objects on edges.
- Mesh quadrilaterals and triangles at intersections with selected line objects.

The basic concept is that each of the three sub-options either alone, or together with other sub-options, defines meshing points along the edges of the selected area object. It is not necessary that opposite sides of four-sided area objects end up with the same number of meshing points defined. Similarly, it is not necessary that sides 2-3 and 3-1 of three-sided area objects end up with the same number of meshing points defined. See the subsection titled "Background Information" earlier in this section for more information on the corner point labeling of area objects.

If opposite sides of four-sided area objects (or sides 2-3 and 3-1 of three-sided area objects) do not end up with the same number of meshing points defined then ETABS adds additional meshing points to the side with fewer points defined until the sides do have an equal number of meshing points. ETABS attempts to locate the added meshing points to provide as uniform a mesh as possible.

The following three sub-subsections discuss each of these meshing sub-options separately. The fourth sub-subsection considers an example where all of the "mesh based on" sub-options are used simultaneously.

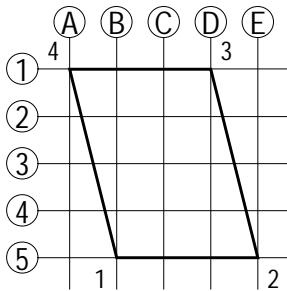
**Tip:**

When meshing at gridline intersections the gridlines must be visible. You can gain some control over meshing with this option by setting certain grid lines invisible if necessary.

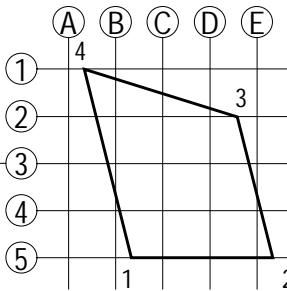
Mesh at Intersection with Visible Gridlines

This section discusses the Intersections with Visible Grids sub-option of the Mesh Quads/Triangles At option for the **Edit menu > Mesh Areas** command. This command works for three and four-sided area objects; it does not work for area objects with more than four sides.

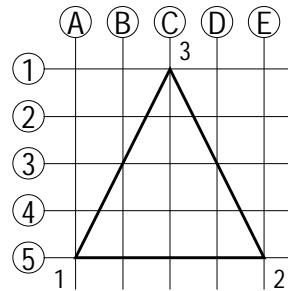
Consider the four-sided area object shown in Figure 31-5a. When you mesh this object at intersections with visible gridlines it appears as twelve objects as shown in Figure 31-5b. Note that the opposite sides of the area object are intersected by the same number of gridlines and thus it is not necessary for ETABS to add additional mesh points. The mesh is created by connecting



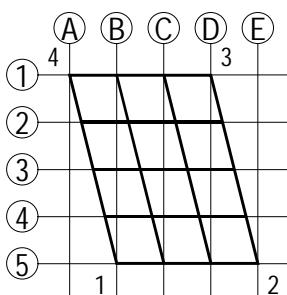
a) Four-sided area object



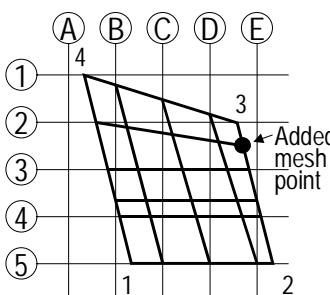
c) Four-sided area object with unequal mesh points



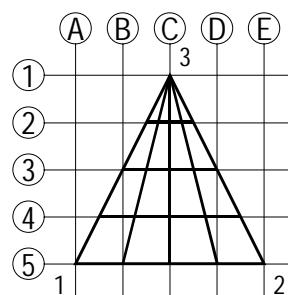
e) Three-sided area object



b) Area object in (a) meshed at intersections with gridlines



b) Area object in (c) meshed at intersections with gridlines



f) Area object in (e) meshed at intersections with gridlines

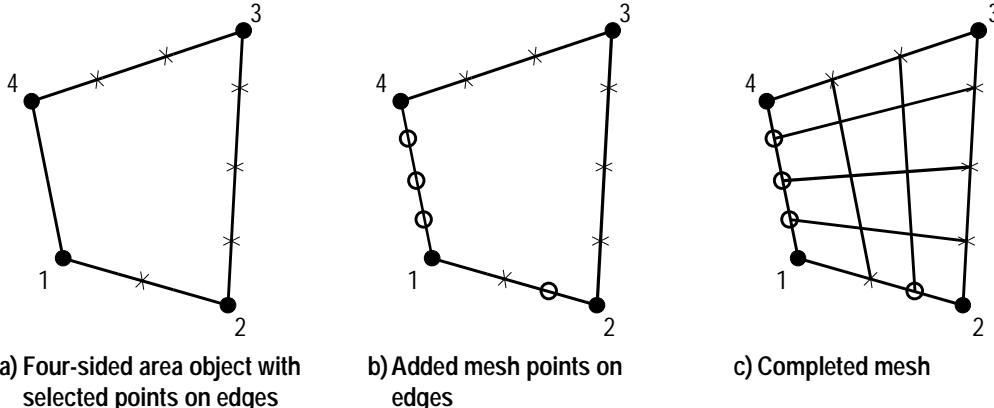
(Above)

Figure 31-5:
Example of meshing area objects at intersections with gridlines

the intersection points on opposite sides of the area object on a one-for-one basis.

Now consider the four-sided area object shown in Figure 31-5c. Note that sides 1-2 and 3-4 both have three intersections with grid lines so no additional mesh points need to be added by ETABS on these sides. Side 2-3 has three intersections and side 4-1 has four intersections. Therefore ETABS adds one mesh point to side 2-3 so that sides 2-3 and 3-4 have the same number of mesh points (intersections, etc.). This added mesh point is shown in Figure 31-5d. The mesh is created by connecting the intersection points on opposite sides of the area object on a one-for-one basis.

Finally, consider the three-sided area object shown in Figure 31-5e. Note that sides 2-3 and 3-1 both have three intersections with grid lines so no additional mesh points need to be added to the area object by ETABS. To create the completed mesh ETABS



31

(Above)

Figure 31-6:
Example of meshing area objects at selected point objects on edges of area object

**Tip:**

When meshing based on selected point objects on the edges of an area object be very careful to make certain that the point objects lie exactly on the edge of the area object. Otherwise the point objects may not be considered in the meshing process.

connects the mesh points (intersections) on sides 2-3 and 3-1 on a one-for-one basis. The gridline intersections on side 1-2 are all connected to point 3. This is illustrated in Figure 31-5f.

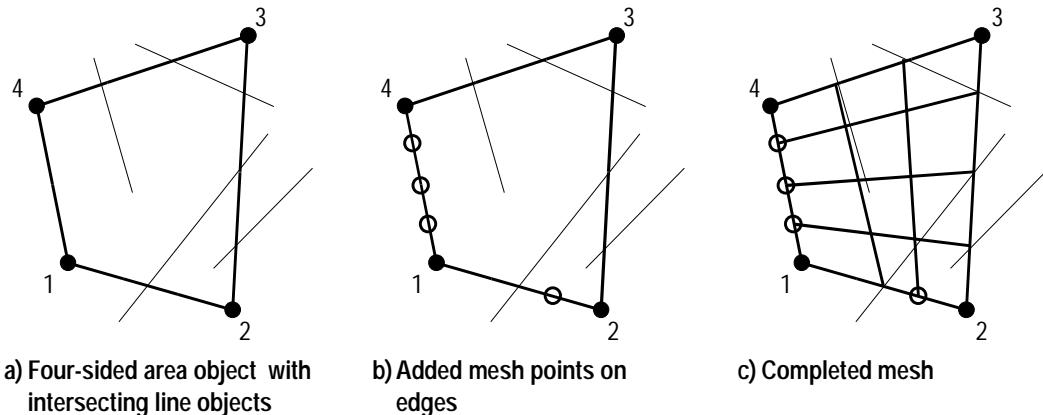
Mesh at Selected Point Objects on Edges

This section discusses the Selected Point Objects on Edges sub-option of the Mesh Quads/Triangles At option for the **Edit menu > Mesh Areas** command. This command works for three and four-sided area objects; it does not work for area objects with more than four sides.

Important note: This sub-option meshes area objects based on selected points that lie along the edge of the area object. It is important to note that the points must lie *exactly* on the edge of the area object, otherwise they may not be considered in the meshing process.

Consider the four-sided area object shown in Figure 31-6a. Note that selected points are denoted on the sides by X's. Side 1-2 has one selected point and side 3-4 has two selected points. Thus ETABS adds a mesh point to side 1-2. Side 2-3 has 3 selected points and side 4-1 has no selected points. Thus ETABS adds three mesh points to side 4-1. The added meshed points are shown as open circles in Figure 31-6b.

Figure 31-6c shows how ETABS completes the mesh by connecting the mesh points (selected points, etc.) on opposite sides of the area object.



(Above)

Figure 31-7:
Example of meshing area objects at intersections with selected line objects

See the subsection titled "Mesh at Intersection with Visible Gridlines" for discussion of meshing triangular area objects.

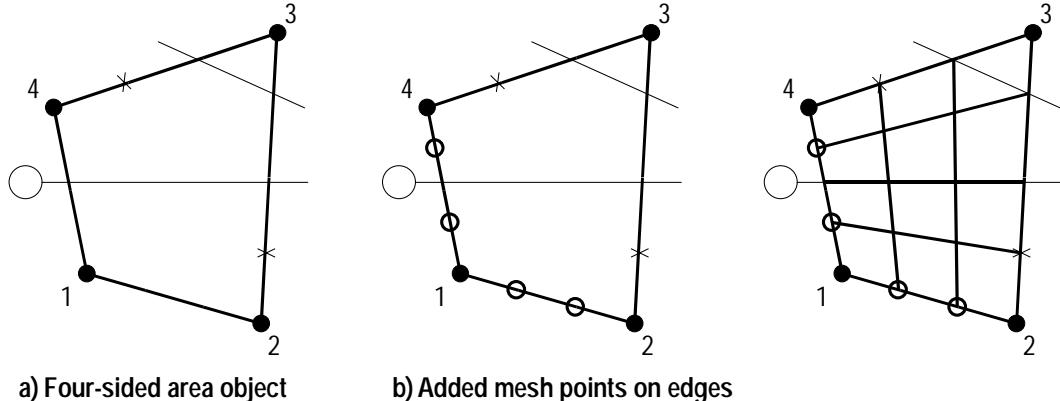
Mesh at Intersections with Selected Line Objects

This section discusses the Intersections with Selected Line Objects sub-option of the Mesh Quads/Triangles At option for the **Edit menu > Mesh Areas** command. This command works for three and four-sided area objects; it does not work for area objects with more than four sides.

Consider the four-sided area object shown in Figure 31-7a. The intersecting line objects are also shown in Figure 31-7a. Side 1-2 has one intersecting line object and side 3-4 has two intersecting line objects. Thus ETABS adds a mesh point to side 1-2. Side 2-3 has 3 intersecting line objects and side 4-1 has no intersecting line objects. Thus ETABS adds three mesh points to side 4-1. The added meshed points are shown as open circles in Figure 31-7b.

Figure 31-7c shows how ETABS completes the mesh by connecting the mesh points (intersecting line objects, etc.) on opposite sides of the area object.

See the subsection titled "Mesh at Intersection with Visible Gridlines" for discussion of meshing triangular area objects.



31

(Above)

Figure 31-8:
Example of meshing area objects

Example with Combined the Mesh Sub-options

Consider the four-sided area object shown in Figure 31-8a. Also shown are an intersecting line object, an intersecting gridline and two selected points on the edges of the area object. The selected points are designated by X's.

Side 1-2 has no intersections and no selected points along it. Side 3-4 has one intersecting line object and one selected point object along it. Thus ETABS adds two mesh points to side 1-2.

Side 2-3 has one intersecting gridline, one intersecting line object and one selected point object along it. Side 4-1 has one intersecting gridline. Thus ETABS adds two mesh points to side 4-1. The added meshed points are shown as open circles in Figure 31-8b.

Figure 31-8c shows how ETABS completes the mesh by connecting the mesh points (intersections, selected point objects, etc.) on opposite sides of the area object.

See the subsection titled "Mesh at Intersection with Visible Gridlines" for discussion of meshing triangular area objects.



Transformation of Loads into the ETABS Analysis Model

This chapter discusses how loads are transformed from your object-based ETABS model into the element-based analysis model. See Chapter 5 for discussion of the analysis model.

Background

Note:

In the analysis model a shell element can only have loads applied at its corner points.

In the ETABS analysis model the frame element is used to model beams, columns and braces. Loads can be applied anywhere along the frame element.

In the ETABS analysis model the shell element is used to model walls, floors and ramps. The shell element can only have loads applied at its corner points.

In the ETABS analysis model the link element is used to model links. The link element can only have loads applied at its end points.

Valid Loading

This section discusses valid loading for point, line and area objects. In this context valid means that the load will be transferred through the model ultimately to a support or a grounded spring.

Point Objects

Note:

Point objects have structural properties if they are assigned a support or a spring. Line objects have structural properties if they are assigned a frame section or a link property. Area objects have structural properties if they are assigned a wall, slab or deck section.

32

Force and moment loads can be applied to point objects in any global axis direction. The load is valid if one of the following is true:

- The point object is an end point of a line object with structural properties (frame section property or link property) and the point object has a complete load path to the ground.
- The point object is a corner point of an area object with structural properties (wall, slab or deck properties) and the point object has a complete load path to the ground.
- The point object itself is connected to the ground, that is, it is restrained.
- The point object lies on a line object with frame section properties (not link properties). The line object must have a complete load path to the ground.

In this case ETABS automatically and internally meshes (divides) the line object at the point object location in the analysis model. Thus, in the analysis model the point object is directly connected to frame elements that have a complete load path to the ground.

- The point object lies on an area object that has structural properties (wall, slab or deck properties). The area object must have a complete load path to the ground. In this case, in the analysis model, ETABS transforms the load either to beams along the edges of the shell element (area object) or to the corner points of the shell element area object.

A load on a point object is not valid, and thus not properly included in the analysis model if there is no load path provided to the ground using one of the methods described above. Upon running the analysis the result will either be that the load is lost (ignored by ETABS) or that it causes an instability.

For additional information on loads applied to point objects see the section titled "Assignments to Point Objects" in Chapter 14.

Line Objects



Tip:

In general, we recommend that you assign line loads to line objects that also have frame section properties and are directly connected to other objects with structural properties that provide a complete load path to the ground.

Forces and moments can be applied to line objects in any local or global axis direction. The load is valid if one of the following is true:

- The line object has frame section properties assigned to it and has a complete load path to the ground.
- The line object lies on an area object that has structural properties (wall, slab or deck properties). The area object must have a complete load path to the ground. In this case, in the analysis model, ETABS transforms the load either to beams along the edges of the shell element (area object) or to the corner points of the shell element.

A load on a line object is not valid, and thus not ultimately included in the analysis if there is no load path provided to the ground using one of the methods described above. When the analysis is run the result will either be that the load is lost (ignored by ETABS) or that it causes an instability.

For additional information on loads applied to point objects see the section titled "Assignments to Line Objects" in Chapter 14.

Area Objects

Force loads can be applied to area objects in any local or global axis direction. The load is valid if one of the following is true:

- The area object has structural properties (wall, slab or deck properties) assigned to it and has a complete load path to the ground.

- The area object, call it F1, lies directly on another area object, call it F2, that has structural properties (wall, slab or deck properties). Area object F2 must have a complete load path to the ground. In this case, in the analysis model, ETABS transfers the load on area object F1 either to beams along the edges of the shell element (area object F2) or to the corner points of the shell element.

A load on an area object is not valid, and thus not ultimately included in the analysis if there is no load path provided to the ground using one of the methods described above. When the analysis is run the result will either be that the load is lost (ignored by ETABS) or that it causes an instability.

For additional information on loads applied to point objects see the section titled "Assignments to Area Objects" in Chapter 14.

Introduction to Load Transformation

The main issue for load transformation in ETABS is how point loads, line loads and area loads that lie on an area object in your object-based ETABS model are represented in the analysis model. The point, line and area loads that are present in your object-based model in any of the locations discussed in the previous section are transformed to act on either frame elements along the edges of shell elements or the corner points of shell elements in the analysis model. The remainder of this chapter describes how this load transformation is done.

There are four distinct types of load transformation in ETABS. Each has its own set of rules. These types of load transformation are:

- **Out-of-plane load transformation for floor-type area objects with deck section properties:** In this case, in the analysis model, loads are transformed either to beams along the edges of shell elements or to the corner points of the shell elements. The load transformation takes into account that the deck only spans in one direction.

- **Out-of-plane load transformation for floor-type area objects with slab section properties that have membrane behavior only (not plate bending or shell behavior):** In this case, in the analysis model, loads are transformed either to beams along the edges of shell elements or to the corner points of the shell elements. The load transformation takes into account that the slab spans in two directions.
- **Out-of-plane load transformation for all other types of area objects not listed in the two above bullet items:** In this case, in the analysis model, loads are transformed directly to the corner points of shell elements using a bilinear interpolation function.
- **In-plane load transformation for all types of area objects:** In this case, in the analysis model, loads are transformed directly to the corner points of shell elements using a bilinear interpolation function.

These cases can be stated in another way as follows. ETABS uses a bilinear interpolation function to transform the loads applied to or on area objects in your object based ETABS model to the corner points of shell elements in the analysis model for all area objects with two special case exceptions. Those two exceptions are:

- Transformation of out-of-plane (vertical) loads for floor-type area objects with deck section properties. Recall that by definition area objects with deck section properties have membrane behavior only.
- Transformation of out-of-plane (vertical) loads for floor-type area objects with slab section properties that have membrane behavior only.

Note that both of these exceptions are associated with vertical loads for floors that have membrane properties only, and thus floors that ETABS can automatically mesh into the analysis model.



Note:

For all cases except vertical loads on membrane floor slabs and decks, ETABS uses a bilinear interpolation function to transform the loads applied to or on area objects in your object based ETABS model to the corner points of shell elements in the analysis model.

Load Transformation for Area Objects

ETABS uses a bilinear interpolation method to transform all *in-plane loads* (area, line and point) applied to area objects in your object-based ETABS model to the corner points of the associated shell elements in the analysis model. Note that the load transformation occurs after any automatic meshing into the analysis model done by ETABS. See Chapter 30 for discussion of the ETABS automatic meshing of membrane floors into the analysis model.

This same method of load transformation is also used to transform *out-of-plane* loads on all area objects except for floor-type (horizontal) area objects with membrane properties only. This exception includes horizontal area objects with deck section properties and horizontal area objects with slab properties that have membrane behavior only (not plate bending behavior or full shell behavior). See the sections titled “Vertical Load Transformation for Floors with Deck Properties” and “Vertical Load Transformation for Floors with Membrane Slab Properties” later in this chapter for discussion of these exceptions.

This section explains how the load transformation is done for point objects. Line and area objects are internally discretized by ETABS into a series of point loads and handled in exactly the same way. In other words, area and line loads are numerically integrated using the method described for point loads.

32



Note:

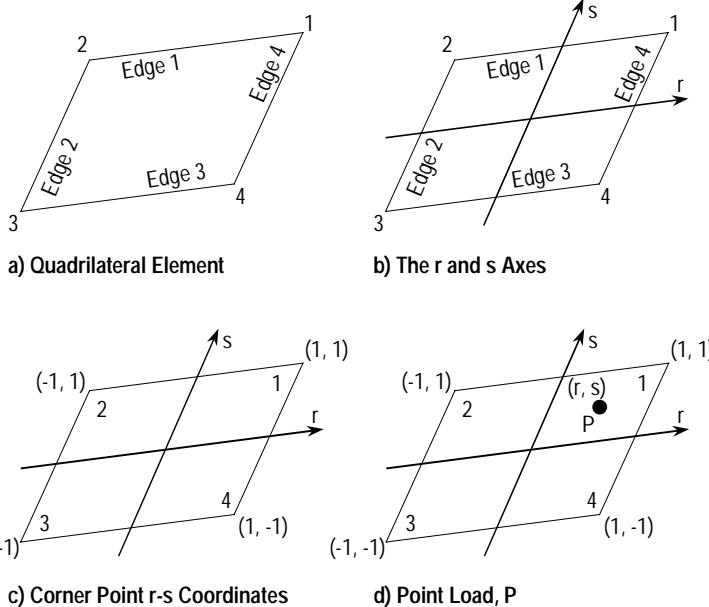
The r and s axes may or may not be perpendicular to each other.

Consider the arbitrary quadrilateral element shown in Figure 32-1a with its corner points and edges labeled 1, 2, 3 and 4 for reference in this discussion. ETABS begins by defining two axes labeled r and s as shown in Figure 32-1b. Axis r goes through the midpoints of edges 2 and 4. Axis s goes through the midpoints of edges 1 and 3. Note that the r and s axes need not be perpendicular and that in this example they are not perpendicular.

Next ETABS normalizes the coordinates in the r-s coordinate system such that the coordinates of the four corner points of the area object are :

- Point 1: $(r_1, s_1) = (1, 1)$
- Point 2: $(r_2, s_2) = (-1, 1)$

Figure 32-1:
Example of transfer
of out-of-plane loads
for other area ob-
jects



- Point 3: $(r_3, s_3) = (-1, -1)$
- Point 4: $(r_4, s_4) = (1, -1)$

These coordinates are shown in Figure 32-1c. Note that this normalization is the key assumption in this method. It is a perfectly valid assumption if the quadrilateral is a square, rectangular or a parallelogram. However, if the area object is not one of these shapes then the interpolation method is not exactly what you might expect. We will discuss this more later in this section. Note that in most cases the area objects in your object-based ETABS model and their associated shell elements in the analysis model are squares, rectangles or parallelograms.

Consider a point load P that is located on the area object at arbitrary coordinates (r, s) in the r-s coordinate system as shown in Figure 32-1d. The distribution of this point load to the four corner points is determined using Equations 32-1a through 32-1d:

$$N_1 = \frac{P}{4} (1 + r)(1 + s) \quad \text{Eqn. 32-1a}$$

$$N_2 = \frac{P}{4} (1-r)(1+s) \quad \text{Eqn. 32-1b}$$

$$N_3 = \frac{P}{4} (1-r)(1-s) \quad \text{Eqn. 32-1c}$$

$$N_4 = \frac{P}{4} (1+r)(1-s) \quad \text{Eqn. 32-1d}$$

where,

N_n = Portion of load P distributed to corner point n ($n = 1, 2, 3$ or 4).

P = Point load acting on area object at location (r, s) in the r - s coordinate system.

r = Coordinate of point load, P , along the r -axis.

s = Coordinate of point load, P , along the s -axis.

Note that Equations 32-1a through 32-1d are based on the assumption that the corner points of the area object fall at coordinates $(1, 1)$, $(-1, 1)$, $(-1, -1)$ and $(1, -1)$.

As an example suppose the coordinates of point P shown in Figure 32-1d are $(r, s) = (0.5, 0.5)$ and that P is a load of 1 (units are not important for example purposes). Then the four corner point reactions are computed as:

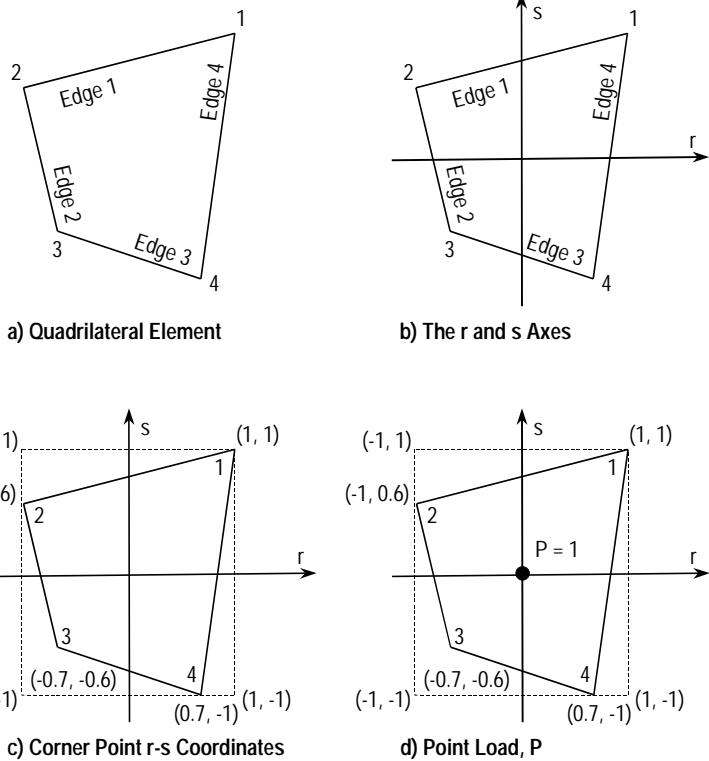
$$N_1 = \frac{P}{4} (1+r)(1+s) = \frac{1}{4} (1+0.5)(1+0.5) = 0.5625 = \frac{9}{16}$$

$$N_2 = \frac{P}{4} (1-r)(1+s) = \frac{1}{4} (1-0.5)(1+0.5) = 0.1875 = \frac{3}{16}$$

$$N_3 = \frac{P}{4} (1-r)(1-s) = \frac{1}{4} (1-0.5)(1-0.5) = 0.0625 = \frac{1}{16}$$

$$N_4 = \frac{P}{4} (1+r)(1-s) = \frac{1}{4} (1+0.5)(1-0.5) = 0.1875 = \frac{3}{16}$$

Figure 32-2:
Example of transfer
of out-of-plane loads
for other area ob-
jects that are not
square, rectangular
or a parallelogram



32

Note that:

$$N_1 + N_2 + N_3 + N_4 = \frac{9}{16} + \frac{3}{16} + \frac{1}{16} + \frac{3}{16} = \frac{16}{16} = 1.00 = P$$

as it should.

Now let's consider an area object that is not a square, rectangle or parallelogram. As previously mentioned, in this case the bilinear interpolation will not be exactly what you might expect. This happens because even though the corner points of the area object do not fall at the coordinates $(1, 1)$, $(-1, 1)$, $(-1, -1)$ and $(1, -1)$, ETABS uses Equations 32-1a through 32-1d for the bilinear interpolation which assumes that the corner points do fall at these coordinates.

Consider the example shown in Figure 32-2a. This is a quadrilateral area object that is not a square, rectangle or parallelogram. As before, the corner points and edges are labeled 1, 2, 3 and 4. Figure 32-2b shows the r-axis going through the midpoints of edges 2 and 4 and the s-axis going through the midpoints of edges 1 and 3.

Figure 32-2c shows a dashed rectangle enclosing the area object. The r-s coordinate system is normalized such that the coordinates of the corner points of this dashed rectangle are (1, 1), (-1, 1), (-1, -1) and (1, -1). The actual coordinates of the corner points of the area object are (1, 1), (-1, 0.6), (-0.7, -0.6) and (0.7, -1) as shown in Figure 32-2c.

Figure 32-2d shows a point load, P, applied at the origin of the r-s coordinate system. Let's again assume that P is a load of 1 (units are not important for example purposes) and calculate the distribution of the load using Equations 32-1a through 32-1d just as ETABS does.

Before making the calculation it is very important to make sure you are not confused by Figures 32-2 c and d. The coordinates shown for the area object in these figures, (1, 1), (-1, 0.6), (-0.7, -0.6) and (0.7, -1), are not the ones ETABS uses when calculating corner point loads. Instead, for calculating the corner point loads ETABS assumes that the corner points coordinates of the area object are (1, 1), (-1, 1), (-1, -1) and (1, -1). Thus we proceed with our calculation using these coordinates.

$$N_1 = \frac{P}{4} (1+r)(1+s) = \frac{1}{4} (1+0)(1+0) = 0.25 = \frac{1}{4}$$

$$N_2 = \frac{P}{4} (1-r)(1+s) = \frac{1}{4} (1-0)(1+0) = 0.25 = \frac{1}{4}$$

$$N_3 = \frac{P}{4} (1-r)(1-s) = \frac{1}{4} (1-0)(1-0) = 0.25 = \frac{1}{4}$$

$$N_4 = \frac{P}{4} (1+r)(1-s) = \frac{1}{4} (1+0)(1-0) = 0.25 = \frac{1}{4}$$

Note that:

$$N_1 + N_2 + N_3 + N_4 = \frac{1}{4} + \frac{1}{4} + \frac{1}{4} + \frac{1}{4} = \frac{4}{4} = 1.00 = P$$

as it should.

Note:

In ETABS static equilibrium is always maintained when transforming loads from the object-based model to the analysis model.

Note that in the example of Figure 32-2 the sum of the loads distributed to the four corners is equal to the applied load. Thus static equilibrium is maintained. However, the amount of load distributed to each of the four corners of the area object is really more suited to an area object shaped like the dashed quadrilateral in Figures 32-2c and d than to the actual area object considered shown in Figure 32-2a.

Figure 32-2 and its accompanying discussion illustrates what is meant earlier in this section by the statement that when the area object is *not* a square, rectangle or parallelogram the bilinear interpolation may not yield results that are exactly what you might expect. In most models your area objects and associated shell elements will be squares, rectangles or parallelograms and this won't be an issue. Even when it is an issue the total applied load is correct, it just may be transformed to the corner points in a slightly different distribution than you might expect.

Vertical Load Transformation for Floors with Deck Properties

The discussion in this section only applies to floor-type (horizontal) area objects with deck section properties. Note that by definition in ETABS deck section properties only have membrane behavior. This discussion further only applies to out-of-plane (vertical) loads acting on these deck sections.

This section describes the transformation of point, line and uniform loads acting on floors with deck sections from your object-based ETABS model to the element-based analysis model. Note that the load transformation occurs after any automatic meshing done by ETABS into the analysis model. See Chapter 30 for discussion of the ETABS automatic meshing for membrane floors.

In ETABS the load distribution for deck sections is one way, that is, the deck is assumed to span in one direction only (the local 1-axis direction of the area object to which the deck is assigned). This is in contrast to slab sections which are assumed to span in two directions. The transformation of loads from the object-based model to the analysis model considers this span direction for floors modeled with deck properties.

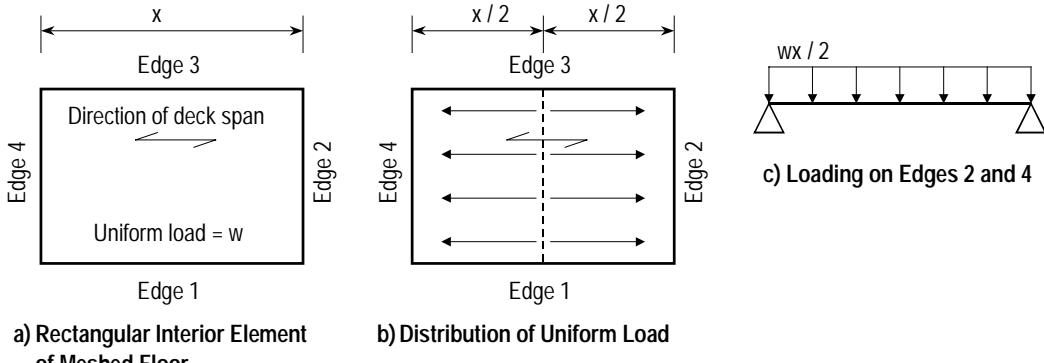
Before the load transformation is done for decks ETABS first automatically meshes the deck into quadrilateral elements as described in Chapter 30 (unless you have already completely meshed it yourself either by drawing it that way or by using the manual meshing methods described in Chapter 31). Once the meshing is complete ETABS knows which sides of the meshed shell elements have real beams along them and which have imaginary beams. It also knows which edges of the meshed shell elements are also edges of the deck. Armed with this information ETABS can begin to transform the load into the analysis model.

Note: If you are unfamiliar with the concept of imaginary beams then you should read Chapter 30 before proceeding further in this section.

The remainder of this section is divided into four subsections. The first subsection describes how loads are transformed for interior meshed deck elements that are rectangular. The second subsection describes how loads are transformed for interior meshed deck elements of any general quadrilateral shape. The third subsection describes how loads are transformed for exterior meshed deck elements that have one or more of their edges along the edge of the deck. The final subsection discusses the effect of openings on the transformation of loads on deck sections.

Rectangular Interior Meshed Element

Figure 32-3a shows a rectangular interior element of a meshed floor with deck section properties. The term interior in this case means that no edge of the element is also an edge of the deck. The edges of the element are labeled 1, 2, 3 and 4 for reference. The direction of the deck span (local 1-axis) is oriented parallel to edges 1 and 3 as indicated by the arrow.



(Above)

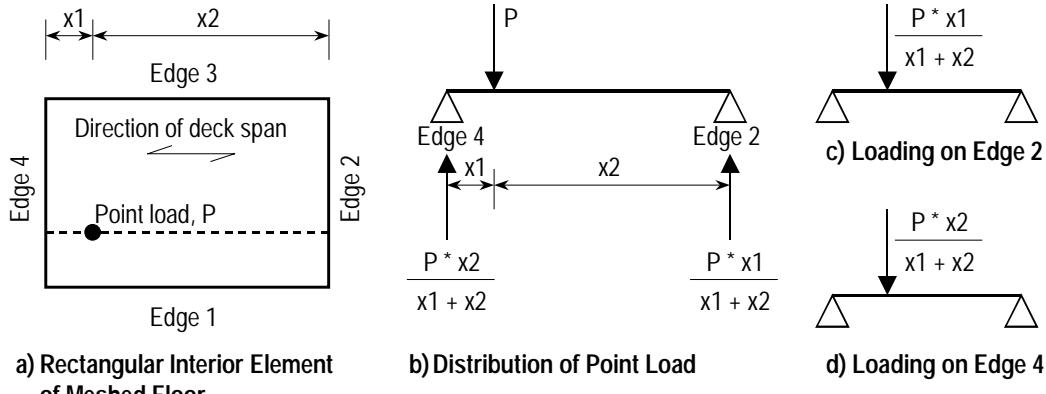
Figure 32-3:
Example of rectangular interior meshed element with a uniform load

Figure 32-3b illustrates how a uniform load, w (force/length², e.g., psf), on this element is transformed into the analysis model. A line is drawn through the center point of the element perpendicular to the span of the deck. In this case that line goes from the center of edge 1 to the center of edge 3. This line is shown dashed. All of the load on the edge 2 side of the line is tributary as a uniform load to the beam (real or imaginary) along edge 2 of the element. Similarly, all of the load on the edge 4 side of the line is tributary as a uniform load to the beam (real or imaginary) along edge 4 of the element. Thus the beams along edges 2 and 4 are loaded as shown in Figure 32-3c.

Note:

If the supporting member at the end point of an imaginary beam is itself imaginary, then the load from the imaginary beam tributary to that end point is lost, that is, it is ignored by ETABS.

If the beams along edges 2 and 4 are real beams then ETABS is done with the uniform load transformation, that is, it has transformed the load onto adjacent beams. If the beams along edges 2 and 4 are imaginary beams then ETABS distributes the uniform load on the imaginary beams to the end points of the imaginary beams as point loads. This distribution assumes that the imaginary beams are simply supported at their end points. The supporting members at the end points of the imaginary beams must be real members such as columns, walls or other beams. If the supporting member at an end point of an imaginary beam is itself imaginary, then the load from the imaginary beam tributary to that end point is lost, that is, it is ignored by ETABS (except in the special case of an exterior meshed element which is discussed later).



(Above)

Figure 32-4:
Example of rectangular interior meshed element with a point load

32

Now consider the case of a point load, P , at an arbitrary location on this rectangular element as shown in Figure 32-4a. ETABS distributes this point load to the appropriate edge beams (based on the direction of the deck span) assuming that the meshed deck element spans simply in the direction of the deck span from one edge to another. In this example the deck spans simply from edge 4 to edge 2 as shown by the dashed line in Figure 32-4a and as shown in Figure 32-4b.

Figure 32-4b shows how ETABS calculates the reactions to be applied as point loads on edge beams 2 and 4 due to the point load. Finally, Figures 32-4c and 32-4d show the loads applied to edges 2 and 4 respectively.

As with the uniform load, if the beams along edges 2 and 4 are real beams then ETABS is done with the point load transformation, that is, it has transformed it to adjacent beams. If the beams along edges 2 and 4 are imaginary beams then ETABS distributes the point load on the imaginary beam to the end points of the imaginary beams as point loads. This distribution assumes that the imaginary beams are simply supported at their end points. The supporting members at the end points of the imaginary beams must be real members such as columns, walls or other beams. If the supporting member at an end point of an imaginary beam is itself imaginary, then the load from the imaginary beam tributary to that end point is lost, that is, it is ignored by ETABS (except in the special case of an exterior meshed element which is discussed later).

A line load is transformed in a similar fashion to that for a point load using a numerical integration technique. The line load is discretized as a series of point loads which are transformed to surrounding beams as previously described for point loads. The series of point loads is then converted back to a line load on the surrounding beams. An area load that does not cover the entire element is also transformed in a similar fashion to that for a point load using a numerical integration technique.

General Interior Meshed Element

Now consider the more general interior element of a meshed floor with deck section properties shown in Figure 32-5a. When transforming loads from the object-based ETABS model to the analysis model, the first thing ETABS does is construct two lines drawn from two corners of the element, parallel to the direction of the deck span until they intersect the edges of the element. These two lines are shown dashed in Figure 32-5b.

Note:

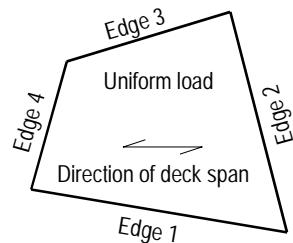
Line and area loads are transformed in a similar fashion to that for point loads.

ETABS uses a numerical integration technique where the lines and areas are discretized as a series of points.

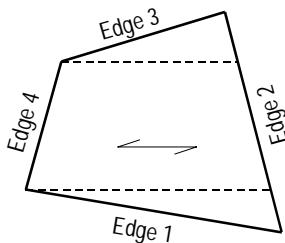
Next ETABS constructs two more lines drawn from the other two corners to the midpoint of the two previously constructed lines. These lines are added in Figure 32-5c. Then ETABS constructs one more line by connecting the midpoints of the two initially constructed lines. This line is added in Figure 32-5d. Now the load can be transformed from the object-based ETABS model to the analysis model.

Figure 32-5e shows how uniform load on this element is transformed (distributed) to its edges. Figures 32-5f, g, h and i show how the load would appear on edges 1, 2, 3 and 4 respectively.

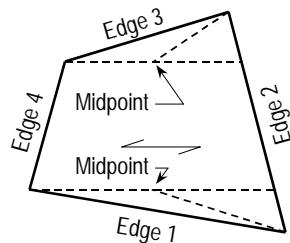
As discussed for the rectangular element, if the beams along the edges are real beams then ETABS is done with the uniform load transformation, that is, it has transformed it to adjacent beams. If the beams along the edges are imaginary beams then ETABS distributes the uniform load on the imaginary beam to the end points of the imaginary beams as point loads. This distribution assumes that the imaginary beams are simply supported at their end points. The supporting members at the end points of the imaginary beams must be real members such as columns, walls or other beams. If the supporting member at an end point of an imaginary beam is itself imaginary, then the load from the imaginary beam tributary to that end point is lost, that is, it is ig-



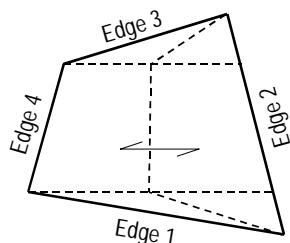
a) General Interior Element of Meshed Floor Deck



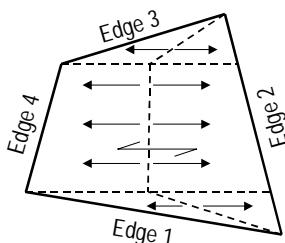
b)



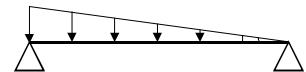
c)



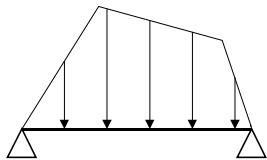
d)



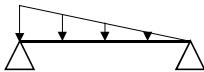
e) Transformation of Uniform Load



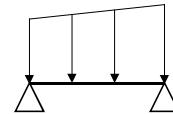
f) Loading on Edge 1



g) Loading on Edge 2



h) Loading on Edge 3



i) Loading on Edge 4

(Above)
Figure 32-5:

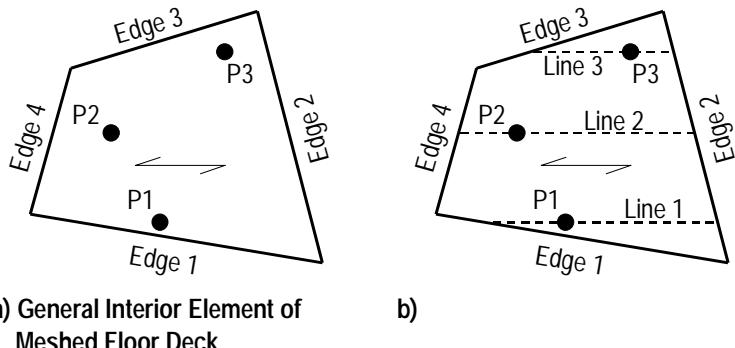
Example of general interior meshed element with a uniform load

nored by ETABS (except in the special case of an exterior meshed element which is discussed later).

Figure 32-6a shows the same general element with three point loads on it labeled P1, P2 and P3. Similar to the previously discussed point load on the rectangular element, ETABS transforms these point loads to the appropriate edge beams (based on the direction of the deck span) assuming that the meshed shell element spans simply from one edge to another.

Thus the point load P1 is transformed from the object-based ETABS model to the analysis model as if it were a point load on Line 1 in Figure 32-6b. Note that Line 1 spans simply from edge 1 to edge 2. Similarly, point load P2 is transformed as if it were a

Figure 32-6:
Example of general interior meshed element with a point load



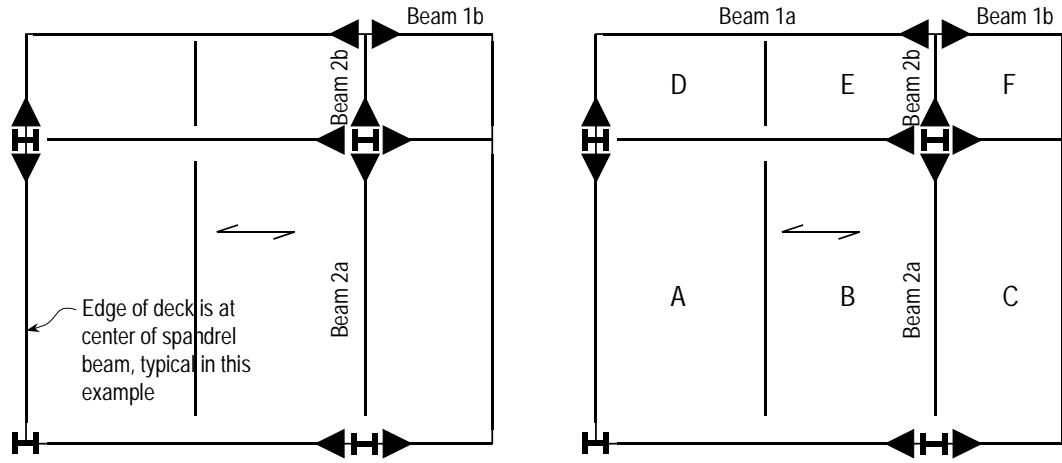
a) General Interior Element of
Meshed Floor Deck

b)

point load on Line 2 where Line 2 spans simply from edge 4 to edge 2. Finally, point load P3 is transformed as if it were a point load on Line 3 where Line 3 spans simply from edge 3 to edge 2. See the discussion of point loads in the previous subsection titled "Rectangular Interior Meshed Element" (for example, see Figure 32-4) for additional information.

As usual if the beams along the edges are real beams then ETABS is done with the point load transformation, that is, it has transformed it to adjacent beams. If the beams along the edges are imaginary beams then ETABS distributes the point load on the imaginary beam to the end points of the imaginary beams as point loads. This distribution assumes that the imaginary beams are simply supported at their end points. The supporting members at the end points of the imaginary beams must be real members such as columns, walls or other beams. If the supporting member at an end point of an imaginary beam is itself imaginary, then the load from the imaginary beam tributary to that end point is lost, that is, it is ignored by ETABS (except in the special case of an exterior meshed element which is discussed later).

Similar to the rectangular element example, a line load or area load covering part of an element is discretized as a series of point loads which are transformed as described above for point loads and then converted into line loads on the surrounding beams (real or imaginary).



32

a) Floor Plan

b) Deck Meshing

Exterior Meshed Element

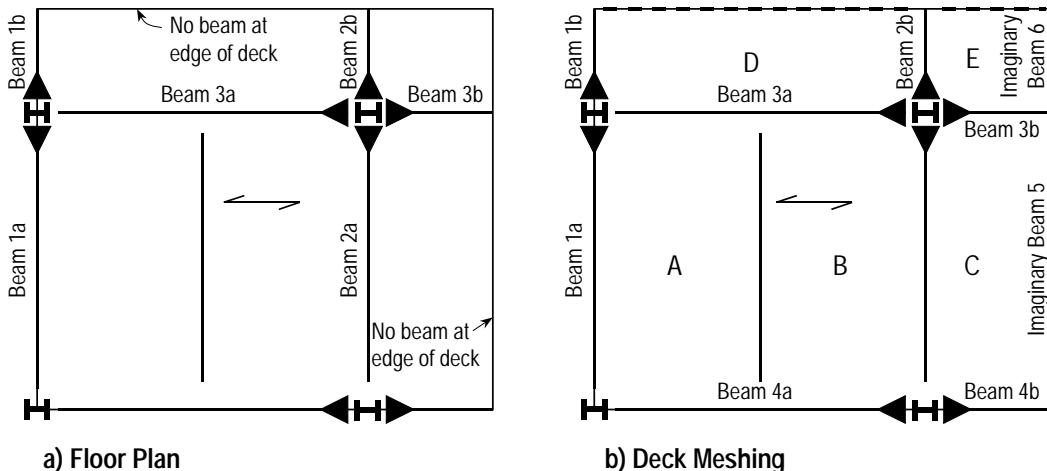
(Above)

Figure 32-7:
*Example of exterior
 meshed elements
 with real beams on
 all sides*

This subsection uses three separate examples to illustrate how ETABS transforms vertical load on exterior deck elements from the object-based ETABS model into the analysis model. In this context an exterior deck element is one that has one or more sides along the edge of the deck.

Consider the example shown in 32-7a. In this instance the deck is fully framed by beams. ETABS meshes the deck as shown in Figure 32-7b into areas A, B, C, D, E and F. Loads are transferred to beams as previously described in the subsection titled "Rectangular Interior Meshed Element." Thus if you fully frame exterior meshed elements, that is, if you put real beams on all four sides of them then the vertical load distribution is the same as previously discussed for an interior meshed element.

Note that for convenience and simplicity of this example the edge of the deck in this example is aligned with the center of the spandrel beams. We will return to discuss this, and show an example where the edge of deck extends beyond the spandrel beam, after discussing more examples in Figures 32-8, 32-9 and 32-10.



(Above)

Figure 32-8:
Example of exterior
meshed elements
with cantilever
beams extending to
edge of deck

Now consider the example shown in Figure 32-8a. In this case cantilever beams are included in the model extending out into the deck overhang area but no edge beams are included.

Figure 32-8b shows the imaginary beams added by ETABS as dashed lines and illustrates that ETABS meshes the deck into five areas labeled A, B, C, D and E. The load transformation from the object-based ETABS model to the analysis model in areas A and B is as previously described in the subsection titled "Rectangular Interior Meshed Element."

The load transformation in area C is also as previously described in the subsection titled "Rectangular Interior Meshed Element." Note that in this case since Imaginary Beam 5 is an imaginary beam, the load tributary to Imaginary Beam 5 is distributed to its end points and applied as point loads to the real cantilever beams (Beam 3b and Beam 4b) that support it.

The load transformation in area D is also as previously described in the subsection titled "Rectangular Interior Meshed Element." Note that all of the load in this area is transformed to Beam 1b and Beam 2b as uniform load because of the direction of the deck span.

Note:

ETABS recognizes several special case conditions for transforming load on exterior deck sections.

Area E represents one of the special case areas that ETABS recognizes. If this area were treated as a typical area (**which it is not!**) then the load would be transformed to Beam 2b and Imaginary Beam 6 (note direction of deck span) as uniform load. Half of the load on Imaginary Beam 6 would be transferred to the end of Beam 3b and the other half of the load on Imaginary Beam 6 would be lost because it is tributary to another imaginary beam. **This, however, is not how ETABS treats area E.**

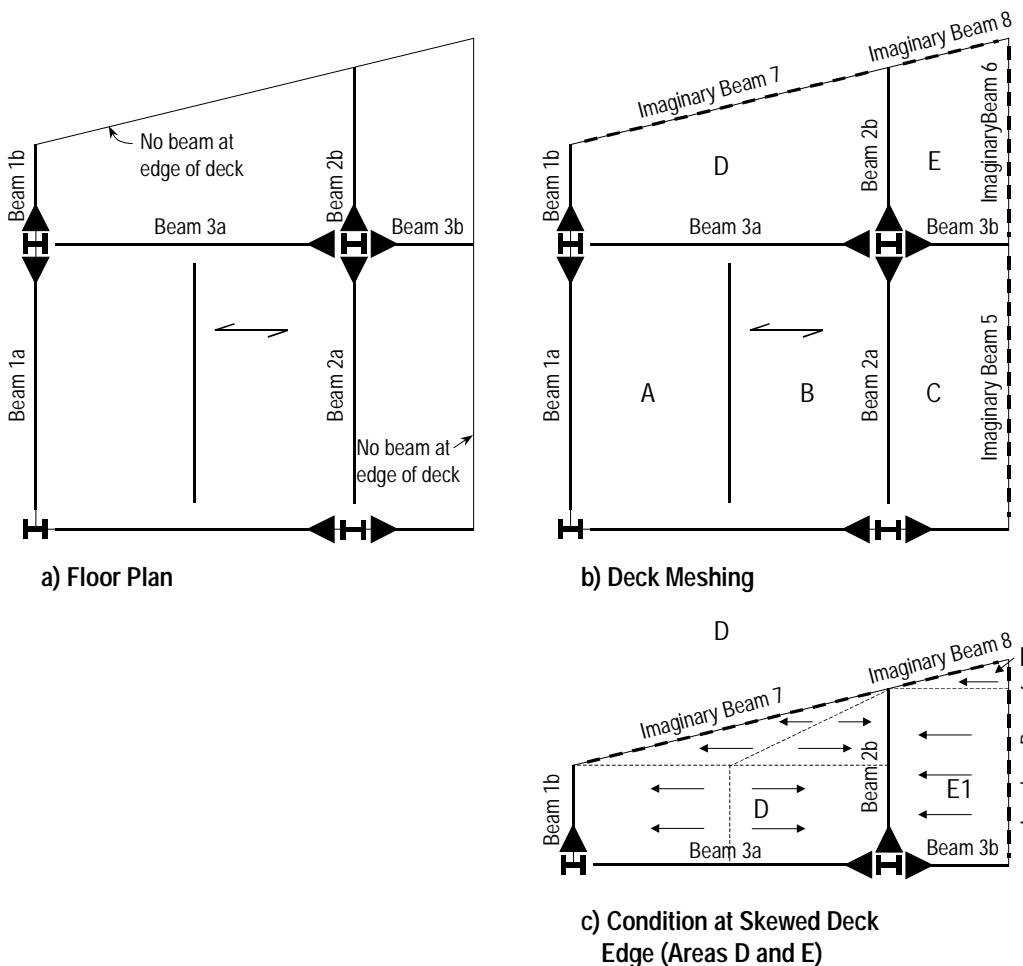
Note:

When distributing vertical load on automatically meshed floors ETABS provides special treatment to area objects at the edge of the deck or slab.

In this special case exterior corner condition, ETABS recognizes that some of the load will be lost if it distributed in the usual fashion. Thus, in this case, ETABS transforms all of the load in Area E to the real cantilever beam, Beam 2b as a distributed load. Again this is a special case that is different from how ETABS transforms the load for typical deck sections. Note that if Beam 2b were also an imaginary beam then ETABS would transform all of the load directly to the column as a point load. This will be addressed later in the example shown in Figure 32-10.

Figure 32-9a shows the same example as in Figure 32-8a except that the top edge of the deck is now skewed. Figure 32-9b shows the imaginary beams created by ETABS and labels the areas of the deck mesh as A, B, C, D and E. The vertical load transformation from the object-based ETABS model to the analysis model in areas A, B and C of Figure 32-9b is identical to that previously described for areas A, B and C of Figure 32-8b.

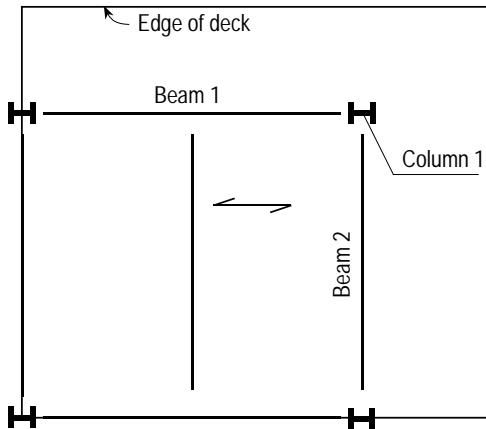
The transformation of vertical load in area D of Figure 32-9 is illustrated in Figure 32-9c. The transformation of load in this area is as described in the previous subsection titled "General Interior Meshed Element." In the rectangular portion of area D half of the load goes to Beam 1b and half goes to Beam 2b as distributed loads. In the triangular portion of area D the load is transformed to Imaginary Beam 7 and Beam 2b as distributed loads. The load tributary to Imaginary Beam 7 is distributed to its end points and applied to Beam 1b and Beam 2b as point loads.



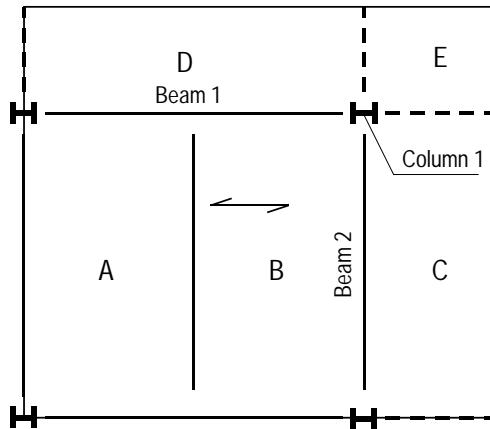
(Above)

Figure 32-9:
Example of exterior
meshed elements
with cantilever
beams extending to
edge of a skewed
deck

Area E in Figure 32-9b again represents one of the special case areas that ETABS recognizes when transforming loads into the analysis model. In this special case exterior corner condition, ETABS recognizes that some of the load will be lost if it is transformed in the usual fashion. Thus ETABS transforms all of the load in the rectangular portion of Area E, labeled E1 in Figure 32-9c, as a uniform load on the real cantilever beam, Beam 2b. The load goes onto Beam 2b, not Beam 3b, because of the direction of the deck span. (As an aside, if the deck span direction were skewed, say at a 45-degree angle, then the load in area E1 is transformed to both Beam 2b and Beam 3b). The load in the triangular portion of area E, labeled E2, is transformed to the



a) Floor Plan



b) Deck Meshing

32

(Above)

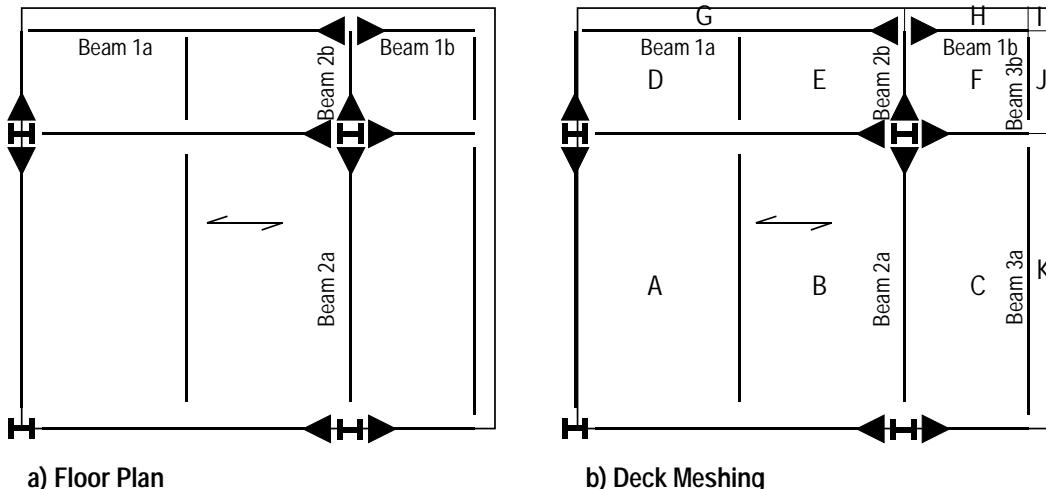
Figure 32-10:
*Example of exterior
 meshed elements
 with overhanging
 slab*

free end of Beam 2b as a point load. Note that if Beam 2b were also an imaginary beam then ETABS transforms all of the load in areas E1 and E2 directly to the column as a point load.

Figure 32-10a shows the condition where an overhanging deck is modeled but overhanging beams (cantilevers) are not included in the model. Figure 32-10b shows how ETABS defines imaginary beams (shown dashed) and how the deck is meshed into five areas labeled A, B, C, D and E. Note that the imaginary cantilever beams are defined parallel to their back spans. See Chapter 30 which includes information on how these imaginary beams are defined.

The vertical load transformation from the object-based ETABS model to the analysis model in areas A and B of Figure 32-10b is identical to that previously described for areas A and B of Figure 32-8b. Areas C, D and E in Figure 32-10b are treated as special cases by ETABS.

All of the load in area C is transformed to Beam 2 as a uniform load. All of the load in area D is transformed to Beam 1 as a uniform load. Note that ETABS makes this assumption even though the orientation of the deck suggests that the load is transformed in the other direction. You might say that for area D ETABS assumes that the deck span direction has been turned 90 degrees. Finally, all of the load in area E is transformed to Column 1 as a



(Above)

Figure 32-11:
Example of exterior
meshed elements
with overhanging
slab

32

point load. The transformation of load is similar if the edges of the deck are skewed.

Now let's return to the example discussed in Figure 32-8. Recall that for convenience and simplicity the edge of the deck in that example is aligned with the center of the spandrel beams. Now suppose we wanted to repeat the example in Figure 32-8 except this time assume that the deck overhangs the beams slightly.

Figure 32-11a shows the example with the slab overhanging the spandrel beams on two sides. Figure 32-11b shows how the deck is automatically meshed into areas A through K. (See Chapter 30 for discussion of ETABS automatic meshing of decks). Note that the example in Figure 32-8 only had areas A through F. Thus areas G through K are added in this new example.

The vertical load transformation from the object-based ETABS model to the analysis model for areas A through F is the same as those described for the example in Figure 32-8. See that example for more information on areas A through F. The vertical load transformation in areas G through K is similar to that described for the example in Figure 32-10:

- The load in area G is transformed to Beam 1a as a distributed load.

- The load in area H is transformed to Beam 1b as a distributed load.
- The load in area I is transformed to the end of Beam 1b as a point load.
- The load in area J is transformed to Beam 3b as a distributed load.
- The load in area K is transformed to Beam 3a as a distributed load.

The Effect of Deck Openings on Load Transformation

32

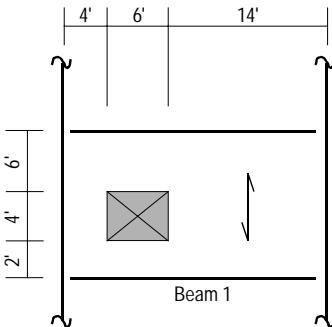
An opening can be either loaded or unloaded. If openings in floors with deck sections are unframed, then for the purposes of transforming loads from the object-based ETABS model to the analysis model, they are treated as if the deck extends right through the opening, that is, in affect, as if the opening is not there. If the opening is framed by beams then the transformation of loads is treated in the usual manner.

Consider the example shown in Figure 32-12. Figure 32-12a illustrates the unframed opening. Figure 32-12b illustrates the framed opening with beams on all sides. The vertical load on the floor is 100 psf. The direction of the deck span is shown. The transformation of the load to the beam labeled Beam 1 in the plan views of Figure 32-12 is illustrated in Figures 32-12c, d, e and f for different conditions. The opening is considered both loaded and unloaded (separately). The opening is also considered to be framed and unframed (separately).

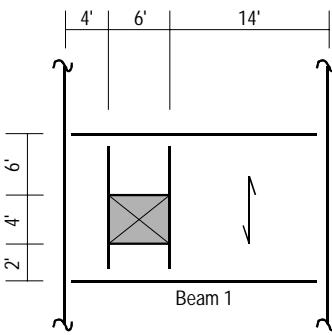
The loading shown on Beam 1 is only that load tributary to Beam 1 from the side of the beam where the opening exists. In Figure 32-12c, the 0.6 klf is from six feet of tributary deck width times 100 psf (0.1 ksf). Noting that in this case the opening is unloaded, the 0.2 klf is from two feet of tributary deck width between Beam 1 and the edge of the opening.

In Figure 32-12d the 0.6 klf is from six feet of tributary deck width. Thus in this example, all of the load on the loaded opening is tributary to Beam 1. This happens because the opening falls within the six-foot tributary width to Beam 1.

Figure 32-12:
Example of effect of
openings on distribution of load over
deck sections

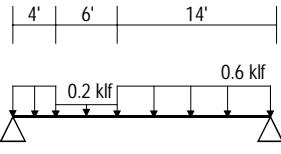


a) Floor Plan with Unframed Opening

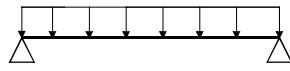


b) Floor Plan with Framed Opening
(Beams on all Sides)

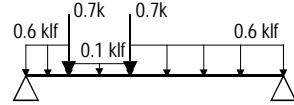
Note: Assume floor loading is 100 psf. Opening is either loaded or unloaded as noted in c, d, e and f which are loading diagrams for Beam 1.



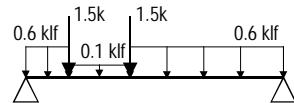
c) Unframed, unloaded opening



d) Unframed, loaded opening



e) Framed, unloaded opening



f) Framed, loaded opening

32

In Figures 32-12e and 32-12f the 0.1 klf is from one foot of tributary deck width between Beam 1 and the edge of the opening. The other half of this two-foot width is tributary to the beam framing the opening. Note that in Figure 32-12e the lower beam framing the opening has a one-foot tributary width whereas in Figure 32-12f, where the opening is loaded it has a three-foot tributary width, two feet of which are from the loaded opening. Thus the point loads on Beam 1 from the framing around the opening are larger in Figure 32-12f.

Vertical Load Transformation for Floors with Membrane Slab Properties

The discussion in this section only applies to floor-type (horizontal) area objects with slab section properties that have membrane behavior only (not plate bending or shell behavior). This discussion further only applies to out-of-plane (vertical) loads acting on these slab sections.

This section describes the transformation of point, line and uniform loads acting on floors with membrane slab sections from your object-based ETABS model to the element-based analysis model. Note that the load transformation occurs after any automatic meshing done by ETABS into the analysis model. See Chapter 30 for discussion of the ETABS automatic meshing for membrane floors.

In ETABS the load distribution for membrane slab sections is two way, that is, the slab is assumed to span in two perpendicular directions. This is in contrast to deck sections that are assumed to span in one direction only. The transformation of loads from the object-based model to the analysis model considers these two span directions for floors modeled with membrane slab properties.

Before the load transformation is done for membrane slabs ETABS first automatically meshes the slab into quadrilateral elements as described in Chapter 30 (unless you have already completely meshed it yourself either by drawing it that way or by using the manual meshing methods described in Chapter 31). Once the meshing is complete ETABS knows which sides of the meshed shell elements have real beams along them and which corner points of the shell element have vertical support. Armed with this information ETABS can begin to transform the load into the analysis model.

The actual distribution of loads on these elements is quite complex. ETABS uses the concept of tributary loads as a simplifying assumption for transforming the loads.

The shell element (area object) is divided into areas supported by the edges that have real beams along them and by the corner points that have actual vertical support (e.g., column, wall, etc.). **All out-of-plane loads** (applied as point, line or area loads) that fall within these tributary areas are then transferred to the respective edges (beams) or corner points (vertical support member).

The shell element is typically divided into tributary areas based on the following rules applied in the order that the rules are listed:

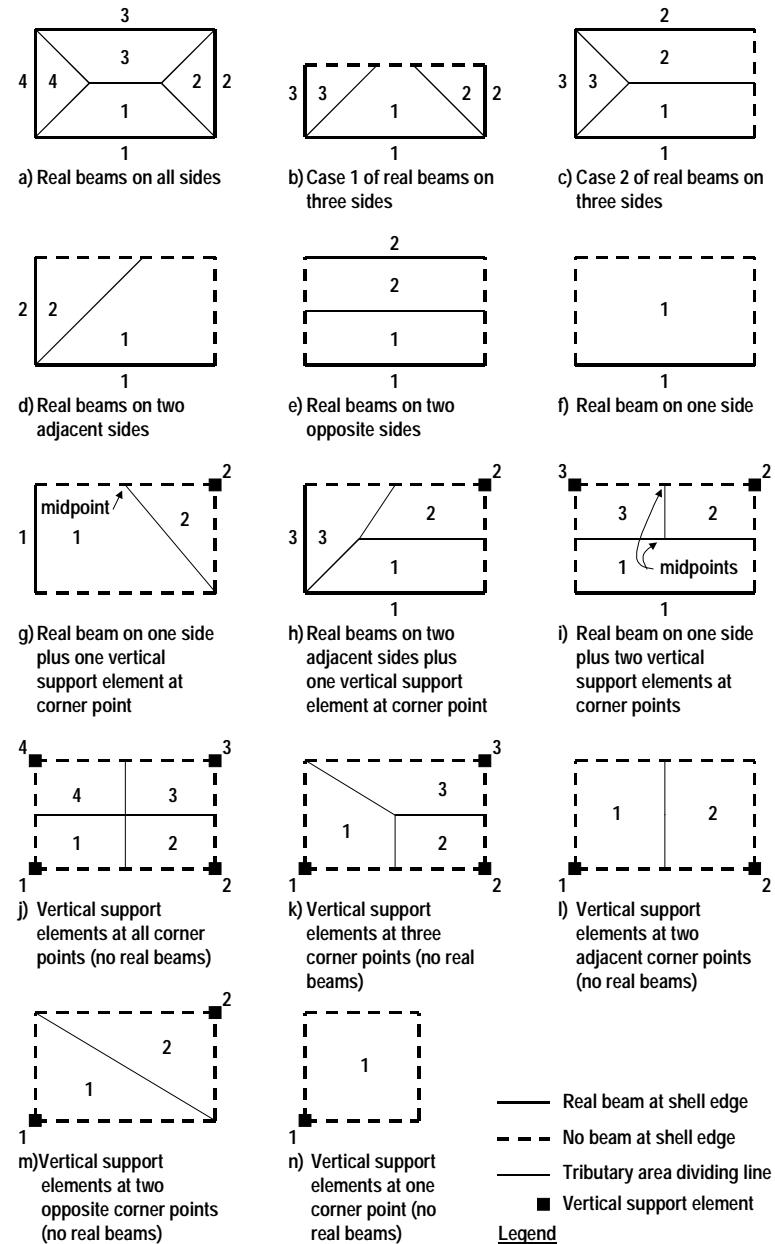
- Lines are drawn that bisect the corner angle of every corner point of the shell where both adjacent edges are supported by real beams.
- Lines are drawn from the midpoint of the edges of the shell element that do not have real beams along them but do have both of their ends supported. These lines are either drawn in a direction such that they would connect to the midpoint of the opposite edge or they are drawn to an appropriate intersection point with other tributary area dividing lines.
- Special case tributary area dividing lines are drawn.

Figure 32-13 shows some of the possibilities for the tributary areas. Each of the items in this figure shows a four-sided shell element. The numbers on the outside of each item correspond to the supporting members for the shell element. These supporting members are either real beams or vertical support elements. The numbers on the inside of the shell element identify the tributary area for the like-numbered supporting element.

The bullet list below discusses each of the items in Figure 32-13.

- Figure 32-13a has real beams on all four sides. The four corner angles are bisected such that pairs of lines from adjacent corners intersect. Then a special line is drawn connecting the two intersection points.

Figure 32-13:
Tributary areas for
various conditions of
a membrane slab

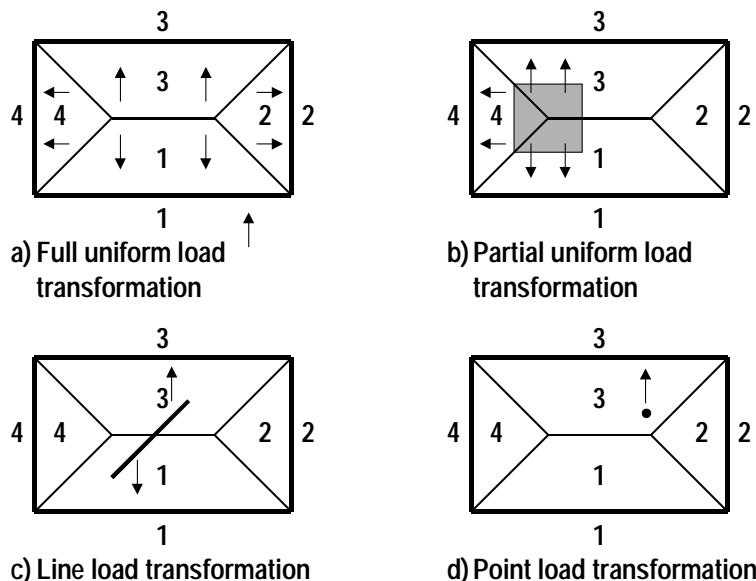


- Figure 32-13b has real beams on three sides. In the case shown here bisecting lines are drawn bisecting the angles at the two corners where real beams meet. These bisecting lines meet the opposite edge of the shell element before they intersect.
- Figure 32-13c has real beams on three sides. In the case shown here bisecting lines are drawn bisecting the angles at the two corners where real beams meet. These bisecting lines intersect before they meet the opposite edge of the shell element. A line is drawn from the midpoint of the edge with no real beam that has both ends supported (by beams) to the intersection point of the lines that bisected the angles.
- Figure 32-13d has real beams on two adjacent sides. A line is drawn bisecting the angle at the corner where the two real beams meet. This line is extended to the opposite edge of the shell element.
- Figure 32-13e has real beams on two opposite sides. A line is drawn connecting the midpoints of the edges with no real beams along them.
- Figure 32-13f has real beams on one side. No tributary area dividing lines are needed. All loads go to the one beam.
- Figure 32-13g has a real beam on one side and one vertical support element. A line is drawn from the midpoint of the edge with no beam along it, but with supports at each end (beam at one end, vertical support element at the other) to the corner with no beam framing into it and no vertical support. Note that in general, this line does *not* bisect the angle at this corner.
- Figure 32-13h has real beam on two sides and one vertical support element. A line is drawn bisecting the angle at the corner point where the real beams meet. Lines are drawn perpendicular from the midpoint of each of the edges with no beam along it until they intersect the line bisecting the angle. The perpendicular line that intersects the line bisecting the angle closest to the corner point

where the real beams meet is kept. The other perpendicular line is realigned such that it connects the midpoint of the edge with no beam along it to the intersection point of the other two lines.

- Figure 32-13i has a real beam on one side and two vertical support elements. A line is drawn connecting the midpoints of the two sides that are adjacent to the side with the real beam along it. A second line is drawn connecting the midpoint of the side opposite the side with the real beam to the midpoint of the first line drawn.
- Figure 32-13j has vertical support at all four corner points and no real beams along the edges. Lines are drawn connecting the midpoints of opposite sides.
- Figure 32-13k has vertical support at three corner points and no real beams along the edges. Lines are first drawn connecting the midpoints of opposite sides. These two lines are then scaled back such that they extend from their intersection point to the side of the element that has vertical support at each end. Finally a line is drawn from the intersection point to the corner with no vertical support. Note that in general, this line does *not* bisect the angle at this corner.
- Figure 32-13l has vertical support at two *adjacent* corner points and no real beams along the edges. A line is drawn connecting the midpoints of opposite sides of the element.
- Figure 32-13m has vertical support at two *opposite* corner points and no real beams along the edges. A line is drawn between the corner points that do not have vertical support.
- Figure 32-13n has vertical support at one corner point and no real beams along the edges. No tributary area dividing lines are needed. All loads go to the one corner point.

Figure 32-14:
Example of load distribution on a membrane slab



Note that the load transformations shown in Figures 32-13 a, b, c, f, l and m do a fairly good job of approximating reality. In these cases a reasonable estimate of the "correct" amount of load is distributed to each supporting element.

The other items in Figure 32-13 are convenient ways to transform the load but may not always do a good job of approximating the real condition. They will always transform the correct total amount of load but may not always distribute the "correct" proportions to each supporting element. If you find you have a lot of these conditions and a "correct" load distribution on each supporting element is important in your model then you may be better off considering plate bending in your floor and manually meshing the slab appropriately. See Chapter 31 for discussion of manual meshing.

Figure 32-14 shows some examples of load transformation for membrane slab shell elements with real beams on all four sides. In the figure the sides of the shell element are numbered 1 through 4 and the corresponding tributary area of the shell element are also numbered 1 through 4.

Figure 32-14a shows how uniform load over the entire shell element is transformed to the real beams along the shell edges. The

load in areas 1 through 4 is projected onto beams 1 through 4, respectively, as full length distributed line loads.

Figure 32-14b shows how a partial uniform area load on the shell element (shown shaded) is transformed to the real beams along the shell edges. The load in areas 1, 3 and 4 is projected onto beams 1, 3 and 4, respectively, as partial length line loads. Beam 2 receives no load.

Figure 32-14c shows how a line load on the shell element is transformed to the real beams along the shell edges. The portion of the line load in area 1 is projected onto beam 1 as a distributed line load. Similarly, the portion of the line load in area 3 is projected onto beam 3 as a distributed line load. Beams 2 and 4 receive no load.

Figure 32-14d shows how a point load on the shell element is transformed to the real beams along the shell edges. Since the point load falls in area 3, the entire point load is projected onto beam 3. Beams 1, 2 and 4 receive no load.

Note the following about projection of the shell element loads onto the edge beams:

- The projection of load is always done in a direction that is perpendicular to the considered edge beam.
- If the projection in the above-specified manner causes some of the load to be projected beyond the end of the edge beam then that load is concentrated at the end of the edge beam.

Overview of ETABS Analysis Techniques

General

This chapter provides a brief overview of ETABS analysis techniques. The types of analysis discussed are linear static analysis, modal (eigenvector and Ritz-vector) analysis, response-spectrum analysis, time-history analysis, initial P-Delta analysis, and non-linear static analysis (including incremental analysis.)

In a given analysis run, you may request an initial P-Delta analysis, a modal analysis, and multiple cases of linear static, response-spectrum, and time-history analysis. Multiple nonlinear static analysis cases may also be defined; these are performed separately from the other analysis types.

Linear Static Analysis

A linear static analysis is automatically performed for each static load case that is defined. The results of different static load cases can be combined with each other and with other linear analysis cases, such as response-spectrum analyses.

Geometric and material nonlinearity are not considered in linear static analysis, except that the effect of the initial P-Delta analysis is included in every static load case. For example, if you define an initial P-Delta analysis for gravity load, then deflections and moments will be increased for lateral static load cases.

Linear static load cases can still be combined when an initial P-Delta analysis has been performed, since the initial P-Delta load is the same for all static load and response-spectrum cases.

33

Modal Analysis

Modal analysis calculates vibration modes for the structure. These can be used to investigate the behavior of a structure, and are required as a basis for subsequent response-spectrum and/or time-history analyses.

Note:

Two types of modal analysis are available: eigenvector analysis and Ritz-vector analysis.

Two types of modal analysis are available: eigenvector analysis and Ritz-vector analysis. Only one of these can be used in a single analysis run.

Eigenvector analysis determines the undamped free-vibration mode shapes and frequencies of the system. These natural Modes provide an excellent insight into the behavior of the structure. They can also be used as the basis for response-spectrum or time-history analyses, although Ritz vectors (discussed below) are *strongly recommended* for this purpose.

ETABS modal analysis can also generate a special set of load-dependent Ritz vectors that take into account the spatial distribution of dynamic load. Although these Ritz vectors may not always correspond to the natural vibration modes of the structure, it has been shown that they are a much better basis for response-spectrum and time-history analysis.

It is *especially important* that you use Ritz-vectors when performing nonlinear time-history analysis.

An initial P-Delta analysis will affect all the modes found. For example, if you define an initial P-Delta analysis using gravity load, the periods of the lateral modes will usually be longer than without the P-Delta effect.

Eigenvector Analysis

Eigenvector analysis involves the solution of the generalized eigenvalue problem shown in Equation 33-1:

$$[\mathbf{K} - \Omega^2 \mathbf{M}] \boldsymbol{\Phi} = \mathbf{0} \quad \text{Eqn. 33-1}$$

where \mathbf{K} is the stiffness matrix, \mathbf{M} is the diagonal mass matrix, Ω^2 is the diagonal matrix of eigenvalues, and $\boldsymbol{\Phi}$ is the matrix of corresponding eigenvectors (mode shapes).



Tip:

It is especially important that you use Ritz-vectors when performing nonlinear time-history analysis.

Each eigenvalue-eigenvector pair is called a natural vibration mode of the structure. The modes are identified by numbers from 1 to n in the order in which the modes are found by the program.

The eigenvalue is the square of the circular frequency, ω , for that mode (unless a frequency shift is used, see below). The cyclic frequency, f , and the period, T , of the mode are related to ω by Equation 33-2:

$$T = \frac{1}{f} \quad \text{and} \quad f = \frac{\omega}{2\pi} \quad \text{Eqn. 33-2}$$

You may specify the number of modes to be found, a convergence tolerance, and the frequency range of interest. These parameters are described in the following subtopics.

Number of Modes

You may specify the number of modes, N , to be found. The program will seek the N lowest-frequency (longest-period) modes. If a non-zero frequency shift has been specified, the program will seek the N modes with frequencies closest to the shift, f_0 . This is discussed in the next subsection.

The number of modes actually found, n , is limited by:

- The number of modes requested, N .
- The number of modes present in the specified frequency range. See the subsection titled "Frequency Range" later in this chapter.
- The number of mass degrees of freedom in the model.

A mass degree of freedom is any *active* degree of freedom that possesses translational mass or rotational mass moment of inertia. The mass may have been assigned directly to the joint or may come from connected objects/elements.

Only the modes that are actually found will be available for any subsequent response-spectrum or time-history analysis processing.

Frequency Range

You may specify a restricted frequency range in which to seek the eigen-modes by using the parameters:

f_0 = the center of the cyclic frequency range, known as the shift frequency.

f_{\max} = the radius of the cyclic frequency range, known as the cutoff frequency.

The program will only seek modes with frequencies f that satisfy Equation 33-3:

$$|f - f_0| \leq f_{\max}$$

Eqn. 33-3

The default value of $f_{\max} = 0$ does not restrict the frequency range of the modes.

Modes are found in order of increasing distance of frequency from the shift. This continues until the cutoff is reached, the requested number of modes is found, or the number of mass degrees of freedom is reached.

A stable structure will possess all positive natural frequencies. When performing a seismic analysis and most other dynamic analyses, the lower-frequency modes are usually of most interest. It is then appropriate to use the default shift of zero, resulting in the lowest-frequency modes of the structure being calculated. If the shift is not zero, response-spectrum and time-history analyses may be performed; however, linear static and initial P-Delta analyses are not allowed.

If the dynamic loading is known to be of high frequency, such as that caused by vibrating machinery, it may be most efficient to use a positive shift near the center of the frequency range of the loading.

A structure that is unstable when unloaded will have some modes with zero frequency. These modes may correspond to rigid-body motion of an inadequately supported structure, or to mechanisms that may be present within the structure. It is not possible to compute the static response of such a structure. However, by using a small negative shift, the lowest-frequency vibration modes of the structure, including the zero-frequency instability modes, can be found. This does require some mass to be present that is activated by each instability mode. Thus the frequency shift may occasionally be helpful in debugging an unstable structure.

A structure that has buckled under P-Delta load will have some modes with zero or negative frequency. During equation solution, the number of frequencies less than the shift is determined and printed in the log file. If you are using a zero or negative shift and the program detects a negative-frequency mode, it will stop the analysis since the results will be meaningless. If you use a positive shift, the program will permit negative frequencies to be found; however, subsequent static and dynamic results are still meaningless.

When using a frequency shift, the stiffness matrix is modified by subtracting from it the mass matrix multiplied by ω_0^2 , where $\omega_0 = 2\pi f_0$. If the shift is very near a natural frequency of the structure, the solution becomes unstable and will be halted during equation solution. Run the analysis again using a slightly different shift frequency.

The circular frequency, ω , of a mode is determined from the shifted eigenvalue, μ , as shown in Equation 33-4:

$$\omega = \sqrt{\mu + \omega_0^2} \quad \text{Eqn. 33-4}$$

Convergence Tolerance

ETABS solves for the eigenvalue-eigenvectors pairs using an accelerated subspace iteration algorithm. During the solution phase, the program prints the approximate eigenvalues after each iteration. As the eigenvectors converge they are removed from the subspace and new approximate vectors are introduced. For details of the algorithm, see Wilson and Tetsuji (1983).

You may specify the relative convergence tolerance, ε , to control the solution; the default value is $\varepsilon = 10^{-6}$. The iteration for a particular mode will continue until the relative change in the eigenvalue between successive iterations is less than 2ε , i.e., until:

$$\frac{1}{2} \left| \frac{\mu_{i+1} - \mu_i}{\mu_{i+1}} \right| \leq \varepsilon \quad \text{Eqn. 33-5}$$

where μ is the eigenvalue relative to the frequency shift, and i and $i+1$ denote successive iteration numbers.

In the usual case where the frequency shift is zero, the test for convergence becomes approximately the same as:

$$\left| \frac{T_{i+1} - T_i}{T_{i+1}} \right| \leq \varepsilon \quad \text{or} \quad \left| \frac{f_{i+1} - f_i}{f_{i+1}} \right| \leq \varepsilon \quad \text{Eqn. 33-6}$$

provided that the difference between the two iterations is small.

Note that the error in the eigenvectors will generally be larger than the error in the eigenvalues. The relative error in the global force balance for a given mode gives a measure of the error in the eigenvector. This error can usually be reduced by using a smaller value of convergence tolerance, at the expense of more computation time.

33

Residual Mass Modes

Note:

In most analyses you would probably not consider residual mass modes. However, an example where you might consider using residual mass modes is in a building with very stiff shear walls at the base where you do not pick up 100% (or close to it) mass participation in your modes.

Residual-mass (missing-mass) modes may be calculated as an option for eigen-analysis. The purpose is to try to approximate high-frequency behavior when the mass participation ratio for a given direction of acceleration load is less than 100%.

For a given acceleration load, say \mathbf{m}_{ux} (see the section titled Acceleration Loads later in this chapter), the missing mass load is that portion of the load which cannot be represented by the eigen modes that have been found. The static displacement corresponding to this missing-mass load is the missing mass mode. One missing mass mode can be calculated for each direction of translational acceleration. If the cumulative mass participation ratio for a given direction of acceleration is 100%, then there is no missing mass and the missing mass mode is not calculated.

A period is assigned to each missing-mass mode that is calculated by the standard Rayleigh quotient method. Typically this period will be some average of all the missing eigen-modes that are excited by the acceleration loads. A mass participation ratio will be computed for the missing-mass modes. In general, the cumulative mass participation ratios, including the missing-mass modes, will still not be 100%, since the missing-mass modes are only static approximations to the high-frequency response, not the true dynamic modes.

When present, residual-mass modes are automatically included in Response-spectrum and Time-history analyses. For Ritz analysis, residual mass modes are always included automatically for all starting load vectors.

Ritz-Vector Analysis

Research has indicated that the natural free-vibration mode shapes are not the best basis for a mode-superposition analysis of structures subjected to dynamic loads. It has been demonstrated (Wilson, Yuan, and Dickens, 1982) that dynamic analyses based on load-dependent Ritz vectors yield more accurate results than the use of the same number of natural mode shapes.

The reason the Ritz vectors yield excellent results is that they are generated by taking into account the spatial distribution of the dynamic loading, whereas the direct use of the natural mode shapes neglects this very important information.

Note:

It has been demonstrated that dynamic analyses based on load-dependent ritz-vectors yield more accurate results than the use of the same number of natural mode shapes.

In addition, the Ritz-vector algorithm automatically includes the advantages of the proven numerical techniques of static condensation, Guyan reduction, and static correction due to higher-mode truncation.

The spatial distribution of the dynamic load vector serves as a **starting load vector** to initiate the procedure. The first Ritz vector is the static displacement vector corresponding to the starting load vector. The remaining vectors are generated from a recurrence relationship in which the mass matrix is multiplied by the previously obtained Ritz vector and used as the load vector for the next static solution. Each static solution is called a **generation cycle**.

When the dynamic load is made up of several independent spatial distributions, each of these may serve as a starting load vector to generate a set of Ritz vectors. Each generation cycle creates as many Ritz vectors as there are starting load vectors. If a generated Ritz vector is redundant or does not excite any mass degrees of freedom, it is discarded and the corresponding starting load vector is removed from all subsequent generation cycles.

Standard eigen-solution techniques are used to orthogonalize the set of generated Ritz vectors, resulting in a final set of Ritz-vector modes. Each Ritz-vector mode consists of a mode shape and frequency. The full set of Ritz-vector modes can be used as a basis to represent the dynamic displacement of the structure.

When a sufficient number of Ritz-vector Modes have been found, some of them may closely approximate natural mode shapes and frequencies. In general, however, Ritz-vector Modes do not represent the intrinsic characteristics of the structure in the same way the natural Modes do. The Ritz-vector modes are biased by the starting load vectors.

You may specify the number of modes to be found and the starting load vectors to be used. These parameters are described in the following subtopics.

Number of Modes

You may specify the total number of modes, N , to be found. The total number of modes actually found, n , is limited by:

- The number of modes requested, N .
- The number of mass degrees of freedom present in the model.
- The number of natural free-vibration modes that are excited by the starting load vectors (some additional natural modes may creep in due to numerical noise).

A mass degree of freedom is any *active* degree of freedom that possesses translational mass or rotational mass moment of inertia. The mass may have been assigned directly to the joint or may come from connected elements.

Only the modes that are actually found will be available for any subsequent response-spectrum or time-history analysis processing.

Starting Load Vectors

You may specify any number of starting load vectors. Each starting load vector may be one of the following:

- An Acceleration Load in the global X, Y, or Z direction.
- A static load case.
- A built-in nonlinear deformation load, as described below.

For response-spectrum analysis, only the acceleration loads are needed. For time-history analysis, one starting load vector is needed for each load case or acceleration load that is used in any time-history case.

If nonlinear time-history analysis is to be performed, an additional starting load vector is needed for *each* independent nonlinear deformation, i.e., each nonlinear degree of freedom in the link elements. You may specify that the program use the built-in nonlinear deformation loads, or you may define your own load cases for this purpose.

If you define your own starting load vectors, do the following for *each* nonlinear deformation:

- Explicitly define a static load case that consists of a set of self-equilibrating forces that activates the desired nonlinear deformation
- Specify that load case as a starting load vector

The number of such load cases required is equal to the number of independent nonlinear deformations in the model.

If several link elements act together, you may be able to use fewer starting load vectors. For example, suppose the horizontal motion of several base isolators are coupled with a diaphragm. Only three starting load vectors acting on the diaphragm are required: two perpendicular horizontal loads and one moment about the vertical axis. Independent load cases may still be required to represent any vertical motions or rotations about the horizontal axes for these isolators.

It is *strongly recommended* that mass (or mass moment of inertia) be present at every degree of freedom that is loaded by a starting load vector. This is automatic for acceleration loads, since the load is caused by mass. If a static load case or nonlinear deformation load acts on a non-mass degree of freedom, the program issues a warning. Such starting load vectors may generate inaccurate Ritz vectors, or even no Ritz vectors at all.

Generally, the more starting load vectors used, the more Ritz vectors must be requested to cover the same frequency range. Thus including unnecessary starting load vectors is not recommended.

In each generation cycle, Ritz vectors are found in the order in which the starting load vectors are specified. In the last generation cycle, only as many Ritz vectors will be found as required to reach the total number of modes, N . For this reason, the most important starting load vectors should be specified first, especially if the number of starting load vectors is not much smaller than the total number of modes.

Acceleration Loads

The program automatically computes six acceleration loads that act on the structure, three due to unit translational accelerations in each of the three global directions, and three due to unit rotational accelerations about the global axes at the global origin. The loads are determined by d'Alembert's principal, and are denoted \mathbf{m}_{ux} , \mathbf{m}_{uy} , \mathbf{m}_{uz} , \mathbf{m}_{rx} , \mathbf{m}_{ry} , and \mathbf{m}_{rz} , respectively.

The translational loads are used for applying ground accelerations in response-spectrum and time-history analyses, and can be used as starting load vectors for Ritz-vector analysis. The translational and rotational loads are used for calculating modal participation measures. See the section titled "Building Modal Info" in Chapter 41 for more information.

**Note:**

The translational Acceleration loads m_{ux} , m_{uy} and m_{uz} are used for applying ground accelerations in response-spectrum and time-history analyses, and can be used as starting load vectors for Ritz-vector analysis.

33

The applied ground acceleration is assumed to be uniform, hence it is the same at the base of each restraint, spring, or grounded link element. The response-spectrum and time-history displacements resulting from acceleration loads are always *relative* to the ground motion.

The acceleration loads are computed for each joint and element and summed over the whole structure. The translational acceleration loads for the joints are simply equal to the negative of the joint translational masses in the joint local coordinate system. These loads are transformed to the global coordinate system. The acceleration loads for the elements are the same in each direction and are equal to the negative of the element masses.

Rotational accelerations cause rotational loads at each joint equal to the negative of the rotational inertia at that joint. Translational loads are also created at the joints and are equal to the negative of the translational mass times the translational acceleration at the joint caused by rotation about the origin.

The acceleration loads can be transformed into any coordinate system. In the global system, the acceleration loads along the positive X, Y, and Z axes are denoted UX, UY, UZ, RX, RY, and RZ, respectively. In a local coordinate system defined for a response-spectrum or time-history analysis, the acceleration loads along the positive local 1, 2, and 3 axes are denoted U1, U2, U3, R1, R2, and R3, respectively.

Response Spectrum Analysis

The dynamic equilibrium equations associated with the response of a structure to ground motion are given by Equation 33-7:

$$\mathbf{K}\mathbf{u}(t) + \mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{M}\ddot{\mathbf{u}}(t) = \mathbf{m}_x\ddot{u}_{gx}(t) + \mathbf{m}_y\ddot{u}_{gy}(t) + \mathbf{m}_z\ddot{u}_{gz}(t) \quad \text{Eqn. 33-7}$$

where \mathbf{K} is the stiffness matrix; \mathbf{C} is the damping matrix; \mathbf{M} is the diagonal mass matrix; \mathbf{u} , $\dot{\mathbf{u}}$, and $\ddot{\mathbf{u}}$ are the relative displacements, velocities, and accelerations of the structure with respect to the ground; \mathbf{m}_x , \mathbf{m}_y , and \mathbf{m}_z are the unit acceleration

loads in the three directions; and \ddot{u}_{gx} , \ddot{u}_{gy} , and \ddot{u}_{gz} are the components of uniform ground acceleration as functions of time.

Response-spectrum analysis seeks the likely maximum response to these equations rather than the full time history. The earthquake ground acceleration in each direction is given as a digitized response-spectrum curve of pseudo-spectral acceleration response versus period of the structure.

Even though accelerations may be specified in three directions, only a single, positive result is produced for each response quantity. The response quantities may be displacements, forces, or stresses. Each computed result represents a statistical measure of the likely maximum magnitude for that response quantity. The actual response can be expected to vary within a range from this positive value to its negative.

No correspondence between two different response quantities is available. No information is available as to when this extreme value occurs during the seismic loading, or as to what the values of other response quantities are at that time.

Response-spectrum analysis is performed using mode superposition (Wilson and Button, 1982). Modes may have been computed using eigenvector analysis or Ritz-vector analysis. Ritz vectors are recommended since they give more accurate results for the same number of modes.

The effect of an initial P-Delta analysis is automatically included in each response-spectrum case since it is included in all of the modes.

See the section titled "Response Spectrum Cases" in Chapter 11 for more information.

Time History Analysis

Time-history analysis is used to determine the dynamic response of a structure to arbitrary loading. The dynamic equilibrium equations to be solved are given by Equation 33-8:

$$\mathbf{K}\mathbf{u}(t) + \mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{M}\ddot{\mathbf{u}}(t) = \mathbf{r}(t) \quad \text{Eqn. 33-8}$$

where \mathbf{K} is the stiffness matrix; \mathbf{C} is the damping matrix; \mathbf{M} is the diagonal mass matrix; \mathbf{u} , $\dot{\mathbf{u}}$, and $\ddot{\mathbf{u}}$ are the displacements, velocities, and accelerations of the structure; and \mathbf{r} is the applied load. If the load includes ground acceleration, the displacements, velocities, and accelerations are relative to this ground motion.

Any number of time-history cases can be performed in a single execution of the program. Each case can differ in the load applied and in the type of analysis to be performed.

Three types of time-history analysis are available:

- **Linear transient:** The structure starts with zero initial conditions or with the conditions at the end of a previous linear transient time-history case that you specify. All elements are assumed to behave linearly for the duration of the analysis.
- **Periodic:** The initial conditions are adjusted to be equal to those at the end of the period of analysis. All elements are assumed to behave linearly for the duration of the analysis.
- **Nonlinear transient:** The structure starts with zero initial conditions or with the conditions at the end of a previous nonlinear transient time-history case that you specify. The link elements may exhibit nonlinear behavior during the analysis. All other elements behave linearly.

The effect of an initial P-Delta analysis is automatically included in each time-history case since it is included in all of the modes used to solve the time-history case.

Mode Superposition

The standard mode-superposition method of response analysis is used by the program to solve the dynamic equilibrium equations of motion for the complete structure. The modes used can be the eigenvector or the load-dependent Ritz-vector modes.

If all of the time-history spatial load vectors are used as starting load vectors for Ritz-vector analysis, then the Ritz vectors will always produce more accurate results than if the same number of eigenvectors is used.

It is especially important to use Ritz vectors when performing nonlinear time-history analysis. In this case, be sure to include the nonlinear link deformation loads as starting load vectors and substantially increase the number of modes requested.

It is up to you to determine if you are using enough modes to adequately represent the time-history response to the applied load. You should check the load participation ratios for the applied loads and the global force balance for each mode.

Ultimately, the best check is to examine the displacements, forces, and stresses obtained with increasing numbers of modes until convergence is obtained. Global quantities, such as roof displacement and base shear will usually converge with a few modes. Local, “stiff” quantities, such as forces near vibrating machinery, or the forces near a nonlinear gap or damper element, may require many more modes to converge.

Modal Damping

The damping in the structure is modeled using modal damping, also known as proportional or classical damping. The damping in mode i may be specified using the parameter ξ_i , which is measured as a fraction of critical damping. The damping value for each mode must satisfy Equation 33-9:

$$0 \leq \xi_i < 1$$

Eqn. 33-9

For a nonlinear transient analysis, this is the only source of modal damping. See the subsection titled “Nonlinear Time-History Analysis” below for important considerations about using modal damping with nonlinear transient analysis.

For linear transient and periodic analyses, additional modal damping may come from effective-damping coefficients that have been specified for link elements in the model. These effective damping values are converted to modal damping ratios assuming proportional damping, i.e., by ignoring any modal cross-coupling terms. These effective modal-damping values will generally be different for each mode, depending upon how much deformation each mode causes in the link elements.

Only effective damping, not the actual damping in the link elements, affects the linear time-history results. The total damping ratio for each mode is the sum of these two sources. The program automatically makes sure that the total is less than one.

Time Steps

For linear transient and periodic analysis, closed-form integration of the modal equations is used to compute the response, assuming linear variation of the time functions between the input data time points. Therefore, numerical instability problems are never encountered, and the time increment may be any sampling value that is deemed fine enough to capture the maximum response values. One-tenth of the time period of the highest mode is usually recommended; however, a larger value may give an equally accurate sampling if the contribution of the higher modes is small.

For nonlinear transient analysis, closed-form integration is again used to compute the response, with the forces from the nonlinear link elements applied to the right-hand side of the dynamic modal equilibrium equations. Since these forces depend upon the dynamic response, the solution is iterated at each time step until the nonlinear forces converge. The accuracy of the results may depend upon the size of the time step, improving with decreasing time step. See subtopic “Nonlinear Time-History Analysis” below for more information.

Nonlinear Time-History Analysis

The method of nonlinear time-history analysis used in ETABS is an extension of the Fast Nonlinear Analysis (FNA) method developed by Wilson (Ibrahimbegovic and Wilson, 1989; Wilson, 1993). The method is extremely efficient and is designed to be used for structural systems which are primarily linear elastic, but which have a limited number of predefined nonlinear elements. In ETABS, all nonlinearity is restricted to the nonlinear link elements.

The FNA method uses modal analysis with iteration to correct for the coupling between modes due to the nonlinear elements. This method is highly accurate when used with appropriate Ritz-vector modes, and has advantages over traditional time-stepping methods in terms of speed and better control over damping and higher-mode effects.

In order to get the best results, the following points should be kept in mind:

- It is strongly recommended that you use Ritz vectors rather than eigenvectors for the modal analysis.
- The Ritz starting load vectors should include all the static loads and/or acceleration loads used to define the time-history cases, plus the nonlinear deformation loads from the link elements.
- The total number of modes requested should be *at least* one or two modes per nonlinear degree of freedom, plus the number of modes that you would normally use for a similar linear time-history analysis. Check that you have enough modes by looking at your results using increasing numbers of modes.
- The effective stiffness properties for the nonlinear link elements should not affect the analysis results, except as described below for modal damping. Effective stiffness values may affect the efficiency of nonlinear time-history analysis. As a guide, choose the effective stiffness to be equal to the actual stiffness, except use zero or

small effective stiffness for dampers, and for gaps or hooks that are normally open.

- Modal damping is computed with respect to the stiffness matrix used for the modal analysis, which includes the effective stiffness from the nonlinear elements. If non-zero modal damping is to be used, then the effective stiffness specified for these elements is important. Large values of effective stiffness may result in unrealistically large values of energy dissipation. For more realistic modeling, use small values of modal damping and use explicit nonlinear dampers or plasticity elements to model energy dissipation.
- The effective damping properties for the nonlinear link elements are not used for nonlinear time-history analysis.
- Mass and/or mass moments of inertia should be present at all nonlinear degrees of freedom.

The program uses an automatic sub-stepping algorithm to choose the best time steps for integrating the nonlinear modal equations of motion. In general, the results you obtain should be insensitive to the size of the output time step that you choose. However, using large time steps may make it difficult to see short-duration behavior, such as the closing of gaps, even though the solution does adequately account for such behavior.

Initial P-Delta Analysis

The initial P-Delta analysis option accounts for the effect of a large compressive or tensile load upon the transverse stiffness of members in the structure. Compression reduces lateral stiffness, and tension stiffens it. This is a type of geometric nonlinearity known as the P-Delta effect. Initial P-Delta analysis does not include large-strain or large-rotation effects.

**Note:**

The P-Delta option is useful for considering the effect of gravity loads upon the lateral stiffness of building structures, as required by certain design codes

This option is particularly useful for considering the effect of gravity loads upon the lateral stiffness of building structures, as required by certain design codes (ACI 1999; AISC 1994). Other applications are possible.

Initial P-Delta analysis in ETABS considers the P-Delta effect of a single loaded state upon the structure. There are two ways to specify this load:

- As a specified combination of static load cases; this is called the P-Delta load combination. For example, this 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.
- As a story-by-story load upon the structure computed automatically from the mass at each level. This approach is approximate, but does not require an iterative solution.

When you request initial P-Delta analysis, it is performed before all linear-static, modal, response-spectrum, and time-history analyses in the same analysis run. The initial P-Delta analysis essentially modifies the characteristics of the structure, affecting the results of all subsequent analyses performed.

As an important exception, initial P-Delta analysis does NOT affect nonlinear-static analysis. Nonlinear static analyses consider the P-Delta effect separately, if requested.

Because the load causing the P-Delta effect is the same for all linear analysis cases, their results may be superposed in load combinations.

Initial P-Delta analysis under a P-delta load combination is iterative in nature, and may considerably increase computation time. Including initial P-Delta analysis may make interpretation of the results more difficult. It is strongly recommended that you perform a preliminary linear analysis to check your model for correctness before using initial P-Delta analysis.

Iterative Solution

When you specify a P-Delta load combination, the following parameters may also be specified to control the iterative solution:

Note:

P-Delta analysis based on specified load cases is an iterative analysis. It may take several iterations to achieve convergence.

- **Maximum Number of Iterations:** This is used to prevent excessive computational time, since 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, then 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, then the results of the analysis may be meaningless, and they should be viewed with great skepticism. Failure to converge may be due to 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.
- The structure is near buckling. The structure should be stiffened against buckling, or the magnitude of the P-Delta load combination reduced.

Buckling

If compressive P-Delta axial forces are present and are large enough, the structure may buckle. Local buckling of individual members or global buckling of the whole structure are both possible. The program makes no distinction between local and global buckling.

If the program detects that buckling has occurred, the analysis is terminated and no results are produced. This is because the analysis of a structure that has buckled requires consideration of large-displacement effects that are only considered during non-linear static analysis.

Initial P-Delta analysis may be used to estimate buckling loads by performing a series of analyses, each time increasing the magnitude of the P-Delta load combination, until buckling is detected. The relative contributions from each static load case to the P-delta load combination must be kept the same, increasing all load case scale factors by the same amount between runs.

It is important to understand that there is no single buckling load for a structure. Rather, there is a different buckling load corresponding to each spatial distribution of loads. If buckling of the structure is a concern under various loading situations, the buckling load should be estimated separately for each situation, as described above, by starting with different P-Delta load combinations.

Practical Application

For most building structures, especially tall buildings, the P-Delta effect of most concern occurs in the columns due to gravity load, including dead and live load. The column axial forces are compressive, making the structure more flexible against lateral loads.

Building codes (ACI 1999; AISC 1994) normally recognize two types of P-Delta effects: the first due to the overall sway of the structure and the second due to the deformation of the member between its ends. The former effect is often significant; it can be accounted for fairly accurately by considering the total vertical load at a story level, which is due to gravity loads and is unaf-

fected by any lateral loads. The latter effect is significant only in very slender columns or columns bent in single curvature (not the usual case); this requires consideration of axial forces in the members due to both gravity and lateral loads.

ETABS can analyze both of these P-Delta effects. However, it is recommended that the former effect be accounted for in the analysis, and the latter effect be accounted for in design by using the applicable building-code moment-magnification factors (White and Hajjar 1991). This is how the design checks in ETABS work.

Nonlinear Static Analysis

Nonlinear static analysis in ETABS offers a wide variety of capabilities, including:

- Material nonlinearity in beams and columns.
- Nonlinear gap, hook, and plasticity behavior in links.
- Geometric nonlinearity, including large deflections and P-Delta effects.
- Incremental construction analysis.
- Static pushover analysis.

Multiple static analysis cases can be defined. For each case, the structure may start with zero initial conditions or with the conditions at the end of a previous nonlinear static case that you specify.

Each analysis case considers a single pattern of loading, specified as a linear combination of static load cases, acceleration loads, and/or vibration mode shapes. Changing patterns of load are modeled by using a sequence of nonlinear static cases.

**Tip:**

You can perform incremental construction analysis using the nonlinear static analysis capability in the nonlinear version of ETABS.

Loads are applied incrementally within an analysis case. The incremental step size is adjusted automatically to account for significant changes in geometry, or for yielding, gapping, and other material nonlinearities. An event-to-event type strategy is used with iteration to correct for equilibrium errors; the stiffness matrix is re-assembled and solved as necessary.

The load pattern may be applied under load or displacement control. Load control is used to apply a known magnitude of load, such as would be required for gravity load. Displacement control applies the load with a variable magnitude to achieve a specified displacement at a control point in the structure. This could be used to push a structure laterally to the point of yielding or collapse. The magnitude of the load required to do this is unknown, and may increase at first and then later decrease.

33

Each nonlinear static case may apply to the whole structure or to any part of it. You may specify changes to the structure within a case or between subsequent cases. Each portion of a nonlinear case that applies to a fixed part of the structure is called a stage. When elements are added from one stage to the next, their contribution to the stiffness, mass, and the specified load pattern are added to the structure. When elements are removed, the stiffness and mass are removed, and all loads that were carried by the removed part of the structure are applied to the remaining structure. These features allow you to perform incremental analysis.

Nonlinear static analysis is independent of all other analysis cases, except that previously calculated mode shapes may be used to define the load pattern.

Initial P-Delta analysis has no affect on nonlinear static cases. P-Delta effects are considered separately within each nonlinear static case, if requested. Unlike initial P-Delta analysis where the P-Delta effect is evaluated under a constant load, the P-Delta effect in nonlinear static analysis may vary with each incremental change in the load acting on the structure.

The results of nonlinear static cases should not normally be added together with each other or with other types of analysis cases. The exception to this would be cases that only include incremental construction, but not geometric or material nonlinearity. If the nonlinear effects of additive loads are required, you

should define nonlinear static cases that combine the loads in the expected sequence. In all situations, you can use nonlinear static cases for design, and they can be included in load combinations of envelope type.



Point Object Output Conventions

Overview

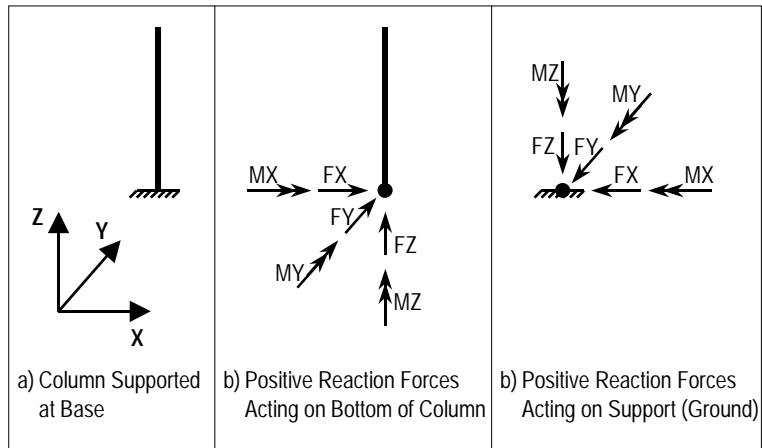
This chapter introduces the types of output for point objects and the sign conventions used to report the output. Recall that the point object local 1, 2 and 3 axes are in the same direction as the global X, Y and Z axes, respectively, always.

The following subsections discuss the sign conventions for various types of point object output.

Displacements

Except for point objects with panel zone assignments, point object displacements are reported with respect to the global coordinate system. Positive translational displacements are in the same direction as the positive global axes. The sense of positive rotations can be determined using the right hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for information on the right-hand rule.

Figure 34-1:
Positive support reaction forces



See the subsection below titled panel zone displacements for point objects that have panel zone assignments.

34

Support Reactions



Note:

With the exception of point object displacements when panel zones are assigned to a point object, all point object output is reported with respect to the global coordinate system.

Support reactions are reported as forces acting on the elements connected to the support. The reaction forces are reported with respect to the global coordinate system. Positive support reactions forces act in the same direction as the positive global axes. The sense of positive moments can be determined using the right hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for information on the right-hand rule.

As an example consider a building column that is supporting gravity load. This gravity load acts in a downward direction. Thus the force imposed on the bottom of the column acts in an upward direction. This is the reaction force reported by ETABS. Since the upward force is in the same direction as the positive global Z-axis the reaction is reported as a positive value acting in the Z direction.

Figure 34-1 illustrates positive support reaction forces.

Spring Forces

Spring forces are reported as forces acting on the elements connected to the support. They are reported with respect to the global coordinate system. Positive spring forces act in the same direction as the positive global axes. The sense of positive moments can be determined using the right hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for information on the right-hand rule.

See the example discussed in the subsection above titled “Support Reactions” for additional information.

Grounded Link Element Forces

Note:

See Chapter 37 for output conventions related to link elements.

34

Grounded link element forces (forces in link elements that are assigned to point objects, not including panel zones) are described in Chapter 37. Chapter 37 also discusses output conventions for link elements that are assigned to line objects.

Panel Zone Output

Internally in the analysis model ETABS creates two joints at locations where panel zones are assigned to point objects. If the panel zone is between columns and a beams then one of the joints is connected to the columns and one is connected to the beams. Column-brace and brace-beam panel zones are similar.

ETABS reports panel zone displacements, panel zone internal deformations and panel zone internal forces. Each of these is described in the three subsections below. Do not confuse panel zone displacements and internal deformations. They are different as described below.

Panel Zone Displacements

ETABS reports displacements for a point object with a panel zone assignment as follows:

- **Beam-Column Connectivity:** Displacements are reported for the joint connected to the column.

- **Beam-Brace Connectivity:** Displacements are reported for the joint connected to the beam.
- **Brace-Column Connectivity:** Displacements are reported for the joint connected to the column.

Panel zone displacements are reported in the global coordinate system. Positive translational displacements are in the same direction as the positive global axes. The sense of positive rotations can be determined using the right hand rule. See the section titled "The Right Hand Rule" in Chapter 23 for information on the right-hand rule.

Panel Zone Internal Deformations

Panel zone internal deformations are calculated as described for link element internal deformations in the section titled "Link Element Internal Deformations" in Chapter 37. They are reported in the panel zone *local coordinate system*. See the subsection titled "Local 2-Axis" under the section titled "Panel Zone Assignments to Point Objects" in Chapter 14 for discussion of the panel zone local coordinate system. The following conventions are used by ETABS:

- **Beam-Column Connectivity:** The i-end of the panel zone is the joint that is connected to the columns. The j-end of the panel zone is the joint that is connected to the beams.
- **Beam-Brace Connectivity:** The i-end of the panel zone is the joint that is connected to the beams. The j-end of the panel zone is the joint that is connected to the braces.
- **Brace-Column Connectivity:** The i-end of the panel zone is the joint that is connected to the columns. The j-end of the panel zone is the joint that is connected to the braces.



Note:

Panel zone internal forces and deformations are reported in the panel zone local coordinate system. All other point object output is reported in the global coordinate system.

Panel Zone Internal Forces

The panel zone internal forces give you a sense of how much unbalanced load is transferred through the panel zone. For example, in a panel zone with beam-column connectivity the internal forces tell you how much unbalanced moment is transferred from the beams to the columns.

Panel zone internal forces are calculated as described for link element internal forces in the section titled "Link Element Internal Forces" in Chapter 37. Since panel zone elements are zero length elements $M_2 = M_{2b}$ and $M_3 = M_{3b}$. The panel zone internal forces are reported in the panel zone *local coordinate system*.



Frame Element Output Conventions

35

General

Note:

 Refer to the section titled "Default Line Object Local Axes" in Chapter 24 for a discussion of the frame element local axes.

This chapter describes the types of output for frame elements and the sign conventions used to report the output. To fully comprehend these output conventions it is important that you have a clear understanding of the local coordinate system for frame elements. Refer to the section titled "Default Line Object Local Axes" in Chapter 24 for a discussion of the frame element local axes. Note that the line object and frame element local axes are the same.

Output for frame elements is reported as frame element internal forces. Tabulated and printed output data is available for frame elements from the following sources:

- Frame element output data tabulated onscreen can be viewed using the **Display menu > Set Output Table Mode** command. See the section titled "Output Table Mode" in Chapter 16 for additional information.

- Frame element output data tabulated in a Microsoft Access database file can be obtained using the **File menu > Export > Save Input/Output as Access Database File** command. See the bullet item labeled “**Save Input/Output as an Access database file**” in the section titled “Exporting Files” in Chapter 8 for additional information.
- Frame element output data can be printed to a printer or to a text file using the **File menu > Print Tables > Analysis Output** command. See the subsection titled “Printing Text Input and Output Tables” under the section titled “Printing from ETABS” in Chapter 8 for additional information.

The tabulated and printed data for frame elements provides output at every output station along the beam. See the section titled “Frame Output Station Assignments to Line Objects” in Chapter 14 for additional information on output stations.

In addition to the tabulated and printed data, frame element output can be displayed on your ETABS model using the **Display menu > Show Member Forces/Stress Diagram > Frame/Pier/Spandrel Forces** command. See the subsection titled “Frame Element, Pier and Spandrel Forces” under the section titled “Member Force and Stress Diagrams” in Chapter 16 for additional information.

The remaining section in this chapter discusses frame element internal force output.

Frame Element Internal Forces

The frame element internal forces are:

- P , the axial force
- V_2 , the shear force in the 1-2 plane
- V_3 , the shear force in the 1-3 plane
- T , the axial torque (about the 1-axis)

- M₂, the bending moment in the 1-3 plane (about the 2-axis)
- M₃, the bending moment in the 1-2 plane (about the 3-axis)

These internal forces and moments are present at every cross section along the length of the frame element.

For each load case and load combination the frame element internal forces and moments are computed and reported at each frame element output station. See the subsection titled “Frame Output Station Assignments to Line Objects” in Chapter 14 for a description of and additional information on output stations.

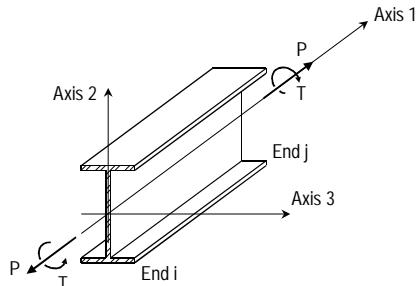
In frame element output displayed in a tabular form, either on the computer screen, printed to a printer or printed to a file, the locations of the output stations are identified by the absolute distance to the station measured from the i-end of the element.

Note:

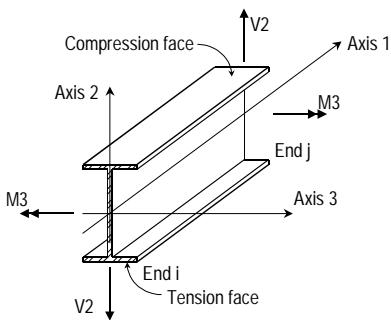
Frame element internal forces are reported at each output station along the frame element. See the subsection titled “Frame Output Station Assignments to Line Objects” in Chapter 14 for discussion of output stations

The sign convention for frame element internal forces is illustrated in Figure 35-1. This sign convention can be described by defining the concept of positive and negative faces of an element. Consider a section cut through the element in the 2-3 plane. At this section the positive 1 face is the face whose outward normal (arrow that is perpendicular to the section and pointing away from the section) is in the positive local 1 direction. At this same section the negative one face is one whose outward normal is in the negative local 1 direction. The positive 2 and 3 faces are those faces with outward normals in the positive local 2 and 3 directions, respectively, from the neutral axis. Note the following about the frame element internal forces:

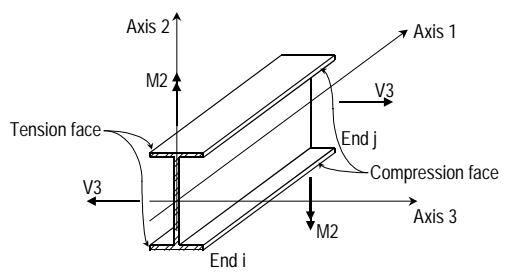
- Positive internal forces (P, V₂ and V₃) and positive axial torque (T) acting on a positive 1 face are oriented in the positive direction of the corresponding element local coordinate axis. For example, when V₂ acting on a positive 1 face is positive it is oriented in the direction of the positive local 2-axis.



a. Positive Axial Force and Torque



b. Positive Moment and Shear in the 1-2 Plane



c. Positive Moment and Shear in the 1-3 Plane

(Above)

Figure 35-1:
*Sign convention for
frame element inter-
nal forces*

- Positive internal forces (P , $V2$ and $V3$) and positive axial torque (T) acting on a negative 1 face are oriented in the negative direction of the corresponding element local coordinate axis. For example, when $V2$ acting on a negative 1 face is positive it is oriented in the direction of the negative local 2-axis.
- Positive $M2$ bending moments cause compression on the positive 3 face and tension on the negative 3 face.
- Positive $M3$ bending moments cause compression on the positive 2 face and tension on the negative 2 face.
- When end offsets along the length of the frame element are present, the internal forces and moments are output at the faces of the supports rather than the ends of the element. No output is produced within the end offset length.

- The right hand rule applies in Figure 35-1 for determining the sense of the moments shown by the double arrows. See the section titled “The Right Hand Rule” in Chapter 23 for information on the right-hand rule.



Chapter 36

Shell Element Output Conventions

36

General

Note:

 Refer to the section titled "Default Area Object Local Axes" in Chapter 23 for a discussion of the shell element local axes.

This chapter describes the types of output for shell elements and the sign conventions used to report the output. To fully comprehend these output conventions it is important that you have a clear understanding of the local coordinate system for shell elements. Refer to the section titled "Default Area Object Local Axes" in Chapter 23 for a discussion of the shell element local axes. Note that the area object and shell element local axes are the same.

Output for shell elements is reported as shell element internal forces and stresses. Tabulated and printed output data is available for shell elements from the following sources:

- Shell element output data tabulated onscreen can be viewed using the **Display menu > Set Output Table Mode** command. See the section titled "Output Table Mode" in Chapter 16 for additional information.

- Shell element output data tabulated in a Microsoft Access database file can be obtained using the **File menu > Export > Save Input/Output as Access Database File** command. See the bullet item labeled “**Save Input/Output as an Access database file**” in the section titled “Exporting Files” in Chapter 8 for additional information.
- Shell element output data can be printed to a printer or to a text file using the **File menu > Print Tables > Analysis Output** command. See the subsection titled “Printing Text Input and Output Tables” under the section titled “Printing from ETABS” in Chapter 8 for additional information.

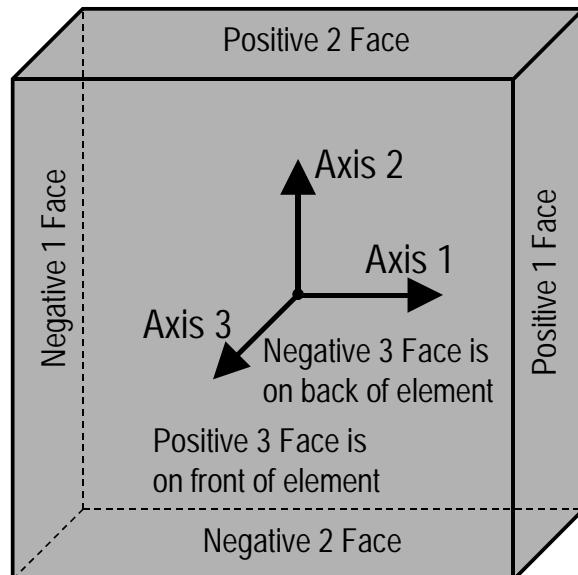
In addition to the tabulated and printed data, shell element output can be displayed on your ETABS model using the **Display menu > Show Member Forces/Stress Diagram > Shell Stresses/Forces** command. See the subsection titled “Shell Forces and Stresses” under the section titled “Member Force and Stress Diagrams” in Chapter 16 for additional information.

The remaining section in this chapter discusses shell element internal forces and stresses output.

Faces of Shell Elements

The six faces of a shell element are defined as the positive 1 face, negative 1 face, positive 2 face, negative 2 face, positive 3 face and negative 3 face as shown in Figure 36-1. In this definition the numbers 1, 2 and 3 correspond to the local axes of the shell element. The positive 1 face of the element is the face that is perpendicular to the 1-axis of the element whose outward normal (pointing away from the element) is in the positive 1-axis direction. The negative 1 face of the element is a face that is perpendicular to the 1-axis of the element whose outward normal (pointing away from the element) is in the negative 1-axis direction. The other faces have similar definitions.

Figure 36-1:
The six faces of a
shell element



Note that the positive 3 face is sometimes called the top of the shell element in ETABS, particularly in the output, and the negative 3 face is called the bottom of the shell element.

Shell Element Internal Forces

The shell element internal forces, like stresses, act throughout the element. They are present at every point on the midsurface of the shell element. ETABS reports values for the shell internal forces at the element nodes. It is important to note that the internal forces are reported as forces and moments per unit of in-plane length.

The basic shell element forces and moments are identified as F_{11} , F_{22} , F_{12} , M_{11} , M_{22} , M_{12} , V_{13} and V_{23} . You might expect that there would also be an F_{21} and M_{21} , but F_{21} is always equal to F_{12} and M_{21} is always equal to M_{12} , so it is not actually necessary to report F_{21} and M_{21} .

Figure 36-2:
 F_{11} forces acting on
shell midsurface

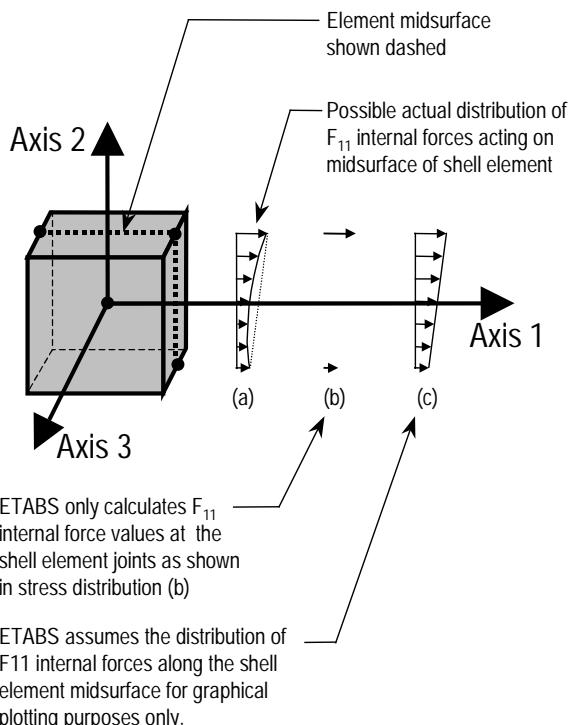
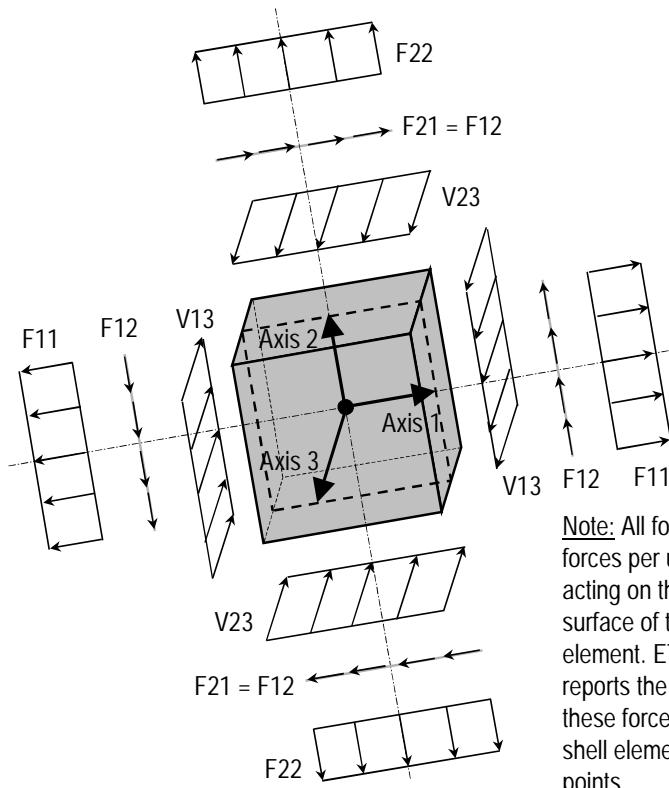


Figure 36-2 shows internal F_{11} forces acting on the midsurface of a shell element. In the figure, the force distribution labeled (a) represents an actual F_{11} force distribution. The force distribution labeled (b) shows how ETABS only calculates the internal forces at the corner points of the shell element. Note that we could calculate these stresses at any location on the shell element. We simply choose to calculate them only at the corner points because that is a convenient location and it keeps the amount of output to a reasonable volume.

The force distribution labeled (c) in Figure 36-2 shows how ETABS assumes that the F_{11} forces vary linearly along the length of the shell element between the calculated F_{11} force values at the element nodes *for graphical plotting purposes only*.

Shell element internal forces F_{11} , F_{22} , F_{12} , M_{11} , M_{22} and M_{12} are calculated from the shell element nodal displacements using a displacement function that is assumed to occur throughout the element. The internal forces are evaluated at the standard 2-by-2 Gauss integration points of the shell element and then extrapolated.

Figure 36-3:
Positive directions
for shell element
internal forces F_{11} ,
 F_{22} , F_{12} , V_{13} and V_{23}



Note: All forces are forces per unit length acting on the mid-surface of the shell element. ETABS only reports the value of these forces at the shell element corner points.

lated to the joints. See Cook, Malkus, and Plesha (1989) for more information.

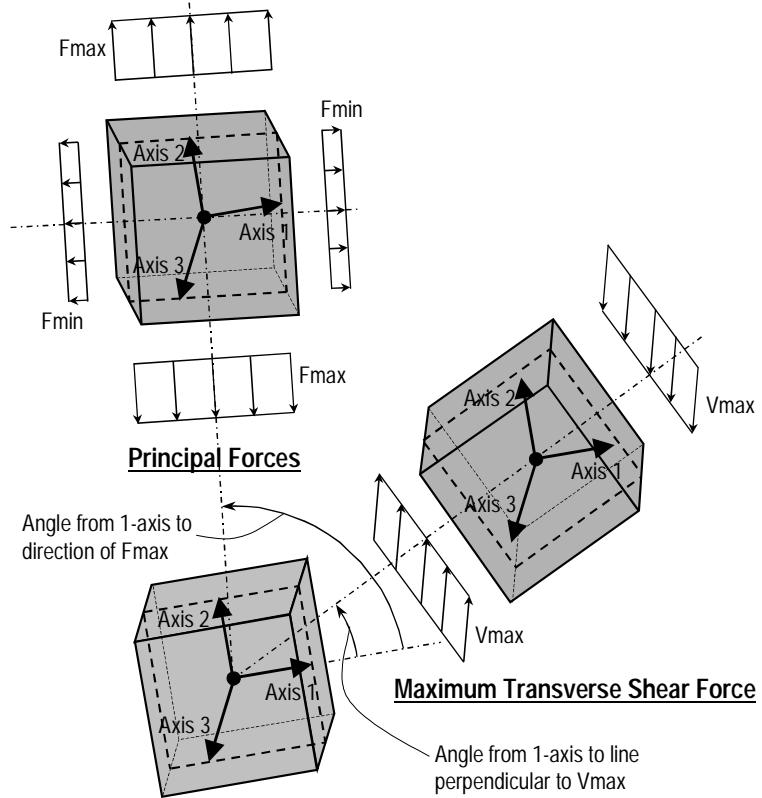
Shell element internal transverse shear forces V_{13} and V_{23} are calculated as shown in equations 36-1a and 36-1b where x_1 and x_2 are in-plane coordinates parallel to the local 1 and 2 axes and the distribution of the moments along the 1 and 2 axes is based on the previously mentioned assumed displacement function.

$$V_{13} = \frac{dM_{11}}{dx_1} - \frac{dM_{12}}{dx_2} \quad \text{Eqn. 36-1a}$$

$$V_{23} = \frac{dM_{12}}{dx_1} - \frac{dM_{22}}{dx_2} \quad \text{Eqn. 36-1b}$$

Figure 36-3 illustrates the positive directions for shell element internal forces F_{11} , F_{22} , F_{12} , V_{13} and V_{23} . **Note that these shell**

Figure 36-4:
Positive directions for shell element principal forces and maximum transverse shear force

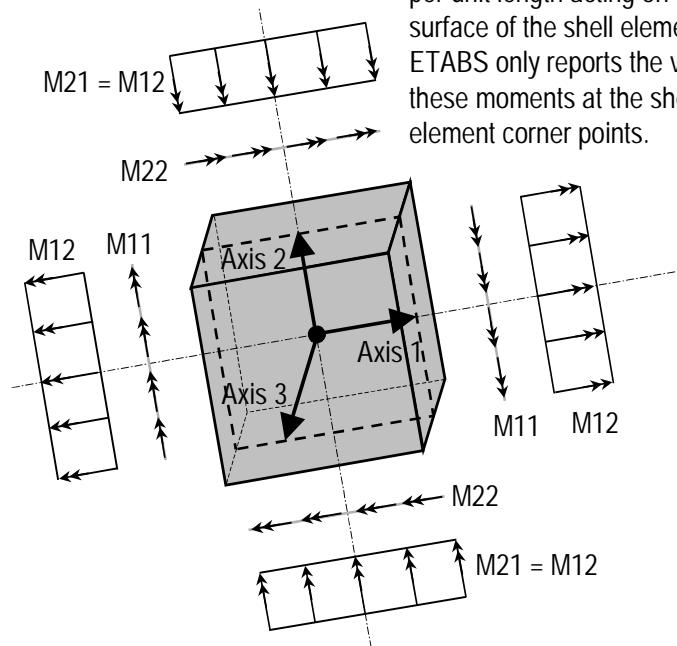


element internal forces are forces per unit length acting on the midsurface of the shell element. ETABS only reports the value of these forces at the shell element corner points.

Figure 36-4 illustrates the positive direction for shell element principal forces, F_{max} and F_{min} . The angle reported for the principal forces is measured in the local 1-2 plane of the shell element from the local 1 axis of the element to the direction of the maximum principal forces. Positive angles appear counter-clockwise when viewed looking down on the top (positive 3 face) of the shell element.

Figure 36-4 also illustrates the positive direction for the shell element maximum transverse shear force, V_{max} . The angle reported for the maximum transverse shear force is measured in the local 1-2 plane of the shell element from the local 1 axis of the element to a line that is perpendicular to the maximum shear

Figure 36-5:
Positive directions
for shell element
internal moments
 M_{11} , M_{22} and M_{12}



36

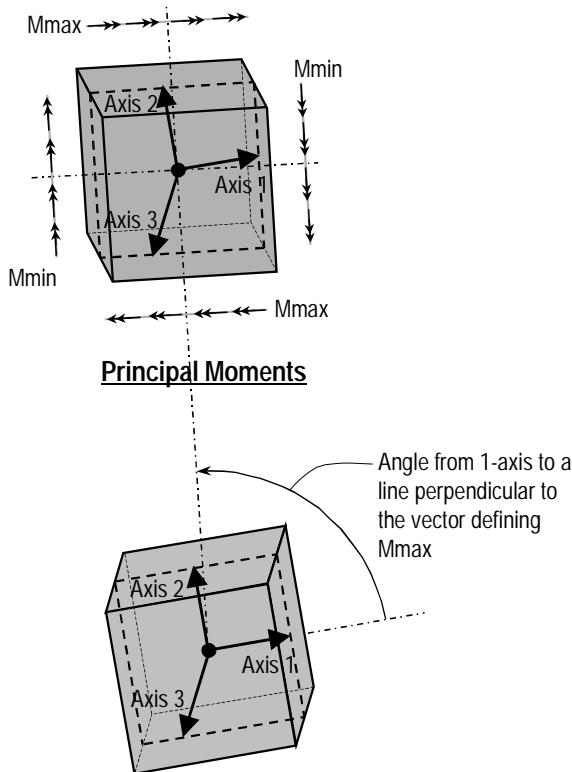
force. Positive angles appear counterclockwise when viewed looking down on the top (positive 3 face) of the shell element.

Note that there is no relationship between the angle for the principal forces and the angle for the maximum transverse shear force. For values of V_{13} and V_{23} at any angle, the maximum transverse shear stress, $V\text{-Max}$, can be calculated from equation 36-2.

$$V\text{-Max} = \sqrt{V_{13}^2 + V_{23}^2} \quad \text{Eqn. 36-2}$$

Figure 36-5 illustrates the positive directions for shell element internal moments M_{11} , M_{22} and M_{12} . **Note that these shell element internal moments are moments per unit length acting on the midsurface of the shell element. ETABS only reports the value of these moments per unit length at the shell element corner points.**

Figure 36-6:
*Positive directions
 for shell element
 principal moments
 M_{max} and M_{min}*



Use the right hand rule to determine the sense of the moments shown in Figure 36-5 and Figure 36-6. See the section titled "The Right hand Rule" in Chapter 23 for more information.

Figure 36-6 illustrates the positive direction for shell element principal moments, M_{max} and M_{min} . The angle reported for the principal moments is measured in the local 1-2 plane of the shell element from the local 1 axis of the element to a line perpendicular to the vector defining the maximum principal moment. Positive angles appear counterclockwise when viewed looking down on the top (positive 3 face) of the shell element.

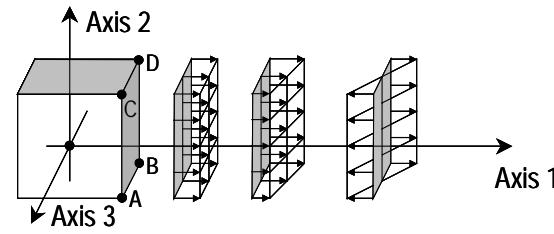
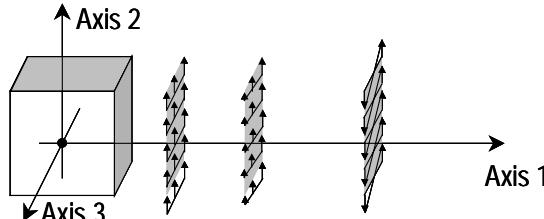
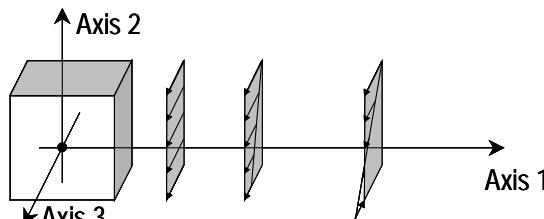
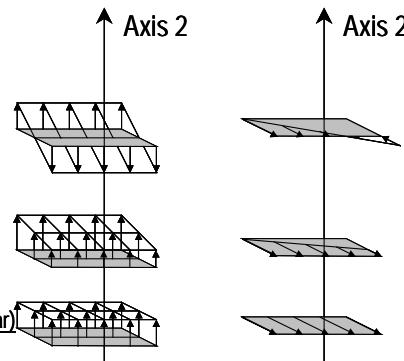
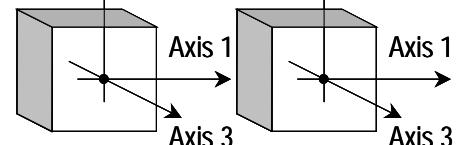
Shell Element Internal Stresses

The basic shell element stresses are identified as S_{11} , S_{22} , S_{12} , S_{13} , and S_{23} . You might expect that there would also be an S_{21} , but S_{21} is always equal to S_{12} , so it is not actually necessary to report

S_{21} . S_{ij} stresses (where i can be equal to 1 or 2 and j can be equal to 1, 2 or 3) are stresses that occur on face i of an element in direction j . Direction j refers to the local axis direction of the shell element. Thus S_{11} stresses occur on face 1 of the element (perpendicular to the local 1 axis) and are acting in the direction parallel to the local 1 axis (that is, the stresses act normal to face 1). As another example, S_{12} stresses occur on face 1 of the element (perpendicular to the local 1 axis) and are acting in the direction parallel to the local 2 axis (that is, the stresses act parallel to face 1, like shearing stresses). Figure 36-7 shows examples of each of these basic types of shell stresses. ETABS reports internal stresses for shell elements at the four corner points of the appropriate face of the element. For example, refer to Figure 36-7a. On the positive 1 face internal stresses are reported by ETABS at points A, B, C and D.

Shell internal stresses are reported for both the top and the bottom of the shell element. As previously mentioned in the section titled "Faces of Shell Elements", the top and bottom of the element are defined relative to the local 3-axis of the element. The positive 3-axis side of the element is considered to be the top of the element. Thus in Figure 36-7a, internal stresses at the top of the element include stresses at the joints labeled A and C and internal stresses at the bottom of the element include stresses at the joints labeled B and D. Figure 36-8 clearly illustrates the points where ETABS reports the shell element internal stress values.

As mentioned in the previous section, shell element internal forces (not stresses) are calculated from the shell element nodal displacements. The shell element internal stresses are then calculated from the shell element internal forces using Equations 36-3a through 36-3f. In these equations t_m is the membrane thickness of the shell, t_b is the bending thickness of the shell and x_3 is the thickness coordinate measured from the midsurface of the element.

a. Examples of membrane direct stresses, S_{11} b. Examples of membrane shear stresses, S_{12} (S_{21} stresses similar)c. Examples of plate transverse shear stresses, S_{13} d. Examples of plate transverse direct stresses, S_{22} e. Examples of plate transverse shear stresses, S_{23}

36

(Above)

Figure 36-7:
Examples of various
types of shell
stresses

$$S_{11} = \frac{F_{11}}{t_m} - \frac{12M_{11}}{t_b^3} x_3$$

Eqn. 36-3a

$$S_{22} = \frac{F_{22}}{t_m} - \frac{12M_{22}}{t_b^3} x_3$$

Eqn. 36-3b

$$S_{12} = \frac{F_{12}}{t_m} - \frac{12M_{12}}{t_b^3} x_3$$

Eqn. 36-3c

$$S_{13} = \frac{V_{13}}{t_b}$$

Eqn. 36-3d

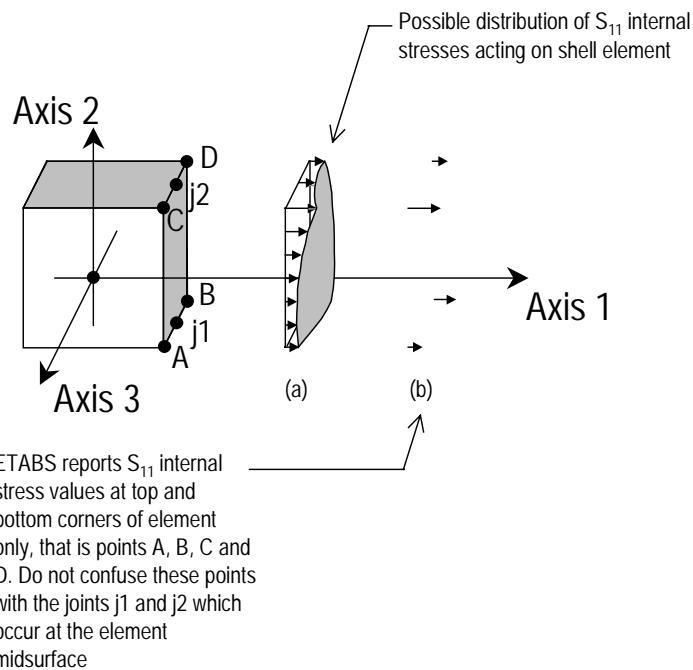
$$S_{23} = \frac{V_{23}}{t_b}$$

Eqn. 36-3e

$$S_{33} = 0$$

Eqn. 36-3f

Figure 36-8:
Locations where
ETABS reports shell
element internal
stresses

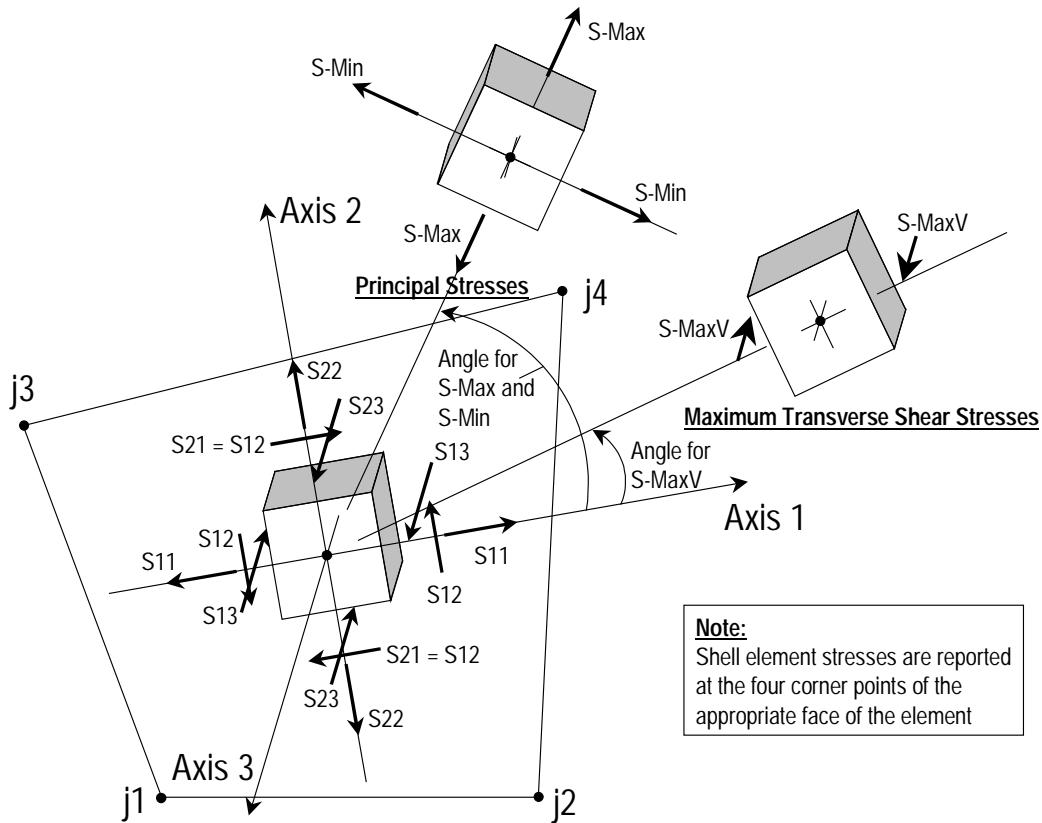


36

The transverse shear stresses calculated from equations 36-3d and 36-3e are average values. The actual transverse shear stress distribution is approximately parabolic; it is zero at the top and bottom surfaces and has its maximum or minimum value at the midsurface of the element. ETABS reports the average transverse shear value. An approximation to the maximum (or minimum) transverse shear stress would be 1.5 times the average shear stress.

Figure 36-9 illustrates the positive directions for shell element internal stresses S_{11} , S_{22} , S_{12} , S_{13} and S_{23} . Also shown are the positive directions for the principal stresses, $S\text{-Max}$ and $S\text{-Min}$, and the positive directions for the maximum transverse shear stresses, $S\text{-Max-V}$. The angle reported for the principal stresses is positive when measured counterclockwise (when viewed from the top of the shell element) from the local 1-axis to the direction of the maximum principal value.

The angle reported for the maximum transverse shear stresses is also measured counterclockwise (when viewed from the top) from the local 1-axis to the direction of the maximum principal



(Above)

Figure 36-9:
Positive directions for shell element internal stresses S_{11} , S_{22} , S_{12} , S_{13} and S_{23}

value. Note that there is no relationship between the angle for the principal stresses and the angle for the maximum transverse shear stresses. For values of S_{13} and S_{23} at any angle, the maximum transverse shear stress, $S\text{-MaxV}$, can be calculated from equation 36-4.

$$S\text{-MaxV} = \sqrt{S_{13}^2 + S_{23}^2}$$

Eqn. 36-4

Other Formulas Relating Shell Element Internal Forces to Internal Stresses

The shell element internal forces and moments can be obtained by integrating the shell element internal stresses over the element thickness. The integration is performed about the midsurface of the element. *Note that this is not the process used by ETABS*

since, as described above, ETABS calculates the internal forces first and then determines the internal stresses from the internal forces.

The internal shell element forces, F_{ij} and V_{ij} (where i can be equal to 1 or 2 and j can be equal to 1, 2 or 3), can be viewed as the forces caused by the S_{ij} stresses acting on face i . Similarly, the internal shell element moments, M_{ij} , can be viewed as the moments caused by the S_{ij} stresses acting on face i . For example, F_{11} forces are caused by S_{11} stresses acting on face 1 and M_{12} moments result from S_{12} stresses acting on face 1. Note that this explanation rationally explains why M_{11} moments act about the local 2-axis and M_{22} moments act about the local 1-axis, a phenomenon that causes confusion for many people.

The shell element internal forces can be calculated from the internal stresses as shown in equations 36-5a through 36-5h. In these equations t_m is the membrane thickness of the shell, t_b is the bending thickness of the shell and x_3 is the thickness coordinate measured from the midsurface of the element. Remember that the shell element internal forces and moments are forces and moments per unit of in-plane length of the shell element.

- Membrane direct forces:

$$F_{11} = \int_{t_m/2}^{+t_m/2} S_{11} dx_3 \quad \text{Eqn. 36-5a}$$

$$F_{22} = \int_{t_m/2}^{+t_m/2} S_{22} dx_3 \quad \text{Eqn. 36-5b}$$

- Membrane shear force:

$$F_{12} = \int_{t_m/2}^{+t_m/2} S_{12} dx_3 \quad \text{Eqn. 36-5c}$$

- Plate bending moments:

$$M_{11} = \int_{t_b/2}^{+t_b/2} S_{11} x_3 dx_3 \quad \text{Eqn. 36-5d}$$

$$M_{22} = \int_{t_b/2}^{+t_b/2} S_{22} x_3 dx_3 \quad \text{Eqn. 36-5e}$$

- Plate twisting moment:

$$M_{12} = \int_{-t_b/2}^{+t_b/2} S_{12} x_3 dx_3 \quad \text{Eqn. 36-5f}$$

- Plate transverse shear forces:

$$V_{13} = \int_{-t_b/2}^{+t_b/2} S_{13} dx_3 \quad \text{Eqn. 36-5g}$$

$$V_{23} = \int_{-t_b/2}^{+t_b/2} S_{23} dx_3 \quad \text{Eqn. 36-5h}$$

Link Element Output Conventions

37

General

Available output for link elements includes the deformation across the element and the internal spring forces reported at the joints (ends) of the element. Each of these is described below.

Link Element Assignments to Point and Line Objects

Note:

Link element properties can be assigned to point and line objects.

Link elements can be assigned to point or line objects. The link elements assigned to point objects are different from those assigned to line objects as described below:

- A link element assigned to a point object is connected to the point object and to the ground unless it is a special panel zone element that has been assigned link properties. See the section titled “Link Property Assignments to Point Objects” in Chapter 14 for discussion of link property assignments made to point objects. Note that

the local axes for the grounded link element are described here.

Output for point objects with panel zone assignments where the panel zone property is based on a specified link property is discussed in Chapter 34.

- A link element assigned to a line object connects (links) the two point objects at the ends of the line object. One of the point objects that the link element is connected to may be restrained if desired. See the section titled “Link Property Assignments to Line Objects” in Chapter 14 for discussion of link property assignments made to line objects. The local axes for the two-joint link elements are the same as those for a line object described in the section titled “Default Line Object Local Axes” in Chapter 24.

Internal Nonlinear Springs

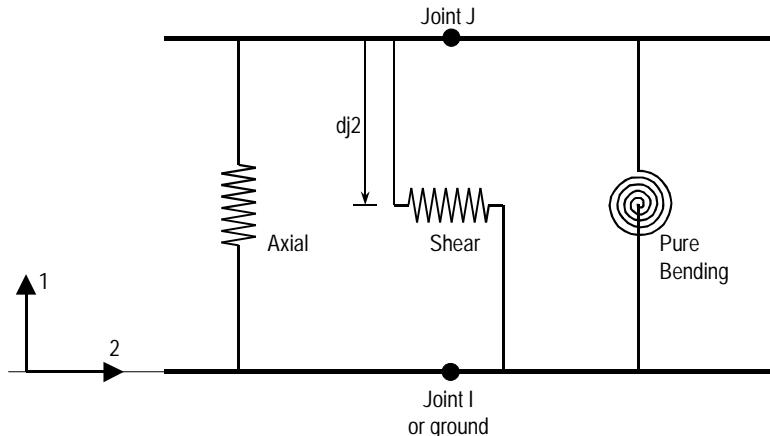
37

Each link element is assumed to be composed of six internal “springs,” one for each of the six internal deformations. Each spring may consist of several components including springs and dashpots. The force-deformation relationships of these springs may be coupled or independent of each other.

Figure 37-1 shows the springs for three of the deformations: axial, shear in the 1-2 plane, and pure bending in the 1-2 plane. It is important to note that the shear spring is located at a distance d_{j2} from joint j. All shear deformation is assumed to occur at this spring; the links connecting this spring to the joints (or ground) are rigid in shear. Deformation of the shear spring can be caused by rotations as well as translations at the joints.

The force in this shear spring produces a linearly varying moment along the length of the link. This moment is taken as zero at the shear spring, which acts as a moment hinge. The moment due to shear is independent of, and additive to, the constant moment in the element due to the pure-bending spring.

Figure 37-1:
Three of the six independent nonlinear springs in a link element



The other three springs that are not shown in Figure 37-1 are for torsion, shear in the 1-3 plane and pure-bending in the 1-3 plane. The spring for shear in the 1-3 plane is located at a distance d_{j3} from point j similar to the spring for shear in the 1-2 plane. The values of d_{j2} and d_{j3} may be different, although normally they are the same.

Link Element Force-Deformation Relationships

There are six force-deformation relationships that govern the behavior of the link element, one for each of the internal springs. The force deformation relationships governing the behavior are:

- Axial: f_{u1} versus d_{u1}
- Shear in the 1-2 plane: f_{u2} versus d_{u2}
- Shear in the 1-3 plane: f_{u3} versus d_{u3}
- Torsion: f_{r1} versus d_{r1}
- Pure bending in the 1-3 plane: f_{r2} versus d_{r2}
- Pure bending in the 1-2 plane: f_{r3} versus d_{r3}

where f_{u1} , f_{u2} and f_{u3} are the internal spring forces in the link local axes directions and f_{r1} , f_{r2} and f_{r3} are the internal spring moments about the link local axes.

Note that each of these force deformation relationships may be zero, linear only, or linear/nonlinear for any given link element. These relationships may be coupled or uncoupled. The internal forces and moments may be related to the deformation rates (velocities) as well as the deformations.

Link Element Internal Deformations

Six independent internal deformations are defined for link elements. These are calculated from the relative displacements of end- j with respect to:

Note:

Six independent internal deformations are defined for link elements.

- End-i for a two-joint link element assigned to a line object.
- The ground for a single-joint link element assigned to a point object.

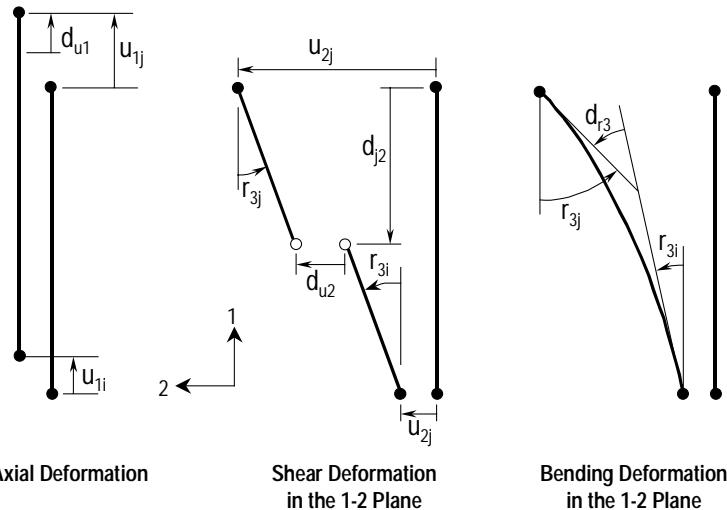
For two-joint link elements assigned to line objects the internal deformations are defined as:

- Axial: $d_{u1} = u_{1j} - u_{1i}$
- Shear in the 1-2 plane: $d_{u2} = u_{2j} - u_{2i} - d_{j2} r_{3j} - (L - d_{j2}) r_{3i}$
- Shear in the 1-3 plane: $d_{u3} = u_{3j} - u_{3i} + d_{j3} r_{2j} + (L - d_{j3}) r_{2i}$
- Torsion: $d_{r1} = r_{1j} - r_{1i}$
- Pure bending in the 1-3 plane: $d_{r2} = r_{2i} - r_{2j}$
- Pure bending in the 1-2 plane: $d_{r3} = r_{3j} - r_{3i}$

where:

- d_{u1} , d_{u2} , d_{u3} , d_{r1} , d_{r2} and d_{r3} , are the internal deformations of the link element.
- u_{1i} , u_{2i} , u_{3i} , r_{1i} , r_{2i} and r_{3i} are the translations and rotations at joint i.
- u_{1j} , u_{2j} , u_{3j} , r_{1j} , r_{2j} and r_{3j} are the translations and rotations at joint j.

Figure 37-2:
Internal deforma-
tions for a two-joint
link element



- d_{j2} is the distance you specify from joint j to the location where the shear deformation d_{u2} is measured (the default is zero).
- d_{j3} is the distance you specify from joint j to the location where the shear deformation d_{u3} is measured (the default is zero).
- L is the length of the element.

It is important to note the negatives of the rotations r_{2i} and r_{2j} have been used for the definition of shear and bending deformations in the 1-3 plane. This provides consistent definitions for shear and moment in both link and frame elements.

Note:

The conventions for displaying shear and moment are the same for frame and link elements.

Also note that all internal translations, rotations and deformations are expressed in terms of the link element local coordinate system. Finally note that shear deformations can be caused by rotations as well as translations. These definitions ensure that all deformations will be zero under rigid-body motion of the element.

Three of these internal deformations, axial, shear in the 1-2 plane and bending in the 1-2 plane are illustrated in Figure 37-2. The other internal deformations are similar.

For single-joint grounded link elements assigned to point objects the internal deformations are the same as above for two-joint link elements, except that the translations and rotations at end-i are taken to be zero:

- Axial: $d_{u1} = u_{1j}$
- Shear in the 1-2 plane: $d_{u2} = u_{2j} - d_{j2} r_{3j}$
- Shear in the 1-3 plane: $d_{u3} = u_{3j} + d_{j3} r_{2j}$
- Torsion: $d_{r1} = r_{1j}$
- Pure bending in the 1-3 plane: $d_{r2} = -r_{2j}$
- Pure bending in the 1-2 plane: $d_{r3} = r_{3j}$

where d_{u1} , d_{u2} , d_{u3} , d_{r1} , d_{r2} and d_{r3} , are the internal deformations of the link element.

37 Link Element Internal Forces

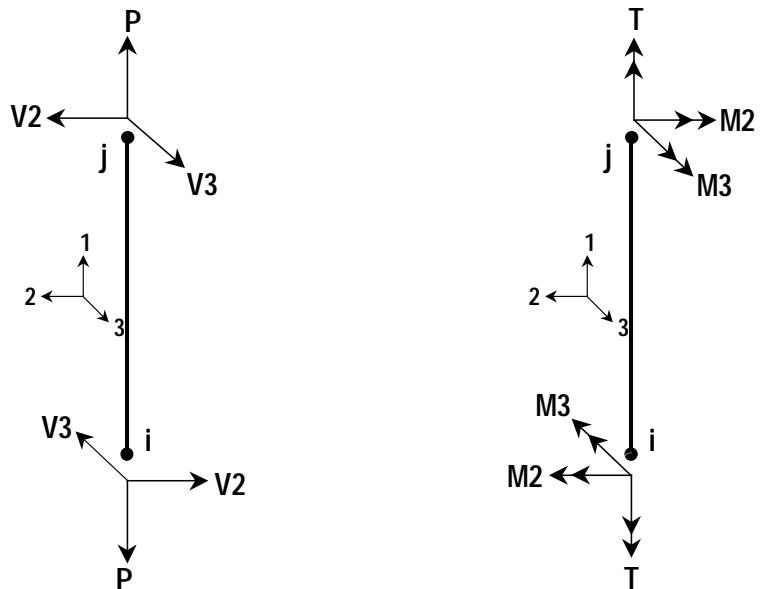
 **Note:** ETABS converts the link element *spring* internal forces (f_{u1} , f_{u2} , f_{u3} , f_{r1} , f_{r2} and f_{r3}) introduced in the previous subsection to link element internal forces (P, V2 and V3) and link element internal moments (T, M2 and M3) that are reported at the end points of the link element. These internal forces and moments have exactly the same meaning (conventions) as frame elements.

Although the link element internal forces are present at every point along the element, they are only reported at the end points of the element.

Figure 37-3 illustrates the positive direction for link element internal forces reported at the link element end points. These link element internal forces and moments are defined in terms of the link element internal *spring* forces and moments:

- Axial: $P = f_{u1}$
- Shear in the 1-2 plane: $V2 = f_{u2}$, $M3_s = (d - d_{j2}) f_{u2}$
- Shear in the 1-3 plane: $V3 = f_{u3}$, $M2_s = (d - d_{j3}) f_{u3}$
- Torsion: $T = f_{r1}$

Figure 37-3:
Positive directions
for link element in-
ternal forces P , V_2 ,
 V_3 , T , M_2 and M_3



- Pure bending in the 1-3 plane: $M_{2b} = f_{r2}$
- Pure bending in the 1-2 plane: $M_{3b} = f_{r3}$

37

where d is the distance from joint j . The total bending moment resultants M_2 and M_3 are composed of shear and bending parts as follows:

- Total bending in the 1-3 plane: $M_2 = M_{2s} + M_{2b}$
- Total bending in the 1-2 plane: $M_3 = M_{3s} + M_{3b}$

These internal forces and moments are present at every cross section along the length of the element. Again, they are only reported at the end points of the link element.

Wall Pier and Spandrel Output Conventions

38

General

This section describes the local axes and output conventions for wall pier and spandrel elements. Output for pier and spandrel elements is reported as element internal forces. Tabulated and printed output data is available for pier and spandrel elements from the following sources:



Note:

Pier and spandrel output forces are only reported at the ends of the elements

- Pier and spandrel element output data tabulated onscreen can be viewed using the **Display menu > Set Output Table Mode** command. See the section titled “Output Table Mode” in Chapter 16 for additional information.
- Pier and spandrel element output data tabulated in a Microsoft Access database file can be obtained using the **File menu > Export > Save Input/Output as Access Database File** command. See the bullet item labeled “**Save Input/Output as an Access database file**” in the section titled “Exporting Files” in Chapter 8 for additional information.

- Pier and spandrel element output data can be printed to a printer or to a text file using the **File menu > Print Tables > Analysis Output** command. See the subsection titled “Printing Text Input and Output Tables” under the section titled “Printing from ETABS” in Chapter 8 for additional information.

In addition to the tabulated and printed data, pier and spandrel element output can be displayed on your ETABS model using the **Display menu > Show Member Forces/Stress Diagram > Frame/Pier/ Spandrel Forces** command. See the subsection titled “Frame Element, Pier and Spandrel Forces” under the section titled “Member Force and Stress Diagrams” in Chapter 16 for additional information.

Important note concerning output locations: It is important to understand that output forces are only reported at the ends of the pier and spandrel elements. Similarly, when output forces for these elements are graphically displayed on the model, actual force values are plotted at the ends of the elements and then those actual values are simply connected with a straight line.

Wall Pier Output Locations and Sign Convention

Wall Pier Local Axes

Two-Dimensional Pier

Typically a wall pier is made up of one or more wall-type area objects or a combination of one or more wall-type area objects and one or more column-type line objects. The local axes of these two-dimensional wall piers are defined as follows:

- The local 1 axis extends from the bottom of the pier to the top of the pier. The positive local 1 axis is in the same direction as the positive global Z-axis.

The positive local 1-axis points from the i-end of the pier to the j-end of the pier.

- The local 1-2 plane is in the plane of the two-dimensional wall pier. The positive local axis 2 is horizontal and it has a positive projection on the global X-axis, or if the plane of the pier is oriented parallel to the global Y-Z plane, axis 2 is in the same direction as the positive global Y-axis.

Note:

In the special case where a wall pier is made up of a single column-type frame element the local axes of the pier are the same as those of the frame element.

- The local 3 axis is horizontal and perpendicular to the plane of the pier. The direction of the positive local 3 axis is determined from the right-hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

In the special case where a wall pier is made up of a single column-type frame element the local axes of the pier are the same as those of the frame element. See the section titled “Default Line Object Local Axes” in Chapter 24 for more information.

If a wall pier is made up of two or more column-type frame elements, but no area objects, then the local axes of the pier are the same as those of the first defined frame element. Your best bet for confirming the local axes orientation in this unusual case is to view them graphically using the **View menu > Set Building View Options** command.

Three-Dimensional Pier

In a three-dimensional wall pier the local axes are defined as follows:

Note:

For three-dimensional piers the relationship between the local and global axes is 1 = Z, 2 = X and 3 = Y.

- The local 1 axis extends from the bottom of the pier to the top of the pier. The positive local 1 axis is 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.

Pier Element Internal Forces

The wall pier element internal forces are similar to the frame element internal forces. They are:

- P, the axial force
- V₂, the shear force in the 1-2 plane
- V₃, the shear force in the 1-3 plane
- T, the axial torque
- M₂, the bending moment in the 1-3 plane (about the 2-axis)
- M₃, the bending moment in the 1-2 plane (about the 3-axis)

Note:

See the section titled “Wall Pier Labeling” in Chapter 48 for important information about defining wall piers

The positive directions of these forces are exactly the same as those described for the frame element. See Figure 35-1 in Chapter 35 for additional information.

38

Note:

The positive directions of output forces for wall piers are exactly the same as those described for the frame element in Chapter 35.

These internal forces and moments are present at every cross section along the height of the wall pier element. They are only reported at the ends (top and bottom) of the pier element. The pier internal forces and moments are reported in the pier element local coordinate system.

Since the pier internal forces and moments are only reported at the ends of the pier in some cases it may be advantageous to model a wall pier with multiple ETABS pier elements. For example, if you want output forces at the midheight of a wall pier, then you may want to model the pier with two ETABS pier elements. See the subsection titled “Assigning Wall Pier Labels” in Chapter 48 for more information.

Wall Spandrel Output Locations and Sign Convention

Note that wall spandrels can only be two-dimensional. There are no three-dimensional wall spandrels in ETABS.

Wall Spandrel Local Axes

- The local 1-2 plane is in the plane of the wall spandrel. The positive local axis 1 is horizontal and has a positive projection on the global X-axis, or if the plane of the spandrel is oriented parallel to the global Y-Z plane, axis 1 is in the same direction as the positive global Y-axis.

The positive local 1-axis points from the i-end of the spandrel to the j-end of the spandrel.

- The positive local 2 axis points upward in the same direction as the positive global Z-axis.
- The local 3 axis is horizontal and perpendicular to the plane of the spandrel. The direction of the positive local 3 axis is determined from the right-hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for more information.

In the case where a wall spandrel is made up of one or more beam-type frame elements the local axes of the spandrel are still the same as those described above. The local axes orientation of a spandrel are independent of the local axes orientation of its component area and line objects.

Spandrel Element Internal Forces

The wall spandrel element internal forces are similar to the frame element internal forces. They are:

Note:

The positive directions of output forces for wall spandrels are exactly the same as those described for the frame element in Chapter 35.

- P, the axial force
- V₂, the shear force in the 1-2 plane
- V₃, the shear force in the 1-3 plane
- T, the axial torque
- M₂, the bending moment in the 1-3 plane (about the 2-axis)
- M₃, the bending moment in the 1-2 plane (about the 3-axis)

The positive directions of these forces are exactly the same as those described for the frame element. See Figure 35-1 in Chapter 35 for additional information.

Note:

See the section titled “Wall Spandrel Labeling” in Chapter 48 for important information about defining wall spandrels.

These internal forces and moments are present at every cross section along the length of the wall spandrel element. They are only reported at the ends of the spandrel element. The spandrel internal forces and moments are reported in the spandrel element local coordinate system.

Since the spandrel internal forces and moments are only reported at the ends of the spandrel in some cases it may be advantageous to model a wall spandrel with multiple ETABS spandrel elements. For example, if you want output forces at the middle of a wall spandrel, then you may want to model the spandrel with two ETABS spandrel elements, one for each half of the spandrel. If you use the ETABS Shear Wall Design postprocessor to design such a spandrel, then note that any reported diagonal steel is based on the modeled spandrel element length (e.g., half the actual spandrel length), not the full spandrel length.



Section Cut Output Conventions

Overview

39

This chapter discusses the output conventions for section cuts. To fully comprehend these output conventions it is important that you have a clear understanding of the local coordinate system for section cuts. It is also important that you have a clear understanding of the location where the section cut forces are reported. Refer to the subsection titled "Defining Section Cuts" under the section titled "Section Cuts" in Chapter 11 for a discussion of both of these items.

Section Cut Forces

Section cut forces are reported at a single point in the local coordinate system defined for the section cut. Six different force components are reported at that single point. They are:

- **F1:** A force in the section cut local 1-axis direction.

**Note:**

Section cut forces are reported with respect to the section cut local coordinate system.

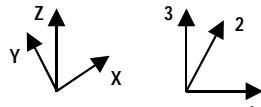
- **F2:** A force in the section cut local 2-axis direction.
- **F3:** A force in the section cut local 3-axis direction.
- **M1:** A moment about the section cut local 1-axis.
- **M2:** A moment about the section cut local 2-axis.
- **M3:** A moment about the section cut local 3-axis.

Section cut forces are reported as forces acting on the objects that make up the group that defines the section cut. An example of this is discussed below. Positive section forces act in the same direction as the positive section cut local axes. The sense of positive moments can be determined using the right hand rule. See the section titled “The Right Hand Rule” in Chapter 23 for information on the right-hand rule.

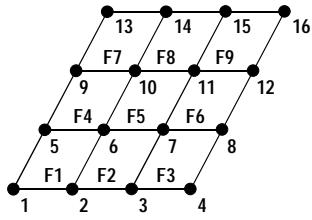
Consider the example shown in Figure 39-1. The global coordinate system axes and the section cut local coordinate system axes are shown in Figure 39-1a. This illustrates that the local coordinate system axes may be different from the global system.

Figure 39-1b shows a floor system that consists of 9 area objects labeled F1 through F9 and 16 associated point objects labeled 1 through 16. Suppose that we want to determine the section cut forces through the floor system at the location identified by the heavy line in Figure 39-1c. This section cut passes through the point objects labeled 3, 7, 11 and 15.

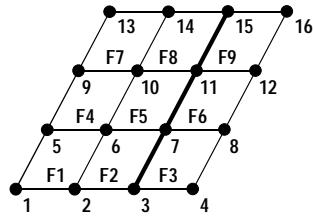
There are two possible groups you could define to use in the section cut definition. Figure 39-1d shows the first possible group which includes area objects F2, F5 and F8 and point objects 3, 7, 11 and 15. Figure 39-1e shows free body diagrams that define the positive direction of section cut forces when the section cut is defined using the group definition shown in Figure 39-1d. Note that the positive section cut forces acting on the *left* free body diagram are in the same direction as the positive section cut local axes shown in Figure 39-1a. The *left* free body diagram is the one that includes the objects that were used to define the group that defined the section cut.



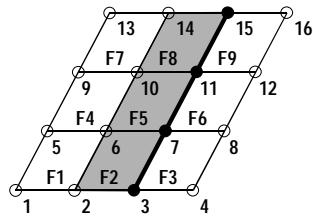
a) Global and Local Axes



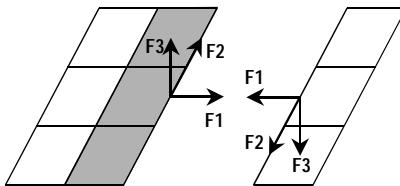
b) Floor System



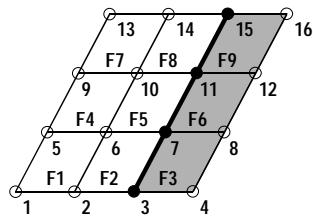
c) Section Cut in Floor System



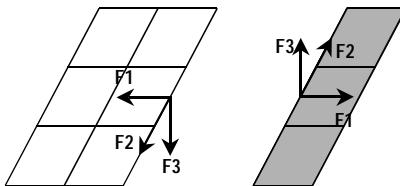
d) One Possible Group



e) Positive Forces for Section Cut Based on (d)



f) Another Possible Group



g) Positive Forces for Section Cut Based on (f)

(Above)
Figure 39-1:
Positive section cut forces

39

Figure 39-1f shows the second possible group that could define the section cut. This group includes area objects F3, F6 and F9 and point objects 3, 7, 11 and 15. Figure 39-1g shows free body diagrams that define the positive direction of section cut forces when the section cut is defined using the group definition shown in Figure 39-1f. Note that the positive section cut forces acting on the *right* free body diagram are in the same direction as the positive section cut local axes shown in Figure 39-1a. The *right* free body diagram is the one that includes the objects that were used to define the group that defined the section cut.

Important note: In the above example, if the diaphragm has a rigid diaphragm assignment then you will not get any forces in the section cut if it is defined as shown in the example. With the rigid diaphragm assignment ETABS is unable to capture section cut forces in the floor.

Tip: If you want to know the forces in a rigid diaphragm you can use the following technique. Define a group that contains all of the point objects at the diaphragm level that have columns, braces, ramps or walls connected to them and also contains all of the columns, braces, ramps and walls that are connected to the diaphragm both from above or from below. Define a section cut based on this group. The force in the section cut is the force in the diaphragm.

Note that in a free body diagram of the diaphragm the forces above and below the diaphragm are in opposite directions. If you add these forces above and below the diaphragm, taking signs into account you are left with the force in the diaphragm. This is what ETABS does in the case described for this tip.



Printed Input Tables

Features of Dialog Box

40

This chapter applies to the input tables that are printed using the **File menu > Print Tables > Input** command and to input tables displayed on the screen using the **Display menu > Set Input Table Mode** command. Note the following about the Print Input Tables dialog box:

Note:

You can print input data for selected objects only if you wish.

- **Select Loads button:** When you print static load information you can click on the Select Loads button to indicate for which load cases you want to display or print the input data. You can select one or more load cases in the resulting list.
- **Selection only:** When you print or display input data, if you make a selection prior to clicking the print or display command, then input data is only provided for the selected objects. Otherwise it is provided for all objects.

**Note:**

You can print input data to a printer or to a text file.

- **Print to File:** Checking the Print to File check box sends your input data to a text file rather than to the printer. When you check this box you can specify a name for the text file.
- **Append:** The Append check box is only active if the Print to File check box is checked. Checking the Append check box appends your input data to the specified file, if the file exists, rather than overwriting it.

Input Data Categories

The input data is broken into three main categories. They are:

- Building data
- Object data
- Static Loads

Most of the input data provided is relatively self-explanatory and thus is not mentioned here. Following are some comments on items in each of the categories that may need a little explanation.

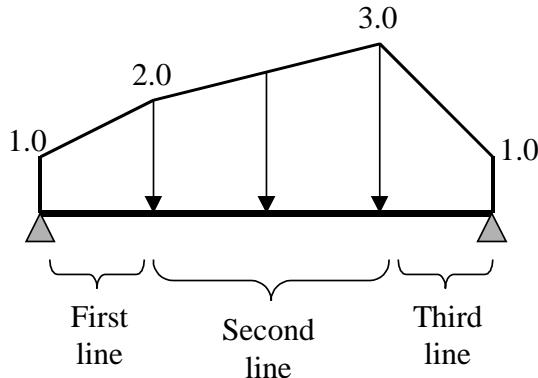
40

Building Data

The building info item in the Building Data area of the dialog box provides story level data information, point object coordinate information and area and line object connectivity information. The connectivity information tells which point objects the corners of area objects connect to and the ends of line objects connect to.

Object Data

The object data provides information on the assignments made to the various types of area, line and point objects in your model.



STORY ID	LINE ID	LOAD TYPE	LOAD DIRECTION	LOAD DISTANCE-A	LOAD VALUE-A	LOAD DISTANCE-B	LOAD VALUE-B
STORY2	B1	FORCE	GRAVITY	0.0000	1.0000	5.0000	2.0000
STORY2	B1	FORCE	GRAVITY	5.0000	2.0000	15.0000	3.0000
STORY2	B1	FORCE	GRAVITY	15.0000	3.0000	20.0000	1.0000

(Above)

Static Loads

Figure 40-1:
Printed input data
for a distributed line
load

The distributed line loads are reported in a slightly unusual way so that the output fits on the page. On each line for a given story ID and line ID a single, linearly varying segment of the distributed load is reported.

Figure 40-1 shows a distributed load with three linearly varying segments and its associated printed output. (Note that your output may appear slightly different). The distributed load intensity is 1.0 at each end, 2.0 at the left quarter-point of the span and 3.0 at the right quarter point. The input data for this distributed load is broken up into three lines. The first line applies to the load between the left end and the left quarter point. The second line applies to the load between the left and right quarter points. The final line applies to the load between the right quarter point and the end of the line object.



Chapter 41

Printed Output Tables

General

41

This chapter applies to the output tables that are printed using the **File menu > Print Tables > Analysis Output** command and to output tables displayed on the screen using the **Display menu > Set Output Table Mode** command. Note the following about output tables:

Note:

You can print output for selected objects only if you wish.

- **Select Loads button:** When you print displacements, reaction or object force output you can click on the Select Loads button to indicate for which load cases or combinations you want output. You can select one or more load cases or combinations in the resulting list.
- **Selection only:** When you print or display output, if you make a selection prior to clicking the print or display command, then output is only provided for the selected objects. Otherwise it is provided for all objects.

- **Envelopes only:** When you print or display output of displacements, reactions or forces, checking the Envelopes Only check box provides envelope-type output. For each output component of each object ETABS reports the maximum value along with its associated load case or combination and the minimum value along with its associated load case or combination.
- **Print to File:** Checking the Print to File check box sends your output to a text file rather than to the printer. When you check this box you can specify a name for the text file.
- **Append:** The Append check box is only active if the Print to File check box is checked. Checking the Append check box appends your output to the specified file, if the file exists, rather than overwriting it.



Note:

You can print output to a printer or to a text file.

Following is a list of some of the analysis output that is provided and references to chapters where this output is discussed.

- **Displacements, reactions and spring forces:** See chapter 34 for discussion of these items
- **Column, beam and brace forces:** See chapter 35 for discussion of these items
- **Pier and spandrel forces:** See chapter 38 for discussion of these items
- **Link forces:** See chapter 37 for discussion of these items
- **Section cut forces:** See chapter 39 for discussion of these items

The remainder of this chapter discusses the building modes, building modal info and building output items.

Building Modes

Checking the building modes check box provides output for all of the building modes. This is an all or nothing feature. You either get output for all of the modes or for none of the modes.

If you want output for a single mode then use the following process:

- Use the **Define menu > Load Combinations** command to define a new load combination. Click on the **Add New Combo** button to open the Load Combination Data dialog box.
- In the Define Combination area click the Case Name drop down box and select Mode. Then click the **Add** button. After you click the **Add** button ETABS prompts you for a mode number.
- Now when you print displacement output for this combination you are getting the mode shape associated with the specified mode number.

Building Modal Info

41

The following subsections describe the output that is provided when you check the Building Modal Info check box.

Modal Periods and Frequencies

The following time-properties are printed for each Mode:

- Period, T, in units of time.
- Cyclic frequency, f, in units of cycles per time; this is the inverse of T.
- Circular frequency, ω , in units of radians per time; $\omega = 2\pi f$.

Modal Participation Factors

The modal participation factors f_{uxn} , f_{uyn} , f_{uzn} , f_{rxn} , f_{ryn} and f_{rzn} are the dot products of the six unit Acceleration Loads with the mode shapes. The participation factors for Mode n corresponding to Acceleration Loads in the global UX, UY, UZ, RX, RY and RZ directions are given by:

$$f_{uxn} = \varphi_n^T m_{ux} \quad \text{Eqn. 41-1a}$$

$$f_{uyn} = \varphi_n^T m_{uy} \quad \text{Eqn. 41-1b}$$

$$f_{uzn} = \varphi_n^T m_{uz} \quad \text{Eqn. 41-1c}$$

$$f_{rxn} = \varphi_n^T m_{rx} \quad \text{Eqn. 41-1d}$$

$$f_{ryn} = \varphi_n^T m_{ry} \quad \text{Eqn. 41-1e}$$

$$f_{rzn} = \varphi_n^T m_{rz} \quad \text{Eqn. 41-1f}$$

where φ_n is the mode shape and m_{ux} , m_{uy} , m_{uz} , m_{rx} , m_{ry} and m_{rz} are the unit Acceleration Loads that are discussed in the section titled "Acceleration Loads" in Chapter 33. These factors are the generalized loads acting on the mode due to each of the Acceleration Loads. They are referred to the global coordinate system.

These values are called "factors" because they are related to the mode shape and to a unit acceleration. The modes shapes are each normalized, or scaled, with respect to the mass matrix such that:

$$\varphi_n^T M \varphi_n = 1 \quad \text{Eqn. 41-2}$$

The actual magnitudes and signs of the participation factors are not important. What is important is the relative values of the six factors for a given mode.

Modal Direction Factors

The modal direction factors provide a measure of how important a Mode is for computing the response to Acceleration Loads in a particular global direction. The modal direction factors d_{uxn} , d_{uyn} , d_{uzn} , d_{rxn} , d_{ryn} and d_{rzn} are the dot products of the six modal participation factors with the modes shapes. The modal direction factors for Mode n corresponding to Acceleration Loads in the global UX, UY, UZ, RX, RY and RZ directions are given by:

Note:

The modal direction factors provide a measure of how important a Mode is for computing the response to Acceleration Loads in a particular global direction.

$$d_{uxn} = \boldsymbol{\varphi}_n^T \mathbf{m}_{ux} \boldsymbol{\varphi}_n \quad \text{Eqn. 41-2a}$$

$$d_{uyn} = \boldsymbol{\varphi}_n^T \mathbf{m}_{uy} \boldsymbol{\varphi}_n \quad \text{Eqn. 41-2b}$$

$$d_{uzn} = \boldsymbol{\varphi}_n^T \mathbf{m}_{uz} \boldsymbol{\varphi}_n \quad \text{Eqn. 41-2c}$$

$$d_{rxn} = \boldsymbol{\varphi}_n^T \mathbf{m}_{rx} \boldsymbol{\varphi}_n \quad \text{Eqn. 41-2d}$$

$$d_{ryn} = \boldsymbol{\varphi}_n^T \mathbf{m}_{ry} \boldsymbol{\varphi}_n \quad \text{Eqn. 41-2e}$$

$$d_{rzn} = \boldsymbol{\varphi}_n^T \mathbf{m}_{rz} \boldsymbol{\varphi}_n \quad \text{Eqn. 41-2f}$$

The sum of the six modal direction factors for any mode is always 1.

41

Modal Effective Mass Factors

The modal effective mass factors for a Mode provide a measure of how important the Mode is for computing the response to the Acceleration Loads in each of the six global directions. Thus it is useful for determining the accuracy of response-spectrum analyses and seismic time-history analyses. The modal effective mass factors provide *no information* about the accuracy of time-history analyses subjected to other loads (other than the Acceleration Loads described in the section titled "Acceleration Loads" in Chapter 33).

The modal effective mass factors for Mode n corresponding to Acceleration Loads in the global UX, UY, UZ, RX, RY and RZ directions are given by:

$$r_{uxn} = \frac{(f_{uxn})^2}{M_{ux}} \quad \text{Eqn. 41-3a}$$

$$r_{uyn} = \frac{(f_{uyn})^2}{M_{uy}} \quad \text{Eqn. 41-3b}$$

$$r_{uzn} = \frac{(f_{uzn})^2}{M_{uz}} \quad \text{Eqn. 41-3c}$$

$$r_{rxn} = \frac{(f_{rxn})^2}{M_{rx}} \quad \text{Eqn. 41-3d}$$

$$r_{ryn} = \frac{(f_{ryn})^2}{M_{ry}} \quad \text{Eqn. 41-3e}$$

$$r_{rzn} = \frac{(f_{rzn})^2}{M_{rz}} \quad \text{Eqn. 41-3f}$$

41 where f_{uxn} , f_{uyn} , f_{uzn} , f_{rxn} , f_{ryn} and f_{rzn} are the participation factors defined in the subsection above titled "Modal Participation Factors"; and M_{ux} , M_{uy} , M_{uz} , M_{rx} , M_{ry} and M_{rz} are the total unrestrained masses acting in the UX, UY, UZ, RX, RY and RZ directions.

The cumulative sums of the modal effective mass factors for all Modes up to Mode n are printed with the individual values for Mode n. This provides a simple measure of how many Modes are required to achieve a given level of accuracy for ground-acceleration loading.

If all eigen Modes of the structure are present, the modal effective mass factors for each of the six Acceleration Loads should generally be 100%.

Static and Dynamic Load Participation Ratios

The static and dynamic load participation ratios provide a measure of how adequate the calculated modes are for representing the response to time-history analyses. These two measures are printed for each of the following spatial load vectors:

- The three unit Acceleration Loads
- All static Load Cases
- All nonlinear deformation loads

The first two represent spatial loads that you can explicitly specify in a time-history analysis, whereas the last represents loads that can act implicitly in a *nonlinear* time-history analysis.

The load participation ratios are expressed as percentages under the heading:

MODAL LOAD PARTICIPATION RATIOS .

Static Load Participation Ratio

The static load participation ratio measures how well the calculated modes can represent the response to a given static load. This measure was first presented by Wilson (1997). For a given spatial load vector p , the participation factor for Mode n is given by:

$$f_n = \varphi_n^T p \quad \text{Eqn. 41-4}$$

where φ_n is the mode shape (vector) of Mode n. This factor is the generalized load acting on the Mode due to load p . Note that f_n is just the usual participation factor when p is one of the unit acceleration loads.

The static participation ratio for this mode is given by:

$$r_n^S = \frac{\left(\frac{f_n}{\omega_n} \right)^2}{u^T p} \quad \text{Eqn. 41-5}$$

where \mathbf{u} is the static solution given by $\mathbf{Ku} = \mathbf{p}$. This ratio gives the fraction of the total strain energy in the exact static solution that is contained in Mode n. Note that the denominator can also be represented as $\mathbf{u}^T \mathbf{Ku}$.

Finally, the cumulative sum of the static participation ratios for all the calculated modes is also printed:

$$R^S = \sum_{n=1}^N r_n^S = \frac{\sum_{n=1}^N \left(\frac{\varphi_n^T \mathbf{p}}{\omega_n} \right)^2}{\mathbf{u}^T \mathbf{p}} \quad \text{Eqn. 41-6}$$

where N is the number of modes found. This value gives the fraction of the total strain energy in the exact static solution that is captured by the N modes.

When solving for static solutions using quasi-static time-history analysis, the value of R^S should be close to 100% for any applied static Loads, and also for all nonlinear deformation loads if the analysis is nonlinear.

Note that when Ritz-vectors are used, the value of R^S will always be 100% for all starting load vectors. This may not be true when eigenvectors are used. In fact, even using all possible eigenvectors will not give 100% static participation if load \mathbf{p} acts on any massless degrees-of-freedom.

Dynamic Load Participation Ratio

The dynamic load participation ratio measures how well the calculated modes can represent the response to a given dynamic load. This measure was developed for ETABS and SAP2000, and it is an extension of the concept of participating mass ratios. It is assumed that the load acts only on degrees of freedom with mass. Any portion of load vector \mathbf{p} that acts on massless degrees of freedom cannot be represented by this measure and is ignored in the following discussion.

For a given spatial load vector \mathbf{p} , the participation factor for Mode n is given by Equation 41-4.

The dynamic participation ratio for this mode is given by:

$$r_n^D = \frac{f_n^2}{a^T p} \quad \text{Eqn. 41-7}$$

where a is the acceleration given by $Ma = p$. The acceleration a is easy to calculate since M is diagonal. The values of a and p are taken to be zero at all massless degrees of freedom. Note that the denominator can also be represented as $a^T Ma$

Finally, the cumulative sum of the dynamic participation ratios for all the calculated modes is also printed:

$$R^D = \sum_{n=1}^N r_n^D = \frac{\sum_{n=1}^N \left(\frac{\varphi_n^T p}{\omega_n} \right)^2}{a^T p} \quad \text{Eqn. 41-8}$$

where N is the number of modes found. When p is one of the unit acceleration loads, r^D is the usual mass participation ratio, and R^D is the usual cumulative mass participation ratio.

When R^D is 100%, the calculated modes should be capable of exactly representing the solution to any time-varying application of spatial load p . If R^D is less than 100%, the accuracy of the solution will depend upon the frequency content of the time-function multiplying load p . Normally it is the high frequency response that is not captured when R^D is less than 100%.

The dynamic load participation ratio only measures how the modes capture the spatial characteristics of p , not its temporal characteristics. For this reason, R^D serves only as a qualitative guide as to whether enough modes have been computed. You must still examine the response to each different dynamic loading with varying number of modes to see if enough modes have been used.

Final Comments on Static and Dynamic Load Participation Ratios

Note:

In a nonlinear dynamic analysis you should always have enough modes to at least get 100% static load participation ratios for your link elements.

The static and dynamic load participation ratios apply to nonlinear dynamic analysis. The static load participation ratios provide you with a measure of how well the modes used have captured the forces and displacements in the link elements. When the static load participation ratio is 100% for a link element then the modes used have captured the force and deformation in that link element. You should always be sure that you have a 100% static load participation ratio for all of your link elements.

The dynamic load participation ratios provide you with a measure of how well the modes used have captured the forces and in the elements directly around a link element. When the dynamic load participation ratio is 100% for a link element then the modes used have captured the forces in the elements adjacent to the link element.

You should strive to be have the dynamic load participation ratio as high as possible for your link elements. If the dynamic load participation ratio is low then local equilibrium at the link element may not be satisfied. In other words, the forces in the elements right around the link element may not balance with those in the link element.

Damping and Accelerations

This information is printed for each response spectrum analysis. The modal damping and the ground accelerations acting in each direction are printed for every Mode under the heading:

RESPONSE SPECTRUM ACCELERATIONS

The damping value printed for each Mode is the sum of the specified CQC or GMC damping ratio plus the modal damping contributed by effective damping in the link elements, if any.

The accelerations printed for each Mode are the actual values as interpolated at the modal period from the response-spectrum curves after scaling by the input scale factors. The accelerations are always referred to the local axes of the response-spectrum analysis. They are identified in the output as U1, U2, and U3.

See the section titled "Link Properties" and the subsection titled "Structural and Function Damping" both in Chapter 11 for information on the modal damping.

Modal Amplitudes

This information is printed for each response spectrum analysis. The response-spectrum modal amplitudes give the multipliers of the mode shapes that contribute to the displaced shape of the structure for each direction of Acceleration. For a given Mode and a given direction of acceleration, this is the product of the modal participation factor and the response-spectrum acceleration, divided by the eigenvalue, ω^2 , of the Mode. These values are printed under the heading:

RESPONSE SPECTRUM MODAL AMPLITUDES

The acceleration directions are always referred to the local axes of the response-spectrum analysis. They are identified in the output as U1, U2, and U3.

Base Reactions

This information is printed for each response spectrum analysis. The base reactions are the total forces and moments about the global origin required of the supports (Restraints and Springs) to resist the inertia forces due to response-spectrum loading. These are printed in the output file under the heading:

RESPONSE SPECTRUM BASE REACTIONS

These are printed separately for each individual Mode and each direction of loading without any combination. The total response-spectrum reactions are then printed after performing modal combination and directional combination.

The reaction forces and moments are always referred to the local axes of the response-spectrum analysis. They are identified in the output as F1, F2, F3, M1, M2, and M3.

Building Output

The following subsections describe the output that is provided when you check the Building Output check box.

Cumulative Center of Mass

The cumulative center of mass data provides a value of mass and center of mass at each story level. The mass value reported for a story level is the sum of the mass at that story level plus all mass above the story level. Similarly, the center of mass reported is the center of mass of the mass at that story level plus all mass above the story level.

Center of Rigidity

Note:

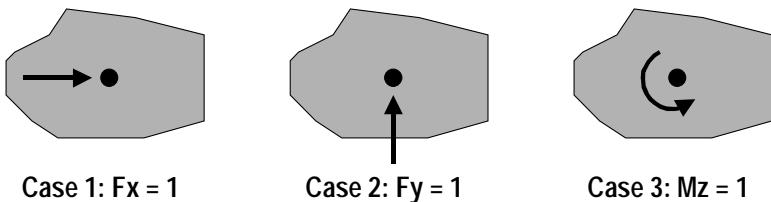
When translational lateral loads are applied at the center of rigidity of a particular floor diaphragm as specified in the ETABS output, with no loads applied to any of the other floor diaphragms, the displacements of that diaphragm will have only translational components with no rotations.

The original concept of center of rigidity dates back to manual rigidity analysis techniques associated with the lateral analysis of single story shear wall buildings. The center of rigidity was defined as the location of the centroid of the stiffnesses of single story lateral resisting elements (typically planar) arbitrarily located in plan. For single story structures the definition worked well because the stiffness for each wall frame was a 1 by 1 matrix with no interstory coupling or compatibility factors to complicate the problem. The analysis technique was extrapolated for multistory lateral analysis whereby multistory buildings were analyzed as a series of single story buildings stacked over one another with no interstory displacement compatibility. Needless to say, for complex three dimensional structures this assumption was approximate at best.

Modern computer techniques do not require the explicit evaluation of the center of rigidity. However, the center of rigidity still needs to be evaluated because some building codes refer to it as a reference point to define design eccentricity requirements in multistory buildings.

In the general three-dimensional analysis of a building, where the behavior is coupled in plan as well as through the height of the structure, the center of rigidity requires a broader definition. In this broader definition when translational lateral loads are applied at the center of rigidity of a particular floor diaphragm,

Figure 41-1:
Three load cases
used to determine
the center of rigidity



with no loads applied to any of the other floor diaphragms, the displacements of that diaphragm will have only translational components with no rotations. It should be noted that the resulting displacements of the diaphragms at other levels in general will contain translational as well as rotational components.

To evaluate the center of rigidity at a particular diaphragm the structure is analyzed for three load cases. The loads are applied at the center of mass (or any arbitrary point). Load case 1 has a unit load applied in the global X direction and results in a diaphragm rotation of R_{zx} . Load case 2 has a unit load applied in the global Y direction and results in a diaphragm rotation of R_{zy} . Load case 3 has a unit moment applied about the global Z-axis giving a diaphragm rotation of R_{zz} . Figure 41-1 illustrates the three load cases.

The center of rigidity relative to the center of mass (or the arbitrary point) is then given by the coordinates (X, Y), where $X = -R_{zy} / R_{zz}$ and $Y = R_{zx} / R_{zz}$. This point is a function of the structural properties and is independent of any loading.

41

As described above, the determination of the center of rigidity can be numerically intensive as it involves a static analysis of the whole structure for three independent load cases for each floor diaphragm. This procedure is implemented in ETABS so that the centers of rigidity for every diaphragm are automatically calculated as part of the solution process.

Story Shears and Overturning Moments

Story shears and overturning moments are reported in the global coordinate system as P , V_X , V_Y , T , M_X and M_Y . The forces are reported at the top of the story, just below the story level itself, and at the bottom of the story, just above the story level below.

The sign convention for story level forces is exactly the same as that for frame elements, shown in Figure 35-1 in Chapter 35 with the bottom of the story corresponding to the i-end of the frame element and the top of the story corresponding to the j-end of the frame element.



Chapter 42

Database Input/Output Tables

General

You can use the **File menu > Export > Save Input/Output as Access Database File** command to save input data and analysis output data in a Microsoft Access 97 database (*.mdb) file. The data is saved in tables in the database file.

42

If you have Microsoft Access, or some other database that can read Microsoft Access 97 files then you can open this database file and manipulate the input and output data any way you please. You can also create your own customized output formats in this way.

All input data that can be printed using the **File menu > Print Tables > Input** command is saved in a tabular form in the database file. Similarly, all analysis output data that can be printed using the **File menu > Print Tables > Analysis Output** command is saved in a tabular form in the database file. The format of the database tables is similar to those printed from the File menu.

Refer to the documentation for Printed Output Tables in Chapter 41 for information on the analysis output data that is provided in the database file.

Note that you must have saved your file (.edb) at least once prior to creating the database file. If you have not saved your file and you use the **File menu > Export > Save Input/Output as Access Database File** command then ETABS will first bring up the Save Model File As dialog box so that you can save your model. Then it will bring up the Save Access Database File As dialog box so that you can save the database.



Chapter 43

The ETABS Log and Out Files

General

This chapter discusses the contents of the ETABS .log and .out files.

The ETABS Log File

43

The **analysis log file**, more simply called the **log file**, is created during the analysis phase of the program. It contains analysis statistics and results summaries, as well as the details of any warnings or errors detected during this phase.

It is *strongly recommended* that you carefully review the contents of the log file for errors, warnings, and the results summaries, including the global force balance relative errors.

If *errors* are present, the analysis phase of the program will have terminated before completion. If only *warnings* are present, the analysis phase should have completed. In either case, you should

check these errors and warnings carefully before examining the results of the analysis.

The analysis phase is divided into many sub-phases. The time and date at the beginning and end of each of these sub-phases is reported. The date is given in year/month/day format, and the time is given in hour:minute:second format.

**Note:**

The information that scrolls by in the Analysis Window as the analysis runs is saved in the ETABS .log file.

The following additional information is provided in the analysis log file:

- The amount of memory allocated for data storage, in bytes.
- The number of elements formed.
- The number of diaphragm constraints formed.
- The number of equations formed, the size of the stiffness matrix, and the number of loads solved.
- The number of natural frequencies of the model below frequency **shift** if eigenvector analysis is performed with a frequency shift. See the subsection title "Frequency Range" in Chapter 33 for more information.
- For modal analysis, the control parameters and the number of Modes found.
- For eigenvector analysis, the number of iterations performed.
- For Ritz-vector analysis, the number of generation cycles performed and the reasons for discarding the starting load vectors (if any).
- For response-spectrum analysis, the number of analyses performed.
- For each time-history analysis, the control parameters and the number of time steps completed.

- For each nonlinear time-history analysis, the number of substeps and iterations completed and other statistics.
- The global force balance relative errors. See the section titled "Global Force Balance" later in this chapter for more information.

The ETABS Out File

The following information is provided in the ETABS .out file.

- Diaphragm constraint masses and center of mass location.
- Displacement degrees of freedom are listed in a tabular form
- Global force balance. See the section titled "Global Force Balance" later in this chapter.

Global Force Balance



Tip:

You should always review the global force balance as a check on the validity of the structural model and the analysis.

43

For each Analysis Case, the sum of all joint forces and moments acting on the structure should be in equilibrium. The program computes and prints a global force balance for the following types of Analysis Cases: static loads, modes, Response spectrum cases, and load combinations. No global force balance is computed for time histories, nonlinear static load cases, or load combinations. **You should always review the global force balance as a check on the validity of the structural model and the analysis.**

For each Analysis Case, the program computes the resultants at the global origin for all joint forces and moments acting on the structure. Separate resultants are computed for each type of joint force:

- Applied loads

- Inertial loads
- Spring forces
- Link forces
- Restraint Forces (Reactions)
- Diaphragm constraint forces
- P-Delta forces

Each force resultant is computed as the sum of the forces acting on all joints in the structure. Each moment resultant is computed as the sum of the moments acting on all joints in the structure, plus the moments about the origin of the forces acting on all joints in the structure. This results in three force and three moment components, all referred to the global coordinate system.

The resultants are also computed for the total of all these different forces and moments acting on the structure. These total resultants should be zero if the structure is in exact equilibrium. Due to the approximate nature of computer arithmetic, the totals may not be exactly zero. However, their values relative to the magnitude of the contributing forces gives a measure of the accuracy and stability of the solution.

The resultant forces and moments are always printed in the .out file for all Analysis Cases under the heading:

GLOBAL FORCE BALANCE

In addition, relative equilibrium errors are printed in the log file under the heading:

GLOBAL FORCE BALANCE RELATIVE ERRORS

Here the total force and moment components are expressed as a percentage of the maximum possible equilibrium error. The maximum possible error is computed as follows:

- For each component (FX, FY, FZ, MX, MY, and MZ), the absolute values of the resultants for applied loads, inertial loads, spring forces, link forces, reactions, constraint forces, and P-Delta forces are summed.

- The maximum of the absolute sums for FX, FY, and FZ is determined.
- The maximum of the absolute sums for MX, MY, and MZ is determined.
- The maximum possible error for the force components is the maximum of the absolute force sums, or the maximum of the absolute moment sums divided by the average moment arm for the structure, whichever is larger.
- The maximum possible error for the moment components is the maximum of the absolute moment sums, or the maximum of the absolute force sums multiplied by the average moment arm for the structure, whichever is larger.

This definition, while complicated, helps assure that only numerically meaningful equilibrium errors are indicated as such.

It is strongly recommended that you always review the global force balance in the output file and the relative equilibrium errors in the log file. These results can alert you to potential problems with the structural model or the analysis. In particular, you should look for the following:

- The accuracy of the computer calculations is on the order of 10^{-15} , or 10^{-13} percent. If the relative force or moment error for the Load Cases is significantly larger than this, it may indicate that the structure is unstable or that the stiffness matrix is ill-conditioned.
- You can expect somewhat larger equilibrium errors for Vibration Modes than for Load Cases. For Eigen Modes, the errors generally reflect the convergence tolerance and can be reduced by using a smaller tolerance. For Ritz Modes, the errors indicate the fact that the Ritz modes are not the true Eigen Modes of the structure. These errors can generally be reduced by requesting more modes. Some Modes with large errors may be orthogonal to the Ritz starting vectors and have no effect upon response-spectrum and time-history analyses.

- The diaphragm constraint forces should be self-equilibrating, i.e., the resultant constraint forces and moments should be essentially zero. If the values are significantly different from zero (compared to the other resultants), the constraints may be poorly defined. An example of this is a diaphragm constraint whose points are not all in the same plane. In-plane forces in such a diaphragm can cause moments that are not captured by the structural model. Such errors in the constraints affect the validity of the model even though the overall equilibrium of the structure may be satisfied.

**Note:**

A high relative equilibrium error is not always indicative of a problem in your model. It simply indicates the possibility of a problem that should be investigated further.

Important Note: The relative equilibrium errors are a good first check for possible problems in your model. However, high relative equilibrium errors are not always indicative of a problem with your model. For example, suppose that a total force component in your model is equal to 1E-10 and that the maximum possible error for that component is also 1E-10. In this case ETABS reports the relative equilibrium error as 100%. Even though the relative equilibrium error is 100%, the maximum possible error of 1E-10 is in general not a problem.

When you see a high relative equilibrium error you should take it as a clue to investigate further by looking at the global force balance to see if a problem really exists. Again a high relative equilibrium error in itself does not always mean there is a problem in your model.



Inserting ETABS Output into Written Reports

General

In some instances you may want to include some output from ETABS in your written reports. This chapter provides a few ideas for how you might do this. Both tabular and graphical types of output are discussed.

44

Tabular Output

Note:

You can insert tabular and graphical output from ETABS into your written reports.



The simplest and most straight forward way to include tabular output from ETABS in your written reports is to print the desired output to a file and then cut the data from the file and paste it into your report. You can then reformat it if necessary. Use the **File menu > Print Tables** command to initially print out the output. Be sure to check the Print to File check box when selecting the data to be output.

A second more powerful way to bring data into your reports is to use the **File menu > Export > Save Input/Output as Access**

**Tip:**

You can create customized tabular output for your reports using Microsoft Access.

Database File command to save the input and output for your model in a series of tables in a Microsoft Access .mdb file (compatible with Microsoft Access 97). If you are familiar with Microsoft Access, and you have an available copy of the program, then you can then create your own customized reports (output) within Microsoft Access. These customized reports can then be incorporated into your written report.

Graphical Output

You can use the **File menu > Export > Save Graphics as Enhanced Metafile** command to save the graphics displayed in the currently active window to a Windows enhanced metafile (.emf) file. You can then insert this picture directly into your report. Alternatively, you may want to insert the picture into a graphics program (e.g., Microsoft PowerPoint) and add some annotations or make some other changes before pasting it into your report.

An alternative method of getting graphical output for your reports is to use the built-in Windows screen capture features. If you press the Print Screen key on your keyboard then Windows copies the entire screen as a picture to the clipboard. You can then paste this picture into a graphics program or even directly into your report. If you press the Alt key and the Print Screen key simultaneously on your keyboard, then Windows copies the active window on the screen to the clipboard as a picture.

Some of the (black and white) figures in this manual that show pictures of the ETABS interface or pictures of ETABS models were created using the following technique:

1. Click the **View menu > Set Building View Options** command and select the White Background, Black Objects option in the "View by Colors of" area of the dialog box. (This option is useful if you want to capture black and white pictures. If you want your pictures in color then you may want to use the **Options > Colors > Display** command to change the background color, perhaps to white).
2. Set the ETABS display up as you want it to appear in the picture.

3. Press the Alt key and the Print Screen key simultaneously on your keyboard to copy the screen image to the clipboard as a picture.
4. Paste the picture from the clipboard into a graphics program. (We used Microsoft PowerPoint).
5. If necessary crop and/or resize the picture.
6. In some cases add annotations to the picture.
7. Cut and paste the completed figure from the graphics program into your report as a picture.



Chapter 45

Steel Frame Design

Any line object that ETABS has assigned a Steel Frame design procedure can be designed in the Steel Frame Design postprocessor. See the section titled "ETABS Default Design Procedure Assignments" in Chapter 17 for more information.

This section describes the intended steel frame design procedure and the menu commands available for steel frame design. Before describing the design procedure or menu items it is important that you understand the distinction between analysis sections and design sections. This is discussed first.

45

Analysis Sections and Design Sections

Analysis sections are those section properties used to analyze the model when you click the **Analyze menu > Run Analysis** command. The design section is whatever section has most currently been designed and thus designated the current design section.

It is possible for the last used analysis section and the current design section to be different. For example you may have run your analysis using a W18X35 beam and then found in the design that a W16X31 beam worked. In this case the last used analysis section is the W18X35 and the current design section is the W16X31. Before you complete the design process you want to make sure that the last used analysis section and the current design section is the same. The **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command, which is useful for this task, is discussed more in later subsections.

ETABS keeps track of the analysis section and the design section separately. Note the following about analysis and design sections:

- Anytime you assign a line object a frame section property using the **Assign menu > Frame/Line > Frame Section** command ETABS assigns this section as both the analysis section and the design section.
- Whenever you run an analysis using the **Analyze menu > Run Analysis** command (or its associated toolbar button) ETABS always sets the analysis section to be the same as the current design section.
- When you use the **Assign menu > Frame/Line > Frame Section** command to assign an auto select list to a frame section ETABS initially sets the design section to be the section with the median weight in the auto select list.
- Anytime you unlock your model ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Design menu > Steel Frame Design > Select Design Combo** command to change a design load combination ETABS deletes your design results but it does not delete or change the design section.



Note:

Any time you unlock your model your design results (and analysis results) are deleted.

- Anytime you use the **Define menu > Load Combinations** command to change a design load combination ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Options menu > Preferences > Steel Frame Design** command to change any of the steel frame design preferences ETABS deletes your design results but it does not delete or change the design section.
- Anytime you do something that causes your static nonlinear analysis results to be deleted then the design results for any load combination that includes static nonlinear forces are also deleted. Typically your static nonlinear analysis and design results are deleted when you do one of the following:
 - ✓ Use the **Define menu > Frame Nonlinear Hinge Properties** command to redefine existing or define new hinges.
 - ✓ Use the **Define menu > Static Nonlinear/Pushover Cases** command to redefine existing or define new static nonlinear load cases.
 - ✓ Use the **Assign menu > Frame/Line > Frame Nonlinear Hinges** to add or delete hinges.

Again note that this only deletes results for load combinations that include static nonlinear forces.

Steel Frame Design Procedure

Following is a typical steel frame design process that might occur for a new building. Note that the sequence of steps you may take in any particular design may vary from this but the basic process will probably be essentially the same.

1. Use the **Options menu > Preferences > Steel Frame Design** command to choose the steel frame design code and to review other steel frame design preferences and revise them if necessary. Note that there are default values provided for

all steel frame design preferences so it is not actually necessary for you to define any preferences unless you want to change some of the default preference values.

2. Create the building model. See the section titled "Modeling Process" in Chapter 6 for more information.
3. Run the building analysis using the **Analyze menu > Run Analysis** command.
4. Assign steel frame overwrites, if needed, using the **Design menu > Steel Frame Design > View/Revise Overwrites** command. Note that you must select frame elements first before using this command. Also note that there are default values provided for all steel frame design overwrites so it is not actually necessary for you to define any overwrites unless you want to change some of the default overwrite values.
5. Designate design groups, if desired, using the **Design menu > Steel Frame Design > Select Design Group** command. Note that you must have already created some groups by selecting objects and clicking the **Assign menu > Group Names** command.
6. If you want to use any design load combinations other than the default ones created by ETABS for your steel frame design then click the **Design menu > Steel Frame Design > Select Design Combo** command. Note that you must have already created your own design combos by clicking the **Define menu > Load Combinations** command.
7. Designate lateral displacement targets for various load cases using the **Design menu > Steel Frame Design > Set Lateral Displacement Targets** command.
8. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to run the steel frame design.



Note:

Steel frame design is an iterative process. You must run the analysis and design multiple times to complete the design process.

9. Review the steel frame design results. To do this you might do one of the following:
 - a. Click the **Design menu > Steel Frame Design > Display Design Info** command to display design input and output information on the model.
 - b. Right click on a frame element while the design results are displayed on it to enter the interactive design mode and interactively design the frame element. Note that while you are in this mode you can revise overwrites and immediately see the results of the new design.

If you are not currently displaying design results you can 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.

- c. Use the **File menu > Print Tables > Steel Frame Design** command to print steel frame design data. If you select a few frame elements before using this command then data is printed only for the selected elements.
10. Use the **Design menu > Steel Frame Design > Change Design Section** command to change the design section properties for selected frame elements.
11. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to rerun the steel frame design with the new section properties. Review the results using the procedures described above.
12. Rerun the building analysis using the **Analyze menu > Run Analysis** command. Note that the section properties used for the analysis are the last specified design section properties.
13. Compare your lateral displacements with your lateral displacement targets.

14. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to rerun the steel frame design with the new analysis results and new section properties. Review the results using the procedures described above.
15. Again use the **Design menu > Steel Frame Design > Change Design Section** command to change the design section properties for selected frame elements, if necessary.
16. Repeat the process in steps 12, 13, 14 and 15 as many times as necessary.
17. Select all frame elements and click the **Design menu > Steel Frame Design > Make Auto Select Section Null** command. This removes any auto select section assignments from the selected frame elements (if they have the Steel Frame design procedure).
18. Rerun the building analysis using the **Analyze menu > Run Analysis** command. Note that the section properties used for the analysis are the last specified design section properties.
19. Verify that your lateral displacements are within acceptable limits.
20. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to rerun the steel frame design with the new section properties. Review the results using the procedures described above.
21. Click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command to verify that all of the final design sections are the same as the last used analysis sections.
22. Use the **File menu > Print Tables > Steel Frame Design** command to print selected steel frame design results if desired.

It is important to note that design is an iterative process. The sections that you use to run your original analysis are not typically the same sections that you end up with at the end of the design process. You always want to be sure to run a building

analysis using your final frame section sizes and then run a design check based on the forces obtained from that analysis. The **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command is useful for making sure that the design sections are the same as the analysis sections.

The following section describes the menu items available on the **Design menu > Steel Frame Design** submenu.

Steel Frame Design Menu Commands

This section describes each of the steel frame design menu commands that are available in ETABS. You can find these commands by clicking **Design menu > Steel Frame Design**.

Select Design Group

Note:

Frame elements designed as a group are all given the same section size



Tip:

Frame elements 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 frame elements and clicking the **Assign menu > Group Names** command.
- After the group is 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.

Select Design Combo

Click the **Design menu > Steel Frame Design > Select Design Combo** command to open the Design Load Combinations Selection dialog box. Here you can review the default steel frame design load combinations defined by ETABS and/or you can designate your own design load combinations.

In the dialog box 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 dialog box for use with the **Add** and **Remove** buttons, if desired.

The default steel frame design load combinations have names like DSTL1, etc.

View/Revise Overwrites

Tip:

The steel frame design overwrites only apply to the frame elements that they are specifically assigned to.

Use the **Design menu > Steel Frame Design > View/Revise Overwrites** command to review and/or change the steel frame overwrites. You may not need to assign any steel frame overwrites; however the option is always available to you.

The steel frame design overwrites are basic properties that apply only to the frame elements that they are specifically assigned to. 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).

You can 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 then no changes are made to the item. This is true whether you have one frame element selected or multiple frame elements selected.

Set Lateral Displacement Targets

Click the **Design menu > Steel Frame Design > Set Lateral Displacement Targets** command to open the Displacement Optimization dialog box 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 dialog box you 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 these are specified to your satisfaction in the Loads area then click the **Add** button. You can specify as many load cases and displacements as you want.

If you want to modify an existing displacement optimization specification then click on it in the Loads area to highlight it. Note that the data for the specification appears in the boxes at the top of the Loads area. Modify the data in the boxes as desired. Then click the **Modify** button.

45



Tip:

Frame elements must have auto select list assignments to be designed using displacement optimization.

If you want to delete an existing displacement optimization specification then click on it in the Loads area to highlight it. Note that the data for the specification appears in the boxes at the top of the Loads area. Then click the **Delete** button.

Note the following about the displacement optimization performed by ETABS:

- ETABS 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 per-

centage 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 ETABS 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 you do not have the appropriate sections in your auto select lists then you may never reach your displacement target no matter how many times you iterate by rerunning the analysis and design.

Start Design/Check of Structure

To run a steel frame design simply 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 there are no frame elements with the Steel Frame design procedure in the model.

If you have selected frame elements when you click this command then only the selected frame elements are designed. If no frame elements are selected when you click this command then all steel frame elements with the Steel Frame design procedure are designed.

Interactive Steel Frame Design

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 you can 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.

See the section titled "ETABS Interactive Steel Frame Design" later in this chapter for more information.

Display Design Info

You can review some of the results of the steel frame design directly on the ETABS 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.

Make Auto Select Section Null

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 done with actual frame sections assigned, not auto select sections.

45

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 frame element.

The **Make Auto Select Section Null** command only works on a selection that you make. Thus you should select the elements whose auto select sections are to be made null prior to executing this command. If you do not select any elements this command will not be available. Often you may want to select all elements prior to 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 then use the **Assign menu > Frame/Line > Frame Section** command to change the section property.

Change Design Section

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.

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 whose design sections are to be changed prior to 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 re-running the analysis.



Reset Design Section to Last Analysis

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 you should select the elements whose design sections are to be reset prior to implementing this command. If you do not select any elements this command will not be available.

Verify Analysis vs Design Section

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 then 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 ETABS 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 ask it to. 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.

Reset All Steel Overwrites

The **Design menu > Steel Frame Design > Reset All Steel Overwrites** command resets the overwrites for all frame sections with the Steel Frame design procedure back 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 ETABS database (*.edb) file.

Delete Steel Design Results

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 ETABS database (*.edb) file. Note that deleting your steel design results does not delete your current design section (next analysis section).

ETABS Interactive Steel Frame Design

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 in the Steel Stress Check Information dialog box. If you are not currently displaying design results you can 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.

The following bullet items describe the features that are included in the Steel Stress Check Information dialog box.

- **Story ID:** This is the story level ID associated with the frame element.
- **Beam:** This is the label associated with a frame element that is a beam. If the frame element is a column then this item is titled Column. If the frame element is a brace then this item is titled Brace.
- **Column:** This is the label associated with a frame element that is a column. If the frame element is a beam then this item is titled Beam. If the frame element is a brace then this item is titled Brace.
- **Brace:** This is the label associated with a frame element that is a brace. If the frame element is a beam then this item is titled Beam. If the frame element is a column then this item is titled Column.
- **Analysis section:** This is the section property that was used for this frame element in the last analysis. Thus the design forces are based on a frame element of this section property. For your final design iteration the Current Design/Next Analysis section property and the Last Analysis section property should be the same.
- **Design section:** This is the current design section property. If the frame element is assigned an auto select list then the section displayed in this dialog box initially defaults to the optimal section.

If there is no auto select list assigned to the frame element then the element design is done for the section property specified in this edit box.

45

It is important to note that subsequent analyses use the section property specified in this list box for the next analysis section for the frame element. Thus the forces and moments obtained in the next analysis will be based on this section.

You can change the Current Design/Next Analysis section property in the overwrites. Click the **Overwrites** button to do this.



Tip:

The section property displayed for the Current Design/Next Analysis item is used by ETABS as the section property for the next analysis run.

- **Table showing stress details:** This table shows the stress ratios obtained for each design load combination at each output station along the frame element. Initially the worst stress ratio is highlighted.

Following are the headings in the table:

- ✓ **Combo ID:** This is the name of the design load combination considered.
- ✓ **Station location:** This is the location of the station considered, measured from the i-end of the frame element.
- ✓ **Ratio:** This is the total PMM stress ratio for the element. When stress ratios are reported for this item they are followed by either (T) or (C). The (T) item indicates that the axial component of the stress ratio is tension. The (C) item indicates that the axial component of the stress ratio is compression. Note that typically the interaction formulas are different depending on whether the axial stress is tension or compression.
- ✓ **AxL:** This is the axial component of the PMM stress ratio.
- ✓ **B-Maj:** This is the bending component of the PMM stress ratio for bending about the major axis. See the subsection titled "Local Axes Assignments to Line Objects" in Chapter 14 for a definition of the major axis.
- ✓ **B-Min:** This is the bending component of the PMM stress ratio for bending about the minor axis. See the subsection titled "Local Axes Assignments to Line Objects" in Chapter 14 for a definition of the minor axis.

- ✓ **Maj Shr Ratio:** This is the shear stress ratio for shear acting in the major direction of the frame element. See the subsection titled "Local Axes Assignments to Line Objects" in Chapter 14 for a definition of the major direction.
- ✓ **Min Shr Ratio:** This is the shear stress ratio for shear acting in the minor direction of the frame element. See the subsection titled "Local Axes Assignments to Line Objects" in Chapter 14 for a definition of the minor direction.



- Note:**

You can change the steel frame design overwrites and instantaneously see the new design results.

- **Details button:** Clicking this button displays design details for the frame element. You can print this information by selecting Print from the File menu that appears at the top of the window displaying the design details.
- **Overwrites button:** You can click this button to access and make revisions to the steel frame overwrites and then immediately see the new design results. If you modify some overwrites in this mode and you exit both the Steel Frame Design Overwrites form and the Steel Stress Check Information dialog box by clicking their respective **OK** buttons then the changes you made to the overwrites are permanently saved.

When you exit the Steel Frame Design Overwrites form by clicking the **OK** button the changes are temporarily saved. If you then exit the Steel Stress Check Information dialog box by clicking the **Cancel** button the changes you made to the steel frame overwrites are considered temporary only and are not permanently saved. Permanent saving of the overwrites does not actually occur until you click the **OK** button in the Steel Stress Check Information dialog box, not the Steel Frame Design Overwrites dialog box.



Chapter 46

Concrete Frame Design

Note:

A concrete frame element is designed as either a beam or a column depending on how its frame section property was designated when it was defined using the **Define menu > Frame Sections** command.

Any line object that ETABS has assigned a Concrete Frame design procedure can be designed in the Concrete Frame Design postprocessor. See the section titled "ETABS Default Design Procedure Assignments" in Chapter 17 for more information.

The concrete frame design postprocessor can design or check concrete columns and can design concrete beams.

Important Note: A concrete frame element is designed as either a beam or a column depending on how its frame section property was designated when it was defined using the **Define menu > Frame Sections** command. Note that when using this command, once you specify that a section has a concrete material property you can click on the **Reinforcement** button and specify whether it is a beam or a column. See the section titled "Reinforcing for Concrete Frame Section Properties" in Chapter 11 for more information.

This section describes the intended concrete frame design procedure and the menu commands available for concrete frame design. Before describing the design procedure or menu items it is important that you understand the distinction between analysis sections and design sections. This is discussed first.

Analysis Sections and Design Sections

Analysis sections are those section properties used to analyze the model when you click the **Analyze menu > Run Analysis** command. The design section is whatever section has most currently been designed and thus designated the current design section.

It is possible for the last used analysis section and the current design section to be different. For example you may have run your analysis using a 12 inch by 24 inch beam and then found in the design that a 12 inch by 18 inch beam worked. In this case the last used analysis section is the 12 inch by 24 inch beam and the current design section is the 12 inch by 18 inch beam. Before you complete the design process you want to make sure that the last used analysis section and the current design section is the same. The **Design menu > Concrete Frame Design > Verify Analysis vs Design Section** command, which is useful for this task, is discussed more in later subsections.

Note:

Any time you unlock your model your design results (and analysis results) are deleted.

ETABS keeps track of the analysis section and the design section separately. Note the following about analysis and design sections:

- Any time you assign a line object a frame section property using the **Assign menu > Frame/Line > Frame Section** command ETABS assigns this section as both the analysis section and the design section.
- Whenever you run an analysis using the **Analyze menu > Run Analysis** command (or its associated toolbar button) ETABS always sets the analysis section to be the same as the current design section.

- Anytime you unlock your model ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Design menu > Concrete Frame Design > Select Design Combo** command to change a design load combination ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Define menu > Load Combinations** command to change a design load combination ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Options menu > Preferences > Concrete Frame Design** command to change any of the concrete frame design preferences ETABS deletes your design results but it does not delete or change the design section.
- Anytime you do something that causes your static nonlinear analysis results to be deleted then the design results for any load combination that includes static nonlinear forces are also deleted. Typically your static nonlinear analysis and design results are deleted when you do one of the following:
 - ✓ Use the **Define menu > Frame Nonlinear Hinge Properties** command to redefine existing or define new hinges.
 - ✓ Use the **Define menu > Static Nonlinear/Pushover Cases** command to redefine existing or define new static nonlinear load cases.
 - ✓ Use the **Assign menu > Frame/Line > Frame Nonlinear Hinges** to add or delete hinges.

Again note that this only deletes results for load combinations that include static nonlinear forces.

Concrete Frame Design Procedure

Following is a typical concrete frame design process that might occur for a new building. Note that the sequence of steps you may take in any particular design may vary from this but the basic process will probably be essentially the same.

1. Use the **Options menu > Preferences > Concrete Frame Design** command to choose the concrete frame design code and to review other concrete frame design preferences and revise them if necessary. Note that there are default values provided for all concrete frame design preferences so it is not actually necessary for you to define any preferences unless you want to change some of the default preference values.
2. Create the building model. See the section titled "Modeling Process" in Chapter 6 for more information.
3. Run the building analysis using the **Analyze menu > Run Analysis** command.
4. Assign concrete frame overwrites, if needed, using the **Design menu > Concrete Frame Design > View/Revise Overwrites** command. Note that you must select frame elements first before using this command. Also note that there are default values provided for all concrete frame design overwrites so it is not actually necessary for you to define any overwrites unless you want to change some of the default overwrite values.
5. If you want to use any design load combinations other than the default ones created by ETABS for your concrete frame design then click the **Design menu > Concrete Frame Design > Select Design Combo** command. Note that you must have already created your own design combos by clicking the **Define menu > Load Combinations** command.
6. Click the **Design menu > Concrete Frame Design > Start Design/Check of Structure** command to run the concrete frame design.



Note:

Concrete frame design is an iterative process. You must run the analysis and design multiple times to complete the design process.

7. Review the concrete frame design results. To do this you might do one of the following:
 - a. Click the **Design menu > Concrete Frame Design > Display Design Info** command to display design input and output information on the model.
 - b. Right click on a frame element while the design results are displayed on it to enter the interactive design mode and interactively design the frame element. Note that while you are in this mode you can revise overwrites and immediately see the results of the new design.

If you are not currently displaying design results you can 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.

 - c. Use the **File menu > Print Tables > Concrete Frame Design** command to print concrete frame design data. If you select a few frame elements before using this command then data is printed only for the selected elements.
 8. Use the **Design menu > Concrete Frame Design > Change Design Section** command to change the design section properties for selected frame elements.
 9. Click the **Design menu > Concrete Frame Design > Start Design/Check of Structure** command to rerun the concrete frame design with the new section properties. Review the results using the procedures described above.
 10. Rerun the building analysis using the **Analyze menu > Run Analysis** command. Note that the section properties used for the analysis are the last specified design section properties.
 11. Click the **Design menu > Concrete Frame Design > Start Design/Check of Structure** command to rerun the concrete frame design with the new analysis results and new section properties. Review the results using the procedures described above.

12. Again use the **Design menu > Concrete Frame Design > Change Design Section** command to change the design section properties for selected frame elements, if necessary.
13. Repeat the process in steps 10, 11 and 12 as many times as necessary.
14. Rerun the building analysis using the **Analyze menu > Run Analysis** command. Note that the section properties used for the analysis are the last specified design section properties.
15. Click the **Design menu > Concrete Frame Design > Start Design/Check of Structure** command to rerun the concrete frame design with the new section properties. Review the results using the procedures described above.
16. Click the **Design menu > Concrete Frame Design > Verify Analysis vs Design Section** command to verify that all of the final design sections are the same as the last used analysis sections.
17. Use the **File menu > Print Tables > Concrete Frame Design** command to print selected concrete frame design results if desired.

It is important to note that design is an iterative process. The sections that you use to run your original analysis are not typically the same sections that you end up with at the end of the design process. You always want to be sure to run a building analysis using your final frame section sizes and then run a design check based on the forces obtained from that analysis. The **Design menu > Concrete Frame Design > Verify Analysis vs Design Section** command is useful for making sure that the design sections are the same as the analysis sections.

The following section describes the menu items available on the **Design menu > Concrete Frame Design** submenu.

Concrete Frame Design Menu Commands

This section describes each of the concrete frame design menu commands that are available in ETABS. You can find these commands by clicking **Design menu > Concrete Frame Design**.

Select Design Combo

Click the **Design menu > Concrete Frame Design > Select Design Combo** command to open the Design Load Combinations Selection dialog box. Here you can review the default concrete frame design load combinations defined by ETABS and/or you can designate your own design load combinations.

In the dialog box 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 dialog box for use with the **Add** and **Remove** buttons, if desired.

The default concrete frame design load combinations have names like DCON1, etc.

View/Revise Overwrites

46

 **Tip:**

The concrete frame design overwrites only apply to the frame elements that they are specifically assigned to.

Use the **Design menu > Concrete Frame Design > View/Revise Overwrites** command to review and/or change the concrete frame overwrites. You may not need to assign any concrete frame overwrites; however the option is always available to you.

The concrete frame design overwrites are basic properties that apply only to the frame elements that they are specifically assigned to. Some of the default overwrite values are based on concrete frame preferences. Thus you should define the prefer-

ences before defining the overwrites (and, of course, before designing or checking any concrete frame members).

You can 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 then no changes are made to the item. This is true whether you have one frame element selected or multiple frame elements selected.

Start Design/Check of Structure

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 then only the selected frame elements are designed. If no frame elements are selected when you click this command then all concrete frame elements with the Concrete Frame design procedure are designed.

Interactive Concrete Frame Design

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 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 you can 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.

See the subsection titled "ETABS Interactive Concrete Frame Design" later in this chapter for more information.

Display Design Info

You can review some of the results of the concrete frame design directly on the ETABS 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.

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 whose design sections are to be changed prior to 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.

46

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 re-running the analysis.

Reset Design Section to Last Analysis

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 you should select the elements whose design sections are to be reset prior to implementing this command. If you do not select any elements this command will not be available.

Verify Analysis vs Design Section

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 then 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 ETABS 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. 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 Concrete Frame design procedure.

Reset All Concrete Overwrites

The **Design menu > Concrete Frame Design > Reset All Concrete Overwrites** command resets the overwrites for all frame sections with the Concrete Frame design procedure back 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 ETABS database (*.edb) file.

Delete Concrete Design Results

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 ETABS database (*.edb) file. Note that deleting your concrete design results does not delete your current design section (next analysis section).

ETABS Interactive Concrete Frame Design

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 in the Concrete Design Information dialog box. If you are not currently displaying design results you can 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.

The following bullet items describe the features that are included in the Concrete Design Information dialog box.

**Tip:**

The section property displayed for the Current Design/Next Analysis item is used by ETABS as the section property for the next analysis run.

- **Story ID:** This is the story level ID associated with the frame element.
- **Beam:** This is the label associated with a frame element that has been assigned a concrete frame section property that is designated as a *beam*. See the important note on the first page of this chapter for more information.
- **Column:** This is the label associated with a frame element that has been assigned a concrete frame section property that is designated as a *column*. See the important note on the first page of this chapter for more information.
- **Analysis section:** This is the section property that was used for this frame element in the last analysis. Thus the design forces are based on a frame element of this section property. For your final design iteration the Current Design/Next Analysis section property and the Last Analysis section property should be the same.
- **Design section:** This is the current design section property. The current element design is done for the section property specified in this edit box.

It is important to note that subsequent analyses use the section property specified in this list box for the next analysis section for the frame element. Thus the forces and moments obtained in the next analysis are based on this section.

You can change the Current Design/Next Analysis section property in the overwrites. Click the **Overwrites** button to do this.

- **Table showing reinforcement information:** This table shows the output information obtained for each design load combination at each output station along the frame element. For columns that are designed by ETABS the item with the largest required amount of longitudinal reinforcing is initially highlighted. For columns that are checked by ETABS the item with the largest capacity ratio is initially highlighted. For beams the item with the

largest required amount of bottom steel is initially highlighted.

Following are the possible headings in the table:

- ✓ **Combo ID:** This is the name of the design load combination considered.
- ✓ **Station location:** This is the location of the station considered measured from the i-end of the frame element.
- ✓ **Longitudinal reinforcement:** This item applies to columns only. It also only applies to columns for which ETABS designs the longitudinal reinforcing. It is the total required area of longitudinal reinforcing steel.
- ✓ **Capacity ratio:** This item applies to columns only. It also only applies to columns for which you have specified the location *and* size of reinforcing bars and thus ETABS checks the design. This item is the capacity ratio.

The capacity ratio is determined by first extending a line from the origin of the PMM interaction surface to the point representing the P, M₂ and M₃ values for the designated load combination. Call the length of this first line L₁. Next a second line is extended from the origin of the PMM interaction surface *through* the point representing the P, M₂ and M₃ values for the designated load combination until it intersects the interaction surface. Call the length of this line from the origin to the interaction surface L₂. The capacity ratio is equal to L₁/L₂.

- ✓ **Major shear reinforcement:** This item applies to columns only. It is the total required area of shear reinforcing per unit length for shear acting in the column major direction. See the subsection titled "Local Axes Assignments to Line Objects" in Chapter 14 for a definition of the major direction.

- ✓ **Minor shear reinforcement:** This item applies to columns only. It is the total required area of shear reinforcing per unit length for shear acting in the column minor direction. See the subsection titled "Local Axes Assignments to Line Objects" in Chapter 14 for a definition of the minor direction.
- ✓ **Top steel:** This item applies to beams only. It is the total required area of longitudinal top steel at the specified station.
- ✓ **Bottom steel:** This item applies to beams only. It is the total required area of longitudinal bottom steel at the specified station.
- ✓ **Shear steel:** This item applies to beams only. It is the total required area of shear reinforcing per unit length at the specified station for loads acting in the local 2-axis direction of the beam.

Note:

*You can change
the concrete
frame design
overwrites and
instantaneously
see the new
design results.*

- **Details button:** Clicking this button displays design details for the frame element. You can print this information by selecting Print from the File menu that appears at the top of the window displaying the design details.
- **Overwrites button:** You can click this button to access and make revisions to the concrete frame overwrites and then immediately see the new design results. If you modify some overwrites in this mode and you exit both the Concrete Frame Design Overwrites form and the Concrete Design Information dialog box by clicking their respective **OK** buttons then the changes you made to the overwrites are permanently saved.

When you exit the Concrete Frame Design Overwrites form by clicking the **OK** button the changes are temporarily saved. If you then exit the Concrete Design Information dialog box by clicking the **Cancel** button the changes you made to the concrete frame overwrites are considered temporary only and are not permanently saved. Permanent saving of the overwrites does not actually occur until you click the **OK** button in the Con-

crete Design Information dialog box, not the Concrete Frame Design Overwrites dialog box.

- **Interaction button:** Clicking this button displays the bi-axial interaction curve for the concrete section at the location in the element that is highlighted in the table.



Chapter 47

Composite Beam Design

Any line object that ETABS has assigned a Composite Beam design procedure can be designed in the Composite Beam Design postprocessor. See the section titled "ETABS Default Design Procedure Assignments" in Chapter 17 for more information.

This section describes the intended composite beam design procedure and the menu commands available for composite beam design. Before describing the design procedure or menu items it is important that you understand the distinction between analysis sections and design sections. This is discussed first.

47

Analysis Sections and Design Sections

Analysis sections are those section properties used to analyze the model when you click the **Analyze menu > Run Analysis** command. The design section is whatever section has most currently been designed and thus designated the current design section.

It is possible for the last used analysis section and the current design section to be different. For example you may have run your

analysis using a W18X35 beam and then found in the design that a W16X31 beam worked. In this case the last used analysis section is the W18X35 and the current design section is the W16X31. Before you complete the design process you want to make sure that the last used analysis section and the current design section is the same. The **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command, which is useful for this task, is discussed more in later subsections.

ETABS keeps track of the analysis section and the design section separately. Note the following about analysis and design sections:

- Anytime you assign a beam a frame section property using the **Assign menu > Frame/Line > Frame Section** command ETABS assigns this section as both the analysis section and the design section.
- Whenever you run an analysis using the **Analyze menu > Run Analysis** command (or its associated toolbar button) ETABS always sets the analysis section to be the same as the current design section.
- When you use the **Assign menu > Frame/Line > Frame Section** command to assign an auto select list to a frame section ETABS initially sets the design section to be the beam with the median weight in the auto select list.
- Anytime you unlock your model ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Design menu > Composite Beam Design > Select Design Combo** command to change a design load combination ETABS deletes your design results but it does not delete or change the design section.
- Anytime you use the **Define menu > Load Combinations** command to change a design load combination ETABS deletes your design results but it does not delete or change the design section.



Note:

Any time you unlock your model your design results (and analysis results) are deleted.

- Anytime you use the **Options menu > Preferences > Composite Beam Design** command to change any of the composite beam design preferences ETABS deletes your design results but it does not delete or change the design section.
- Anytime you do something that causes your static nonlinear analysis results to be deleted then the design results for any load combination that includes static nonlinear forces are also deleted. Typically your static nonlinear analysis and design results are deleted when you do one of the following:
 - ✓ Use the **Define menu > Frame Nonlinear Hinge Properties** command to redefine existing or define new hinges.
 - ✓ Use the **Define menu > Static Nonlinear/Pushover Cases** command to redefine existing or define new static nonlinear load cases.
 - ✓ Use the **Assign menu > Frame/Line > Frame Nonlinear Hinges** to add or delete hinges.

Again note that this only deletes results for load combinations that include static nonlinear forces.

Composite Beam Design Procedure

Following is a typical composite beam design process that might occur for a new building. Note that the sequence of steps you may take in any particular design may vary from this but the basic process will probably be essentially the same.

1. Use the **Options menu > Preferences > Composite Beam Design** command to choose the composite beam design code and to review other composite beam design preferences and revise them if necessary. Note that there are default values provided for all composite beam design preferences so it is not actually necessary for you to define any preferences unless you want to change some of the default preference values.

2. Create the building model. See the section titled "Modeling Process" in Chapter 6 for more information.
3. Run the building analysis using the **Analyze menu > Run Analysis** command.
4. Assign composite beam overwrites, if needed, using the **Design menu > Composite Beam Design > View/Revise Overwrites** command. Note that you must select beams first before using this command. Also note that there are default values provided for all composite beam design overwrites so it is not actually necessary for you to define any overwrites unless you want to change some of the default overwrite values.
5. Designate design groups, if desired, using the **Design menu > Composite Beam Design > Select Design Group** command. Note that you must have already created some groups by selecting objects and clicking the **Assign menu > Group Names** command.
6. If you want to use any design load combinations other than the default ones created by ETABS for your composite beam design then click the **Design menu > Composite Beam Design > Select Design Combo** command. Note that you must have already created your own design combos by clicking the **Define menu > Load Combinations** command.

Note that for composite beam design you specify separate design load combinations for construction loading, final loading considering strength, and final loading considering deflection. Design load combinations for each of these three conditions are specified using the **Design menu > Composite Beam Design > Select Design Combo** command.

7. Click the **Design menu > Composite Beam Design > Start Design/Check of Structure** command to run the composite beam design.

8. Review the composite beam design results. To do this you might do one of the following:

- a. Click the **Design menu > Composite Beam Design > Display Design Info** command to display design input and output information on the model.
- b. Right click on a beam while the design results are displayed on it to enter the interactive design mode and interactively design the beam. Note that while you are in this mode you can also view diagrams (load, moment, shear and deflection) and view design details on the screen.

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.

- c. Use the **File menu > Print Tables > Composite Beam Design** command to print composite beam design data. If you select a few beams before using this command then data is printed only for the selected beams.
9. Use the **Design menu > Composite Beam Design > Change Design Section** command to change the beam design section properties for selected beams.
10. Click the **Design menu > Composite Beam Design > Start Design/Check of Structure** command to rerun the composite beam design with the new section properties. Review the results using the procedures described above.
11. Rerun the building analysis using the **Analyze menu > Run Analysis** command. Note that the beam section properties used for the analysis are the last specified design section properties.

12. Click the **Design menu > Composite Beam Design > Start Design/Check of Structure** command to rerun the composite beam design with the new analysis results and new section properties. Review the results using the procedures described above.
13. Again use the **Design menu > Composite Beam Design > Change Design Section** command to change the beam design section properties for selected beams, if necessary.
14. Repeat the process in steps 11, 12 and 13 as many times as necessary.
15. Select all beams and click the **Design menu > Composite Beam Design > Make Auto Select Section Null** command. This removes any auto select section assignments from the selected beams.
16. Rerun the building analysis using the **Analyze menu > Run Analysis** command. Note that the beam section properties used for the analysis are the last specified design section properties.
17. Click the **Design menu > Composite Beam Design > Start Design/Check of Structure** command to rerun the composite beam design with the new section properties. Review the results using the procedures described above.
18. Click the **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command to verify that all of the final design sections are the same as the last used analysis sections.
19. Use the **File menu > Print Tables > Composite Beam Design** command to print selected composite beam design results if desired.

It is important to note that design is an iterative process. The sections that you use to run your original analysis are not typically the same sections that you end up with at the end of the design process. You always want to be sure to run a building analysis using your final beam section sizes and then run a design check based on the forces obtained from that analysis. The **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command is useful for making sure that the design sections are the same as the analysis sections.

The following section describes the menu items available on the **Design menu > Composite Beam Design** submenu.

Composite Beam Design Menu Commands

This section describes each of the composite beam design menu commands that are available in ETABS. You can find these commands by clicking **Design menu > Composite Beam Design**.

Select Design Group

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 ETABS 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 is 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.

**Tip:**

Beams designed as a part of a group must be assigned auto select section lists.

- 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.

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.

Select Design Combo

Click the **Design menu > Composite Beam Design > Select Design Combo** command to open the Design Load Combinations Selection dialog box. Here you can review the default composite beam design load combinations defined by ETABS and/or you can designate your own design load combinations. Note that for composite beam design separate design load combinations are specified for construction loading, final loading considering strength, and final loading considering deflection. Each of these types of design load combinations is specified in a separate tab in the dialog box.

In the dialog box 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 dialog box for use with the **Add** and **Remove** buttons, if desired.

The default composite beam design load combinations have names like DCMPC1, etc. These are described below:

- **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 ETABS 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 ETABS 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 ETABS default for deflection considerations under final loading in composite design.

View/Revise Overwrites

Use the **Design menu > Composite Beam Design > View/Revise Overwrites** command to review and/or change the composite beam overwrites. You may not need to assign any composite beam overwrites; however the option is always available to you. If you are using cover plates or user-defined shear connector patterns then you must assign them through the overwrites. This is the only place available to assign these items.



Tip:

The composite beam design overwrites only apply to the beams that they are specifically assigned to.

The composite beam design overwrites are basic properties that apply only to the beams that they are specifically assigned to. Some of the default overwrite values are based on composite beam preferences. Thus you should define the preferences before defining the overwrites (and, of course, before designing or checking any composite beam).

47

You can 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 either 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 then no changes are made to the item. This is true whether you have one beam selected or multiple beams selected.

Start Design/Check of Structure

To run a composite beam design simply click **Design menu > Composite Beam 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 composite beams in the model.

If you have selected composite beams when you click this command then only the selected beams are designed. If no beams are selected when you click this command then all composite beams are designed.

Interactive Composite Beam Design

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 again.

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.

See the subsection titled "Interactive Composite Beam Design and Review" later in this chapter for more information.

Display Design Info

You can review some of the results of the composite beam design directly on the ETABS 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.

Make Auto Select Section Null

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 done with actual beam sections assigned, not auto select sections.

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 you should select the elements whose auto select sections are to be made null prior to executing this command. If you do not select any elements this command will not be available. Often you may want to select all elements prior to 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 then use the **Assign menu > Frame/Line > Frame Section** command to change the section property.

Change Design Section

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 use the **Design menu > Composite**

Beam 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 whose design sections are to be changed prior to 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 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 beam you can 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.

Reset Design Section to Last Analysis

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 you should select the elements whose design sections are to be reset prior to implementing this command. If you do not select any elements this command will not be available.

Verify Analysis vs Design Section

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 then 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 ETABS tells you how many beams have different analysis and design sections and then selects those beams, if you ask it to. 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 composite beam sections.

Reset All Composite Beam Overwrites

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 ETABS database (*.edb) file.

Delete Composite Beam Design Results

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 ETABS database (*.edb) file. Note that deleting your composite beam design results does not delete your current design section (next analysis section).

Interactive Composite Beam Design and Review

Right click on a beam while the design results are displayed on it to enter the interactive design mode and interactively design the beam in the Interactive Composite Beam Design and Review dialog box. 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.

The following subsections describe the features that are included in the Interactive Composite Beam Design and Review dialog box.

Member Identification Area of Dialog Box



Tip:

If a beam is redesigned as a result of a change made in the Interactive Composite Beam Design and Review dialog box then the design group is ignored and only the single beam is considered.

47

Story ID

This is the story level ID associated with the composite beam.

Beam Label

This is the label associated with the composite beam.

Design Group

This list box displays the name of the design group that the beam is assigned to *if that design group was considered in the design of the beam*. If the beam is part of a design group but the design group was not considered in the design then N/A is displayed. If the beam is not assigned to any design group then "NONE" is displayed.

If a beam is redesigned as a result of a change made in the Interactive Composite Beam Design and Review dialog box then the design group is ignored and only the single beam is considered. Thus as soon as you design a beam once in the Interactive Composite Beam Design and Review dialog box the Design Group box either displays N/A or None.

You can not directly edit the contents of this list box.

Section Information Area of Dialog Box

Auto Select List

This drop-down box displays the name of the auto select list assigned to the beam. If no auto select list is assigned to the beam then NONE is displayed. You can change this item to another auto select list or to NONE while in the dialog box and the design results are immediately updated. If you change this item to NONE then the design is done for the Current Design/Next Analysis section property.

Optimal

If an auto select list is assigned to the beam then this list box displays the optimal section as determined by either beam weight or price depending on what is specified in the composite beam preferences. If no auto select list is assigned to the beam then N/A is displayed for this item.

You can not directly edit the contents of this list box.

47

Last Analysis

This list box displays the name of the section that was used for this beam in the last analysis. Thus the beam forces are based on a beam of this section property. For your final design iteration the Current Design/Next Analysis section property and the Last Analysis section property should be the same.

You can not directly edit the contents of this list box.



Tip:

The section property displayed for the Current Design/Next Analysis item is used by ETABS as the section property for the next analysis run.

Current Design/Next Analysis

This list box displays the name of the current design section property. If the beam is assigned an auto select list then the section displayed in this dialog box initially defaults to the optimal section.

If there is no auto select list assigned to the beam then the beam design is done for the section property specified in this edit box.

It is important to note that subsequent analyses use the section property specified in this list box for the next analysis section for the beam. Thus the forces and moments obtained in the next analysis are based on this beam size.

There are two ways that you can change the Current Design/Next Analysis section property. The first is to double click on any section displayed in the Acceptable Sections List. This updates the Current Design/Next Analysis section property to the section you double clicked in. The second way to change the property is to click the **Sections** button that is documented later in this chapter.

Important note: Changes made to the Current Design/Next Analysis section property are permanently saved (until you revise them again) if you click the **OK** button to exit the Interactive Composite Beam Design and Review dialog box. If you exit the dialog box by clicking the **Cancel** button then these changes are considered temporary and are not permanently saved.

Acceptable Sections List Area of Dialog Box

The Acceptable Sections List includes the following information for each beam section that is acceptable for all considered design load combinations.

- Section name
- Steel yield stress, Fy
- Connector layout

- Camber
- Ratio

**Tip:**

When you see a single beam displayed in the Acceptable Sections List in a red font this means that none of the sections considered were acceptable.

Typically the ratio displayed is the largest ratio obtained considering the stress ratios for positive moment, negative moment and shear for both construction loads and final loads, and also considering the ratio obtained by dividing the actual maximum deflection by the allowable deflection. The one exception to this occurs when none of the beam sections considered is acceptable. This is discussed below.

If the beam is assigned an auto select list then there may be many beam sections in the Acceptable Sections List. If necessary you can use the scroll bar to scroll through the acceptable sections. The optimal section is initially highlighted in the list.

If the beam is not assigned an auto select list then there is only one beam section in the Acceptable Sections List. It is the same section as specified in the Current Design/Next Analysis edit box.

There will always be at least one beam shown in the Acceptable Sections List, even if none of the beams considered are acceptable. In the case where no beams are acceptable ETABS displays the section with the lowest ratio in a red font. Thus when you see a single beam displayed in the Acceptable Sections List in a red font this means that none of the sections considered were acceptable.

Redefine Area of Dialog Box

47

Sections Button

You can use the **Sections** button to change the Current Design /Next Analysis section property. Using this button you can designate a new section property whether or not that section property is displayed in the Acceptable Sections List.

When you click on the **Sections** button the Select Sections Properties dialog box appears which lets you assign any frame section property to the beam. Note that if an auto select list is assigned to

the beam then using the **Sections** button sets the auto select list assignment to NONE.

Overwrites Button

You can click this button to access and make revisions to the composite beam overwrites and then immediately see the new design results. If you modify some overwrites in this mode and you exit both the Composite Beam Overwrites form and the Interactive Composite Beam Design and Review dialog box by clicking their respective **OK** buttons then the changes you made to the overwrites are permanently saved.

When you exit the Composite Beam Overwrites form by clicking the **OK** button the changes are temporarily saved. If you then exit the Interactive Composite Beam Design and Review dialog box by clicking the **Cancel** button the changes you made to the composite beam overwrites are considered temporary only and are not permanently saved. Permanent saving of the overwrites does not actually occur until you click the **OK** button in the Interactive Composite Beam Design and Review dialog box, not the overwrites dialog box.

Temporary Area of Dialog Box

Combos Button

You can click this button to access and make *temporary* revisions to the design load combinations considered for the beam. This may be useful for example if you want to see the results for one particular load combination. You can temporarily change the considered design load combinations to be just the one you are interested in and review the results.

The changes made here to the considered design load combinations are temporary. They are not saved when you exit the Interactive Composite Beam Design and Review dialog box regardless of whether you click **OK** or **Cancel** to exit it.

Show Details Area of Dialog Box

Diagrams Button

Clicking the **Diagrams** button brings up a dialog box that allows you to display the following four types of diagrams for the beam.

- Applied loads
- Shear diagram
- Moment diagram
- Deflection diagram

The diagrams are plotted for specific design load combinations specified in the dialog box by you.

Details Button

Clicking this button displays design details for the beam. This same information can be printed using the **File menu > Print Tables > Composite Beam Design** command.

Shear Wall Design

Overview

This chapter discusses some of the basics of shear wall design in ETABS. Topics covered include labeling of pier and spandrel sections, the intended design procedure for shear walls, ETABS menu commands for shear wall design and using the Section Designer utility to define shear wall reinforcing.

Wall Pier Labeling

48

General

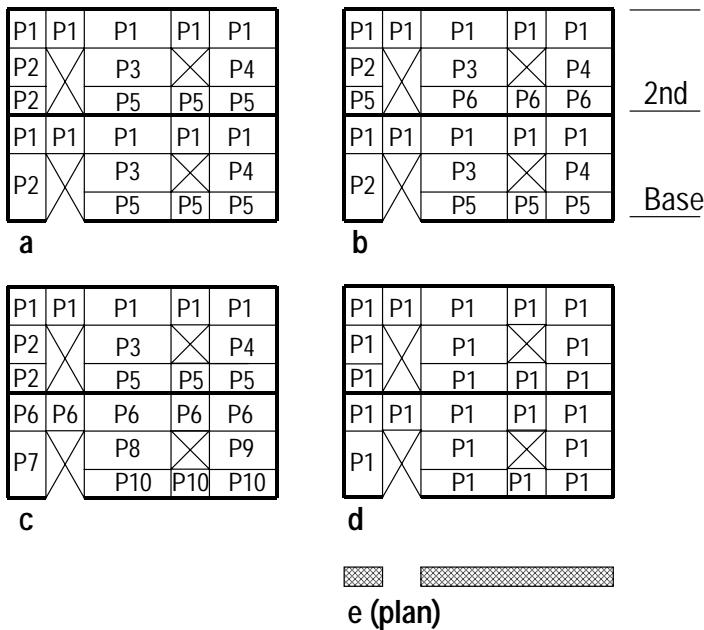
Note:

Piers are only designed at stations located at the top and the bottom of the pier.

Wall pier forces are output at the top and bottom of wall pier elements. Also wall pier design is only performed at stations located at the top and bottom of wall pier elements.

Each area object that makes up a part of a wall may be assigned one pier label (and one spandrel label). You can not assign a single area object multiple wall pier labels. Area objects at the same

Figure 48-1:
Examples of wall pier labeling



Note:

Area objects at the same story level with the same pier label are assumed by ETABS to be part of the same pier.

story level with the same pier label are assumed by ETABS to be part of the same pier.

Wall pier labels are used to identify wall piers. Once a wall pier is assigned a label, and an analysis has been run, forces can be output for the wall pier and it can be designed.

A single wall pier can not extend over multiple stories. It must be fully contained within one story level.

Assigning Wall Pier Labels

Figure 48-1 illustrates some possible wall pier labeling arrangements for a two-story wall. Note that the layout of the wall is similar at the two levels except that at the upper level the pier to the left of the door opening is broken into two area objects.

Figure 48-1a shows what may be the most common way we would expect the piers to be labeled. At the upper level, Pier P1 is defined to extend all the way across the wall above the openings. Pier P2 makes up the wall pier to the left of the door open-

ing. Pier P3 occurs between the door and window openings. Pier P4 occurs between the window opening and the edge of the wall. Pier P5 occurs below the window opening between the door and the edge of the wall. A similar labeling of piers occurs at the lower level. Note the following about the wall pier labeling scheme shown in Figure 48-1a:

Note:

Wall piers are always associated with the story level directly above them.

- Wall piers are always associated with the story level directly above them. Thus in Figure 48-1a the upper level wall piers are associated with the Roof level and the lower level wall piers are associated with the 2nd level. Because the wall piers are associated with story levels you are able to repeat wall pier labels at different levels as shown in the figure.
- When we refer to wall pier P1 at the Roof level in Figure 48-1a we are referring to the pier across the entire width of the wall that is made up of the five area objects given the pier label P1. Similarly pier P2 at the Roof level is made up of the two area objects to the left of the door opening.
- Wall pier design is performed at the top and bottom of each pier. Thus for wall pier P2 at the Roof level, design is performed at the top and bottom of the door opening. No design is performed near the midheight of the door opening because the design is done at the top and bottom of the wall pier, not the top and bottom of each area object that makes up the wall pier.
- Wall pier forces are reported at the top and bottom of each pier. Thus for wall pier P2 at the Roof level wall pier forces are reported (printed) for locations at the top and bottom of the door opening. For graphic representation on the model the forces are plotted at the top and bottom of the pier and connected with a straight line.
- If, for example, you are not interested in either design or output forces for wall piers P1 and P5 at the Roof level, then you should simply not provide wall pier labels for those area objects.

Tip:

If you need to mesh an existing area object to define a wall pier you can select the area object(s) and use the Edit menu > Mesh Areas command.

Figure 48-2:
Example of possibly incomplete wall pier labeling

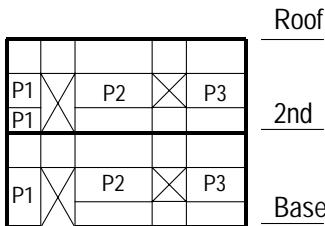


Figure 48-1b shows what you might do if you want a design section at near the midheight of the Roof level pier on the left side of the door opening. Notice that the two area objects are given different pier labels, P2 and P5.

Figure 48-1c illustrates that you do not have to repeat your pier numbers at each level. You can give each wall pier a unique label if you want to. Even with unique names the piers are still associated with story levels. For example, in Figure 48-1c, pier P7 is associated with the 2nd level.

Figure 48-1d illustrates that you could give all of the area objects the same label if you wanted to, P1 in this case. For this condition wall design would be performed across the entire wall at each story level (i.e., the top and bottom of each pier), and wall forces would be provided for the entire wall at each story level.

In Figure 48-1d, the design of the bottom of the lower level pier is based on the section shown in Figure 48-1e. ETABS would assume that the two areas that comprise these sections are rigidly connected.

You could only specify pier labels for some of the area objects in the wall. Figure 48-2 shows an example of this. In this case, for design, you would not capture the overall effects at the top and bottom of each story level like you would if you defined the piers as shown in Figure 48-1. Thus in general, if you are going to design the wall, we recommend that you define the piers as shown in Figure 48-1. There is nothing wrong with defining the piers only as shown in Figure 48-2 other than you may not get all of the needed design information.

Wall Spandrel Labeling

General

Note:

Spandrels are only designed at stations located at the left and right ends of the spandrel.

Wall spandrel forces are output at the left and right ends of wall spandrel elements. Also wall spandrel design is only performed at stations located at the left and right ends of wall spandrel elements.

Each area object that makes up a part of a wall may be assigned one spandrel label (and one pier label). You can not assign a single area object multiple wall spandrel labels.

Wall spandrel labels are used to identify wall spandrels. Once a wall spandrel is assigned a label, and an analysis has been run, forces can be output for the wall spandrel and it can be designed.

Assigning Wall Spandrel Labels

Figure 48-3 illustrates some possible wall spandrel labeling arrangements for a two-story wall. Note that this is the same two-story wall illustrated in Figure 48-1 for the discussion of wall pier labeling.

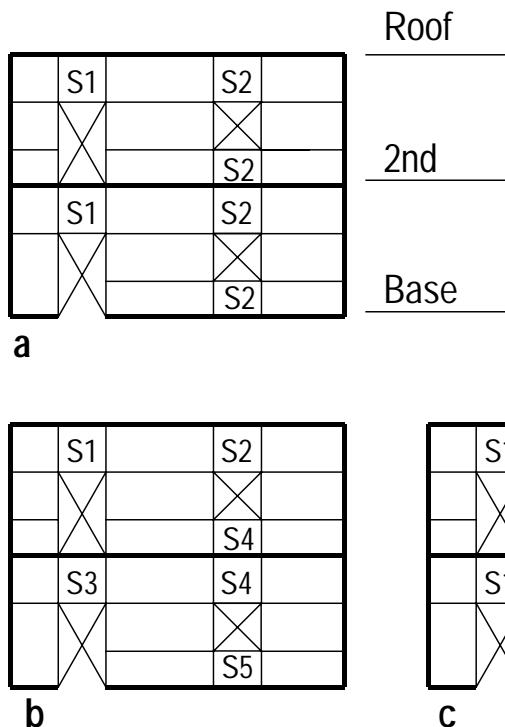
Figure 48-3a shows what may be the most common condition for wall spandrel labeling. Note the following about the wall spandrel labeling scheme shown in Figure 48-3a:

Note:

Unlike wall pier elements, a single wall spandrel element can include area objects from two adjacent story levels.

- Use the following method to determine which story level a pier spandrel is associated with.
 - ✓ Start with the upper-most area object in the spandrel. Check if the top of the object is at a story level. If it is then that is the story associated with the spandrel. If it is not then check if the bottom of the area object is at a story level. If it is then that is the story associated with the spandrel.
 - ✓ If a story level has not been located continue down to the next highest area object and check for story levels at the top or bottom of the object.

Figure 48-3:
Examples of wall
spandrel labeling



- ✓ Continue the above process until a level is located. Thus a spandrel is typically associated with the highest story level that it touches or intersects.
- ✓ If the spandrel does not actually touch or intersect a story level then it is associated with the story level just above it. An example of this is discussed later.

- In Figure 48-3a the upper wall spandrel label S1 is associated with the Roof level and the lower S1 is associated with the 2nd level. The upper wall spandrel label S2 is associated with the Roof level, the middle spandrel made up of two area objects labeled S2 is associated with the 2nd level and the lowest S2 spandrel is associated with the Base level.
- Because the wall spandrels are associated with story levels you are able to repeat wall spandrel labels at different levels as shown in the figure.



Tip:

If you need to mesh an existing area object to define a wall spandrel you can select the area object(s) and use the **Edit menu > Mesh Areas** command.

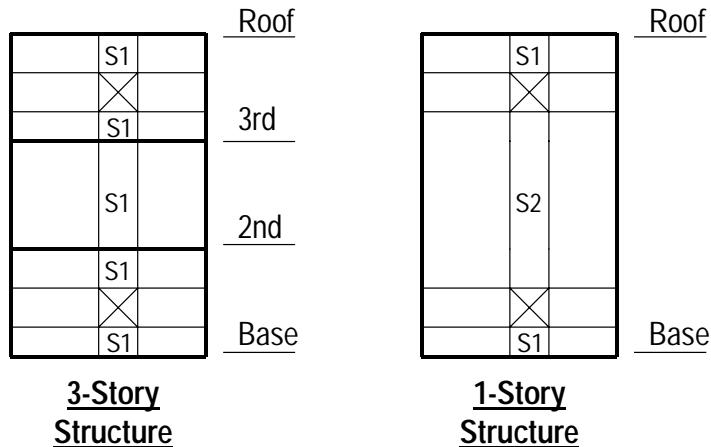
- When we refer to wall spandrel S2 at the 2nd level in Figure 48-3a we are referring to the spandrel that is made up of the two area objects given the spandrel label S2.
- Wall spandrel design is performed at the left and right sides of each spandrel. Thus for wall spandrel S1 at the Roof level design is performed at the left and right sides of the door opening.
- Wall spandrel forces are reported at the left and right sides of each spandrel. Thus for wall spandrel S1 at the Roof level wall spandrel forces are reported (printed) for locations at the left and right sides of the door opening. For graphic representation on the model the forces are plotted at the left and right sides of the spandrel and connected with a straight line.
- If you are not interested in either design or output forces for certain wall spandrels, then you should simply not provide wall spandrel labels for those area objects.

Figure 48-3b illustrates that you do not have to repeat spandrel numbers at each level. You can give each wall spandrel a unique label if you want to. Even with unique names the spandrels are still associated with story levels. For example, in Figure 48-3b, spandrel S4 is associated with the 2nd level.

Figure 48-3c illustrates a condition that ETABS will not reject, although it is doubtful that you would mean to label the spandrels like this. Specifically refer to the spandrel at the 2nd level between the windows. Notice that the upper area object for this spandrel is labeled S2 and the lower area object is labeled S3. ETABS will accept this and will design the two objects as separate spandrels.

In the 3-story structure shown in Figure 48-4 the top spandrel labeled S1 is associated with the Roof level. The middle S1 spandrel is associated with the 3rd level, which is the highest story level that it intersects or touches. The lowest S1 spandrel is associated with the Base level.

Figure 48-4:
Additional examples
of wall spandrel la-
beling



In the 1-story structure shown in Figure 48-4 the top spandrel labeled S1 is associated with the Roof level. The middle spandrel labeled S2 is also associated with the Roof level because the spandrel does not intersect or touch any story levels and thus it is associated with the story level directly above it. The lowest S1 spandrel is associated with the Base level.

Shear Wall Design Procedure

Note:

*Shear wall de-
sign in ETABS
is an iterative
process.*

Following is a typical shear wall design process that might occur for a new building. Note that the sequence of steps you may take in any particular design may vary from this but the basic process will be essentially the same.

1. Use the **Options menu > Preferences > Shear Wall Design** command to review the shear wall design preferences and revise them if necessary. Note that there are default values provided for all shear wall design preferences so it is not actually necessary for you to define any preferences unless you want to change some of the default preference values.
2. Create the building model. See the section titled "Modeling Process" in Chapter 6 for more information.

3. Run the building analysis using the **Analyze menu > Run Analysis** command.
4. Assign the wall pier and wall spandrel labels. Use the **Assign menu > Frame/Line > Pier Label**, the **Assign menu > Shell/Area > Pier Label**, the **Assign menu > Frame/Line > Spandrel Label**, and the **Assign menu > Shell/Area > Spandrel Label** commands to do this.

Note that the labels can be assigned before or after the analysis is run.

5. Assign shear wall overwrites, if needed, using the **Design menu > Shear Wall Design > View/Revise Pier Overwrites** and the **Design menu > Shear Wall Design > View/Revise Spandrel Overwrites** commands. Note that you must select piers or spandrels first before using these commands. Also note that there are default values provided for all pier and spandrel design overwrites so it is not actually necessary for you to define any overwrites unless you want to change some of the default overwrite values.

Note that the overwrites can be assigned before or after the analysis is run.

Important note about selecting piers and spandrels: You can select a pier or spandrel simply by selecting any line or area object that is part of the pier or spandrel.

6. If you want to use any design load combinations other than the default ones created by ETABS for your shear wall design then click the **Design menu > Shear Wall Design > Select Design Combo** command. Note that you must have already created your own design combos by clicking the **Define menu > Load Combinations** command.
7. Click the **Design menu > Shear Wall Design > Start Design/Check of Structure** command to run the shear wall design.

8. Review the shear wall design results. To do this you might do one of the following:
 - a. Click the **Design menu > Shear Wall Design > Display Design Info** command to display design input and output information on the model.

- b. Right click on a pier or spandrel while the design results are displayed on it to enter the interactive wall design mode. Note that while you are in this mode you can revise overwrites and immediately see the new design results.

If you are not currently displaying design results you can 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 element.

- c. Use the **File menu > Print Tables > Shear Wall Design** command to print shear wall design data. If you select a few piers or spandrels before using this command then data is printed only for the selected elements.
9. If desired, revise the wall pier and/or spandrel overwrites, rerun the shear wall design, and review the results again. Repeat this step as many times as needed.
10. If desired, create wall pier check sections with user-defined (actual) reinforcing specified for the wall piers using the Section Designer utility. Use the **Design menu > Shear Wall Design > Define Pier Sections for Checking** command to define the sections in Section Designer. Be sure to indicate that the reinforcing is to be checked. Use the **Design menu > Shear Wall Design > Assign Pier Sections for Checking** command to assign these sections to the piers. Rerun the design and verify that the actual flexural reinforcing provided is adequate.

11. Assign these check sections to the piers, change the pier mode from Design to Check, and rerun the design. Verify that the actual flexural reinforcing provided is adequate.
12. If necessary, revise the geometry or reinforcing and rerun the design.
13. Print or display selected shear wall design results if desired.

Note that shear wall design is performed as an iterative process. You can change your wall design dimensions and reinforcing during the design process without rerunning the analysis. However, you always want to be sure that your final design is based on analysis properties (wall dimensions) that are consistent with your design (actual) wall dimensions.

The following section describes the menu items available on the **Design menu > Shear Wall Design** submenu.

Menu Commands for Shear Wall Design

Select Design Combo

Click the **Design menu > Shear Wall Design > Select Design Combo** command to open the Design Load Combinations Selection dialog box. Here you can review the default shear wall design load combinations defined by ETABS and/or you can designate your own design load combinations.

In the dialog box 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 dialog box for use with the **Add** and **Remove** buttons, if desired.

The default shear wall design load combinations have names like DWAL1, etc.

View/Revise Pier Overwrites

Use the **Design menu > Shear Wall Design > View/Revise Pier Overwrites** command to review and/or change the wall pier overwrites. You may not need to assign any wall pier overwrites; however the option is always available to you.

The wall pier design overwrites are basic properties that apply only to the piers that they are specifically assigned to. Note that inputting 0 for most pier overwrite items means to use the ETABS default value for that item.

You can 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, then no changes are made to the pier overwrite item. This is true whether you have one pier selected or multiple piers selected.

Following is a description of the pier overwrite items:

- **Pmax Factor:** This is a factor used to reduce the allowable maximum compressive design strength.

The 1997 UBC limits the maximum compressive design strength, $\phi_c P_n$, to the value given by P_{max} in the following equation.

$$P_{max} = 0.80\phi_c P_{oc} = 0.80\phi[0.85f_c (A_g - A_s) + f_y A_s]$$

In this equation 0.80 is the Pmax Factor. In general you will want to leave this factor at 0.80 which is the default value.

- **LL Factor:** A reducible live load is multiplied by this factor to obtain the reduced live load. If the LL reduction factor is program calculated then it is based on the live load reduction method chosen in the live load reduction preferences which are set through Options menu > Preferences > Live Load Reduction. If you specify your own LL reduction factor then ETABS ignores any reduction method specified in the live load reduction preferences

and simply calculates the reduced live load for a pier or spandrel by multiplying the specified LL reduction factor times the reducible live load.

- **EQ Factor:** The horizontal earthquake factor is a multiplier that is applied to all horizontal earthquake loads. ETABS assumes the following types of loads are horizontal earthquake loads:
 - ✓ Any static load case specified as type "Quake" that has a horizontal component of input load.
 - ✓ Any response spectrum case that has a horizontal component of input load
 - ✓ Any time history case that has a horizontal component of input ground acceleration.

The horizontal EQ factor allows you to specify member-specific reliability/redundancy factors that are required by some codes. The ρ factor specified in Section 1630.1.1 of the 1997 UBC is an example of this.

- **Design is Seismic:** Additional design checks are done for seismic elements compared to nonseismic elements. Also in some cases the strength reduction factors are different.
- **Pier is 3D:** This item tells you whether ETABS considers the pier to be two dimensional or three dimensional. You can not overwrite this item; ETABS simply reports it here for your reference.
- **Overturning Design Type:** The 2D Simplified method designs the pier for overturning assuming that all of the overturning force is resisted by a couple between tension steel at one end of the pier and compression steel at the other end. This is a simplified approximate method. Designs done using this method should always be checked using the 2D or 3D Interaction method.

The 2D or 3D Interaction method designs or checks the pier using P-M (or P-M-M) interaction curves. To use this method you must also assign pier sections to the top and bottom of the pier. In other words, the Section Bottom and Section Top items in the pier overwrites must be filled with something other than NONE.

- **Section Bottom:** This is the name of a pier section defined using the Section Designer utility that is assigned to the bottom of the pier.
- **Section Top:** This is the name of a pier section defined using the Section Designer utility that is assigned to the top of the pier.
- **ThickBot:** Wall pier thickness at bottom of pier.
- **LengthBot:** Wall pier length at bottom of pier.
- **DB1LeftBot:** Length of a user-defined edge member on the left side of a wall pier at the bottom of the wall pier.
- **DB1RightBot:** Length of a user-defined edge member on the right side of a wall pier at the bottom of the wall pier.
- **DB2LeftBot:** Width of a user-defined edge member on the left side of a wall pier at the bottom of the wall pier.
- **DB2RightBot:** Width of a user-defined edge member on the right side of a wall pier at the bottom of the wall pier.
- **ThickTop:** Wall pier thickness at top of pier.
- **LengthTop:** Wall pier length at top of pier.
- **DB1LeftTop:** Length of a user-defined edge member on the left side of a wall pier at the top of the wall pier.
- **DB1RightTop:** Length of a user-defined edge member on the right side of a wall pier at the top of the wall pier.

- **DB2LeftTop:** Width of a user-defined edge member on the left side of a wall pier at the top of the wall pier.
- **DB2RightTop:** Width of a user-defined edge member on the right side of a wall pier at the top of the wall pier.
- **Material:** Concrete material property associated with the pier.
- **PercentMaxT:** Maximum ratio of tension reinforcing allowed in edge members.
- **PercentMaxC:** Maximum ratio of compression reinforcing allowed in edge members.

View/Revise Spandrel Overwrites

Use the **Design menu > Shear Wall Design > View/Revise Spandrel Overwrites** command to review and/or change the wall spandrel overwrites. You may not need to assign any wall spandrel overwrites; however the option is always available to you.

The wall spandrel design overwrites are basic properties that apply only to the spandrels that they are specifically assigned to. Note that inputting 0 for most spandrel overwrite items means to use the ETABS default value for that item.

You can 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 then no changes are made to the item. This is true whether you have one spandrel selected or multiple spandrels selected.

48

Following is a description of the spandrel overwrite items:

- **ThickLeft:** Wall spandrel thickness at left side of spandrel.

- **DepthLeft:** Wall spandrel depth at left side of spandrel.
- **CoverBotLeft:** Distance from bottom of spandrel to centroid of bottom reinforcing on left side of beam.
- **CoverTopLeft:** Distance from top of spandrel to centroid of top reinforcing on left side of beam.
- **SlabWidthLeft:** Slab width for T-beam at left end of spandrel.
- **SlabDepthLeft:** Slab depth for T-beam at left end of spandrel.
- **ThickRight:** Wall spandrel thickness at right side of spandrel.
- **DepthRight:** Wall spandrel depth at right side of spandrel.
- **CoverBotRight:** Distance from bottom of spandrel to centroid of bottom reinforcing on right side of beam.
- **CoverTopRight:** Distance from top of spandrel to centroid of top reinforcing on right side of beam.
- **SlabWidthRight:** Slab width for T-beam at right end of spandrel.
- **SlabDepthRight:** Slab depth for T-beam at right end of spandrel.
- **Material:** Concrete material property associated with the spandrel.
- **Consider Vc:** Toggle switch for whether to consider V_c in computing the shear capacity of the spandrel.

Define Pier Sections for Checking

To define a pier section with reinforcing for checking click the **Design menu > Shear Wall Design > Define Pier Sections for Checking** command. This command allows you to define a pier section using the Section Designer utility. See the subsection titled "Initial Definition of a Wall Pier Section" later in this chapter for more information.

Assign Pier Sections for Checking

Use the **Design menu > Shear Wall Design > Assign Pier Sections for Checking** command to assign a pier a section that has been defined using the Section Designer utility.

Start Design/Check of Structure

To run a shear wall design simply 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 then only the selected piers and/or spandrels are designed. If no piers and/or spandrels are selected when you click this command then all piers and spandrels are designed.

Interactive Wall Design

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 dialog box. If you are not currently displaying design results you can 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.

48

The following two subsections describe the features that are included in the Wall Design dialog box.

Combos Button

You can click this button to access and make *temporary* revisions to the design load combinations considered for the pier or spandrel. This may be useful for example if you want to see the results for one particular load combination. You can temporarily change the considered design load combinations to be just the one you are interested in and review the results.

The changes made here to the considered design load combinations are temporary. They are not saved when you exit the Wall Design dialog box regardless of whether you click **OK** or **Cancel** to exit it.

Overwrites Button

You can click this button to access and make revisions to the pier or spandrel overwrites and then immediately see the new design results. If you modify some overwrites in this mode and you exit both the Overwrites form and the Wall Design dialog box by clicking their respective **OK** buttons then the changes you made to the overwrites are permanently saved.

When you exit the Overwrites form by clicking the **OK** button the changes are temporarily saved. If you then exit the Wall Design dialog box by clicking the **Cancel** button the changes you made to the pier or spandrel overwrites are considered temporary only and are not permanently saved. Permanent saving of the overwrites does not actually occur until you click the **OK** button in the Wall Design dialog box, not the overwrites dialog box.

Display Design Info

You can review some of the results of the shear wall design directly on the ETABS 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.

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 back 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.

Delete Wall Design Results

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 ETABS database (*.edb) file.

At times ETABS automatically deletes your shear wall design results. Following are some of the reasons that this might happen:

- Anytime you unlock your model ETABS deletes your shear wall design results
- Anytime you use the **Design menu > Shear Wall Design > Select Design Combo** command to change a design load combination ETABS deletes your shear wall design results.
- Anytime you use the **Define menu > Load Combinations** command to change a design load combination ETABS deletes your shear wall design results.
- Anytime you use the **Options menu > Preferences > Shear Wall Design** command to change any of the shear wall design preferences ETABS deletes your shear wall design results.

- Anytime you do something that causes your static nonlinear analysis results to be deleted then the design results for any load combination that includes static nonlinear forces are also deleted. Typically your static nonlinear analysis and design results are deleted when you do one of the following:
 - ✓ Use the **Define menu > Frame Nonlinear Hinge Properties** command to redefine existing or define new hinges.
 - ✓ Use the **Define menu > Static Nonlinear/Pushover Cases** command to redefine existing or define new static nonlinear load cases.
 - ✓ Use the **Assign menu > Frame/Line > Frame Nonlinear Hinges** to add or delete hinges.

Again note that this only deletes results for load combinations that include static nonlinear forces.

Using Section Designer to Define Pier Reinforcing

This section provides basic instruction on using the Section Designer utility of ETABS to specify user-defined vertical reinforcing for wall piers. There are many options and features available in Section Designer; it is intended to be used for much more than just defining wall piers. This section does not try to document all of the options and features. Instead it concentrates on a few of the basic features which will help you define wall piers and their vertical reinforcing.

48

Local Axes Definition and Orientation

Before you begin to draw a wall section in Section Designer it is crucial that you understand the local axes definition for the pier and that you understand the orientation that ETABS assumes for the pier.

**Tip:**

Be sure you fully understand the wall pier orientation information here if you are specifying user-defined flexural reinforcing for a wall pier that is unsymmetrical in plan.

In Section Designer you see a plan view section of the pier, *always*. The positive local 2-axis is horizontal pointing to the right, *always*. The positive local 3 axis is vertical pointing up, *always* (unless, of course, you turn your computer monitor upside down). The local 1-axis points toward you.

For both two-dimensional and three-dimensional piers the orientation of the pier local axes is automatically determined by ETABS as described in Chapter 38. The orientation described is built into ETABS and you can not modify it.

You should carefully consider the local axes orientation before beginning to draw your pier section in Section Designer. This will help you avoid having a pier section with the wrong orientation.

Initial Definition of a Wall Pier Section

Starting Section Designer

You begin to define a wall pier section with user-defined vertical reinforcing by clicking **Design menu > Shear Wall Design > Define Pier Sections for Checking**. This command brings up the Pier Sections dialog box where you can click the **Add Pier Section** button to start a new section or click the **Modify>Show Pier Section** button to view and/or modify a previously defined pier section.

**Tip:**

It is usually easier and quicker to start from the analysis pier section geometry rather than starting from scratch.

Clicking the **Add Pier Section** button brings up the Pier Section Data dialog box. The following bullet items discuss the various areas in this dialog box:

- **Section Name:** This is the name of the pier section.
- **Base Material:** This is the material property used for the pier section
- **Add Pier:** The Add New Pier Section option allows you to start the pier section from scratch. See the subsection below titled "Creating a Pier Section from Scratch" for more information.

The Start from Existing Wall Pier option allows you to start with the geometry of an existing wall pier. When you select this option you also specify a story and a wall pier label so ETABS knows which existing pier geometry to use. In cases where the top and bottom geometry of the pier is different ETABS uses the geometry at the *bottom* of the pier. See the subsection below titled "Creating a Pier from the Geometry of an Existing Analysis Pier Section" for more information.

- **Check/Design:** Select the Reinforcement to be Checked option if you want to specify your own reinforcement (location and size) and have ETABS check it.

Select the Reinforcement to be Designed if you want ETABS to determine the required amount of reinforcing for you. In this case you still lay out the reinforcing bars in Section Designer. ETABS will use that layout and report the required percentage of steel. In cases where you use the design option you should, at the end of the design process, always specify your actual final reinforcing and have ETABS check it.

- **Define/Edit>Show Section:** Once you have specified the data in the other areas of the dialog box click the **Section Designer** button to enter the section designer utility. Here you can define the pier geometry and the reinforcing.

When you are done with Section Designer close it and return to the Pier Section data dialog box where you can click the **OK** button to complete the definition of the pier.

Creating a Pier Section from Scratch

Begin defining your pier section from scratch in Section Designer by drawing the concrete section. To do this click the **Draw Polygon Section** button, , located on the side toolbar, or select **Draw menu > Draw Polygon**. You can then proceed to left click the mouse on each corner point of the polygon that describes the wall pier section. You can proceed around the polygon in either a clockwise or a counter-clockwise direction. If you

double click on the last point Section Designer recognizes that you have completed the polygon and draws the shape. Alternatively you can press the Enter key on your keyboard after you single click on the last point to finish the polygon.

Once the pier section is drawn you can add rebar as discussed in the subsection below titled "Revising Rebar Size, Cover and Spacing."

Creating a Pier from the Geometry of an Existing Analysis Pier Section

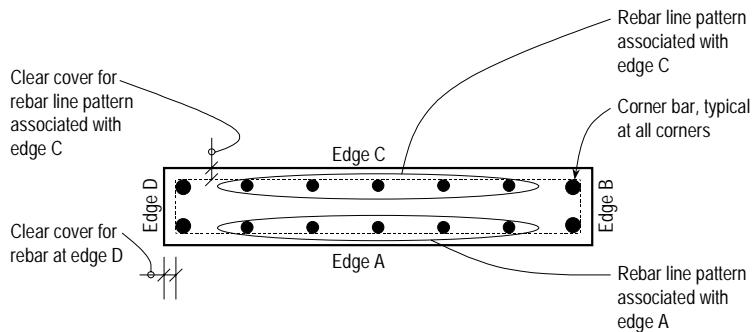
When you choose to create a pier from the geometry of an existing analysis pier section the geometry of the pier is immediately displayed in Section Designer. You can modify the geometry of the section as discussed in the subsection below titled "Modifying the Geometry of the Concrete Section." You can add rebar as discussed in the subsection below titled "Revising Rebar Size, Cover and Spacing."

Modifying the Geometry of the Concrete Section

You can revise the geometry of a polygon by changing the coordinates of the corner points. To do this first click on the **Reshaper** button, , located on the side toolbar. Clicking the **Reshaper** button causes handles to appear on each corner of the polygon (assuming you have created the polygon using one of the methods discussed in the previous section). There are two possible methods to change the geometry of the concrete section:

- Click on one of these handles with the left mouse button, and while still holding the left mouse button down drag the handle to a new location. Release the left mouse button when you have dragged it to the correct location.
- Click on one of these handles with the right mouse button and the Change Coordinates dialog box pops up which allows you to type in new coordinates for the corner point of the polygon.

Figure 48-5:
Rebar example



Revising Rebar Size, Cover and Spacing

General

Note:
The information provided here applies only to the corner rebar and rebar line patterns that are part of the polygon area object in Section Designer. It does not apply to individual rebar elements and rebar line patterns that can also be specified in Section Designer but are not discussed in this section because they are not typically needed for wall piers.

By default, for a polygon section, single rebar elements are provided at each corner of the polygon and rebar line patterns are provided along each face of the polygon. Note the following about these rebar elements:

- Rebar line patterns are defined by a rebar size, maximum center-to-center spacing and clear cover.
- The bars are spaced equally in a rebar line pattern. The equal spacing is measured from the center of the corner bar at one end of the rebar line pattern to the center of the corner bar at the other end of the rebar line pattern.
- Single rebar elements at the corners of a polygon are defined simply by a bar size. The clear cover for these corner bars is determined from the clear cover of the line rebar on either side of the corner bar.

To further illustrate the reinforcing, refer to Figure 48-5. The figure shows a typical wall pier. The four edges of the pier are arbitrarily labeled Edge A, B, C and D for the purposes of this discussion. Note the following about the reinforcing steel illustrated in Figure 48-5:

- There are corner bars located at each of the four corners. Consider the corner bar in the upper left hand corner at the intersection of edge C and edge D. This corner bar is

located such that the clear distance from edge D to the bar is equal to the cover distance specified for the rebar line pattern along edge D. This is true even though the rebar size for the rebar line pattern has been set to "None" as would usually be done for the reinforcing along the edges that define the ends of the wall. Similarly, the clear distance from edge C to the corner bar is equal to the cover distance specified for the rebar line pattern along edge C.

- The corner rebar size may be different at each corner of the pier.
- As mentioned in the first bullet item of this list, the rebar size associated with the rebar line pattern along edges B and D is specified as "None." Note that the cover associated with these rebar line patterns is still valid, even though the rebar size is "None." The cover is still used to locate the corner bars.
- The rebar line pattern along an edge of the pier is parallel to the edge of the pier and extends from the center of the corner bar (or its projection perpendicular to the pier edge) at one end of the considered edge to the center of the corner bar (or its projection perpendicular to the pier edge) at the other end of the considered edge. The rebar line pattern is then divided into equal spaces whose length does not exceed the specified spacing for the rebar line pattern.
- The rebar line pattern size, spacing and cover may be different along each edge of the pier.

48

Methodology

To edit rebar line patterns along an edge of the member simply right click on the rebar line pattern. This pops up the Edge Reinforcing dialog box where you can modify the edge rebar size, maximum spacing and clear cover.

**Tip:**

In Section Designer right click on a rebar to bring up a pop-up dialog box where you can edit the rebar size, spacing and cover.

In the Edge Reinforcing dialog box there is also a check box which when checked applies the specified reinforcing to all edges of the polygon. Note that if you have already specified the rebar size along an edge of the polygon to be "None" then the Apply to All Edges command does not apply the specified reinforcing size and spacing to that edge. It will, however, apply the specified cover to that edge.

To edit corner rebar simply right click on the rebar element. The Corner Point Reinforcing dialog box appears. In this dialog box you can specify the size of the corner bar. There is also a check box that allows you to say that this size applies to all corner bars. Note that if you have already specified the corner rebar size to be "None" then the Apply to All Corners command does not apply the specified reinforcing size at that location.

Modifying Material Properties

The material properties used in Section Designer are the same ones that are defined in ETABS using the **Define menu > Material Properties** command. If you want to modify a material property then you modify the property in ETABS itself, not in the Section Designer utility.

Note that the material property defines both the concrete strength and the rebar yield stress.

To review or change the material property associated with a pier in Section Designer right click on the polygon area object that defines the pier to bring up the Section Information dialog box. In this dialog box one of the items you can change is the material property.

48

Tips and Tricks

Distort Feature

Often when you work with wall piers in Section Designer you find that the piers are very long (in the 2-axis direction) and very skinny (in the 3-axis direction). This can make it difficult to draw rebar in the pier. Section designer has a feature that allows you to distort the view of the pier by applying a distortion factor

to the 3-direction. Note that this distortion factor only applies to how you see the pier in Section Designer. It does not change the actual coordinates or dimensions of the pier.



Tip:

The distort feature makes it easier to specify rebar in a pier that is long and skinny.

If you are having trouble seeing or defining rebar because of the poor aspect ratio of a pier then click the **View menu > Distort 3-Direction** command. This pops up the 3-Direction Distortion Factor dialog box that allows you to specify a distortion factor for the 3-axis direction. If your pier is 120 inches long by 6 inches wide (120 inches in the 2-axis direction and 6 in the 3-axis direction) and you specify a 3-direction distortion factor of 2, then the pier will graphically appear to be 120 inches by 12 inches in Section Designer. Similarly, if the 3-direction distortion factor is input as 5 the 120-inch by 6-inch pier will appear to be 120 inches by 30 inches in Section Designer.

This feature can be very useful when you are working with wall piers.

Interaction Diagrams and Moment-Curvature Plots

You can view an interaction diagram or a moment curvature plot for your pier section at any time in Section Designer.

To view an interaction diagram simply click the **Show Interaction Surface** button, , located on the top toolbar, or click the **Display menu > Show Interaction Surface** command.

To view a moment-curvature plot simply click the **Show Moment-Curvature Curve** button, , located on the top toolbar, or click the **Display menu > Show Moment-Curvature Curve** command.

48

Pier Orientation

Take special care to be sure you are drawing your pier sections in Section Designer with the correct orientation. See the section titled "Local Axes Definition and Orientation" earlier in this chapter for more information.

Assigning Pier Sections

Once you define a pier section with user-defined reinforcing you must assign it to the pier. You can use the **Design menu > Shear Wall Design > Assign Pier Sections for Checking** command to do this. Note that you are also able to define the pier sections through this same command.



References

References

ASCE, 1995

Minimum Design Loads for Buildings and Other Structures - ASCE 7-95, American Society of Civil Engineers, New York, New York, 1995.

BOCA, 1996

The BOCA National Building Code/1996, Building Officials and Code Administrators International, Inc., Country Club Hills, Illinois, 1996.

CEN, 1994

ENV 1998-1-1:1994, Eurocode 8: Design Provisions for Earthquake Resistance of Structures - Part 1-1: General Rules- Seismic Actions and General Requirements for Structures, European Committee for Standardization, Brussels, Belgium, 1994.

R

R. W. Clough, I. P. King and E. L. Wilson, 1963

“Structural Analysis of Multistory Buildings,” *Journal of the Structural Division, ASCE*, Vol. 89, No. 8, 1963.

R. D. Cook, D. S. Malkus and M. E. Plesha, 1989

Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, New York, 1989.

A. K. Gupta, 1990

“Response Spectrum Method,” *Blackwell Scientific Publications, Ltd.*, 1990.

IBC, 1997

International Building Code 2000, International Code Council, Birmingham, Alabama, November, 1997.

NBCC, 1995

National Building Code of Canada, National Research Council of Canada, Ottawa, Canada, 1995.

NEHRP, 1997

NEHRP Recommended Provisions for Seismic Regulations for New Buildings and Other Structures (FEMA 302), Building Seismic Safety Council, Washington, D.C., 1997.

N. M. Newark and W. J. Hall, 1981

Earthquake Spectra and Design, Earthquake Engineering Research Institute, Berkeley, California, 1982.

NZS, 1992

Code of Practice for General Structural Design and Design Loadings for Buildings, Known as the Loadings Standard, Standards New Zealand, Wellington, New Zealand, 1992.

SEAOC, 1996

Recommended Lateral Force Requirements and Commentary, Structural Engineers Association of California, Sacramento, California, 1996.

UBC, 1994

Uniform Building Code, International Conference of Building Officials, Whittier, California, 1994.

UBC, 1997

Uniform Building Code, International Conference of Building Officials, Whittier, California, 1997.

D. W. White and J. F. Hajjar, 1991

“Application of Second-Order Elastic Analysis in LRFD: Research to Practice,” *Engineering Journal*, AISC, Vol. 28, No. 4, pp. 133–148.

E. L. Wilson, 1993

“An Efficient Method for the Base Isolation and Energy Dissipation Analysis of Structural Systems,” *ATC 17-1, Proceedings of Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control*, Applied Technology Council, Redwood City, California, 1993.

E. L. Wilson, 1997

Three Dimensional Dynamic Analysis of Structures with Emphasis on Earthquake Engineering, Computers and Structures, Berkeley, California, 1997.

E. L. Wilson and M. R. Button, 1982

“Three Dimensional Dynamic Analysis for Multicomponent Earthquake Spectra,” *Earthquake Engineering and Structural Dynamics*, Vol. 10.

E. L. Wilson, H. H. Dovey and A. Habibullah, 1981a

“Theoretical Basis for CTABS80: A Computer Program for Three-Dimensional Analysis of Building Systems,” *Technical Report K-81-2*, Computers/Structures International, Oakland, California, 1981.

E. L. Wilson and A. Ibrahimbegovic 1989

“Simple Numerical Algorithms for the Mode Superposition Analysis of Linear Structural Systems with Nonproportional Damping,” *Computers and Structures*, Vol. 33, No. 2, 1989.

E. L. Wilson, A. D. Kiureghian and E. Bayo, 1981b

“A Replacement for the SRSS Method in Seismic Analysis,” *Earthquake Engineering and Structural Dynamics*, Vol. 9, 1981.

E. L. Wilson and I. J. Tetsuji, 1983

“An Eigensolution Strategy for Large Systems,” *Computers and Structures*, Vol. 16.

E. L. Wilson, M. W. Yuan, and J. M. Dickens, 1982

“Dynamic Analysis by Direct Superposition of Ritz Vectors,” *Earthquake Engineering and Structural Dynamics*, Vol. 10, pp. 813–823.



Appendix 1

The ETABS Menu Structure

This appendix lays out the complete menu structure of ETABS. The twelve menus available in ETABS are:

- File menu
- Edit menu
- View menu
- Define menu
- Draw menu
- Select menu
- Assign menu
- Analyze menu
- Display menu
- Design menu
- Options menu
- Help menu

A1

All of the commands available in each of these menus are listed in this appendix. Submenu items are indented in the list.

File Menu Commands

- New Model...
- Open ...
- Save
- Save As...
- Import
 - Open ETABS7 .e2k Text File...
 - Open ETABS6 Text File...
 - Overwrite Story from SAFE .f2k Text File...
 - Overwrite Story from ETABS7 .edb File...
- Export
 - Save Model as ETABS7 .e2k Text File...
 - Save Model as SAP2000 .s2k Text File...
 - Save Story as SAFE .f2k Text File...
 - Save Story as ETABS7 .edb File...
 - Save Story Plan as .DXF...
 - Save as 3D .DXF
 - Save Input/Output as Access Database File...
 - Save Graphics as Enhanced MetaFile...
- Create Video...
 - Time History Animation...
 - Cyclic Animation...
- Print Setup...
- Print Preview for Graphics...
- Print Graphics
- Print Tables...
 - Input...
 - Analysis Output...
 - Steel Frame Design...
 - Concrete Frame Design...
 - Composite Beam Design...
 - Shear Wall Design...
- User Comments and Session Log...
- Display Input/Output Text Files...
- Exit

Edit Menu Commands

- Undo
- Redo
- Cut
- Copy
- Paste...
- Delete
- Add to Model from Template
 - Add 2D Frame...
 - Add 3D Frame...
- Replicate...
- Edit Grid Data...
- Edit Story Data
 - Edit...
 - Insert Story...
 - Delete Story...
- Edit Reference Planes...
- Edit Reference Lines...
- Merge Points...
- Align Points/Lines/Edges...
- Move Points/Lines/Areas...
- Expand/Shrink Areas...
- Merge Areas
- Mesh Areas...
- Join Lines
- Divide Lines...
- Auto Relabel All...

A1

View Menu Commands

- Set 3D View...
- Set Plan View...
- Set Elevation View...
- Set Building View Limits...
- Set Building View Options...
- Rubber Band Zoom
- Restore Full View
- Previous Zoom
- Zoom In One Step
- Zoom Out One Step
- Pan
- Measure
 - Line
 - Area
 - Angle
- Change Axes Location
- Show Selection Only
- Show All
- Save Custom View...
- Show Custom View...
- Refresh Window
- Refresh View

Define Menu Commands

- Material Properties...
- Frame Sections...
- Wall/Slab/Deck Sections...
- Link Properties...
- Frame Nonlinear Hinge Properties...
- Section Cuts...
- Response Spectrum Functions...
- Time History Functions...
- Static Load Cases...
- Response Spectrum Cases...
- Time History Cases...
- Static Nonlinear/Pushover Cases...
- Load Combinations...
- Mass Source...

A1

Draw Menu Commands

- Select Object
- Reshape Object
- Draw Point Objects
- Draw Line Objects
 - Draw Lines (Plan, Elev, 3D)
 - Create Lines in Region or at Clicks (Plan, Elev, 3D)
 - Create Columns in Region or at Clicks (Plan)
 - Create Secondary Beams in Region or at Clicks (Plan)
 - Create Braces in Region or at Clicks (Elev)
- Draw Area Objects
 - Draw Areas (Plan, Elev, 3D)
 - Draw Rectangular Areas (Plan, Elev)
 - Create Areas at Click (Plan, Elev)
 - Draw Walls (Plan)
 - Create Walls in Region or at Click (Plan)
- Draw Developed Elevation Definition...
- Draw Dimension Line
- Snap to
 - Grid Intersections and Points
 - Line Ends and Midpoints
 - Intersections
 - Perpendicular Projections
 - Lines and Edges
 - Fine Grid
- Constrain Drawn Line to
 - None
 - Constant X
 - Constant Y
 - Constant Z
 - Constant Angle

A1

Select Menu Commands

- Select at Pointer/in Window
- Select using Intersecting Line
- Select on XY Plane
- Select on XZ Plane
- Select on YZ Plane
- Select by Groups...
- Select by Frame Sections...
- Select by Wall/Slab/Deck Sections...
- Select by Link Properties...
- Select by Line Object Type...
- Select by Area Object Type...
- Select by Story Level...
- Select All
- Select Invert
- Deselect
 - at Pointer/in Window
 - using Intersecting Line
 - on XY Plane
 - on XZ Plane
 - on YZ Plane
 - by Groups...
 - by Frame Sections...
 - by Wall/Slab/Deck Sections...
 - by Link Properties...
 - by Line Object Type...
 - by Area Object Type...
 - by Story Level...
- All
- Invert

Get Previous Selection

Clear Selection

A1

Assign Menu Commands

Joint/Point

- Rigid Diaphragm...
- Panel Zone...
- Restraints (Supports)...
- Point Springs...
- Link Properties...
- Additional Point Mass...

Frame/Line

- Frame Section...
- Frame Releases/Partial Fixity...
- Frame Rigid Offsets...
- Frame Output Stations...
- Local Axes...
- Frame Property Modifiers...
- Link Properties...
- Frame NonLinear Hinges...
- Pier Label...
- Spandrel Label...
- Line Springs...
- Additional Line Mass...
- Automatic Frame Mesh/No Mesh...
- Mesh It
- Don't Mesh It
- Cancel

Shell/Area

- Wall/Slab/Deck Section...
- Opening...
- Rigid Diaphragm...
- Local Axes...
- Shell Property Modifiers...
- Pier Label...
- Spandrel Label...
- Area Springs...
- Additional Area Mass...

(Note: Assign menu commands continue on the next page)

Assign Menu Commands, continued

- Automatic Membrane Floor Mesh/No Mesh...
 - Mesh It
 - Don't Mesh It
 - Cancel
- Joint/Point Loads
 - Force...
 - Ground Displacement...
 - Temperature...
- Frame/Line Loads
 - Point...
 - Distributed...
 - Temperature...
- Shell/Area Loads
 - Uniform...
 - Temperature...
- Group Names...
- Clear Display of Assigns

Analyze Menu Commands

- Set Analysis Options...
- Run Analysis
- Run Static Nonlinear Analysis

A1

Display Menu Commands

- Show Undeformed Shape
- Show Loads
 - Joint/Point...
 - Frame/Line...
 - Shell/Area...
- Set Input Table Mode...
- Show Deformed Shape...
- Show Mode Shape...
- Show Member Forces/Stress Diagram
 - Support/Spring Reactions...
 - Frame/Pier/Spandrel Forces...
 - Shell Stresses/Forces...
 - Link Forces...
- Show Energy Diagram...
- Show Response Spectrum Curves...
- Show Time History Traces...
- Show Static Pushover Curve...
- Show Section Cut Forces...
- Set Output Table Mode...

Design Menu Commands

- Steel Frame Design
 - Select Design Group...
 - Select Design Combo...
 - View/Revise Overwrites...
 - Set Lateral Displacement Targets...
 - Start Design/Check of Structure
 - Interactive Steel Frame Design
 - Display Design Info...
 - Make Auto Select Section Null...
 - Change Design Section...
 - Reset Design Section to Last Analysis...
 - Verify Analysis vs Design Section...
 - Reset All Steel Overwrites...
 - Delete Steel Design Results...
- Concrete Frame Design
 - Select Design Combo...
 - View/Revise Overwrites...
 - Start Design/Check of Structure
 - Interactive Concrete Frame Design
 - Display Design Info...
 - Change Design Section...
 - Reset Design Section to Last Analysis...
 - Verify Analysis vs Design Section...
 - Reset All Concrete Overwrites...
 - Delete Concrete Design Results...
- Composite Beam Design
 - Select Design Group...
 - Select Design Combo...
 - View/Revise Overwrites...
 - Start Design/Check of Structure
 - Interactive Composite Beam Design
 - Display Design Info...
 - Make Auto Select Section Null...
 - Change Design Section...
 - Reset Design Section to Last Analysis...
 - Verify Analysis vs Design Section...
 - Reset All Composite Beam Overwrites...
 - Delete Composite Beam Design Results...

A1

(Note: Design menu commands continue on the next page)

Design Menu Commands, continued

Shear Wall Design
Select Design Combo...
View/Revise Pier Overwrites...
View/Revise Spandrel Overwrites...
Define Pier Sections for Checking...
Assign Pier Sections for Checking...
Start Design/Check of Structure
Interactive Wall Design
Display Design Info...
Reset All Pier/Spandrel Overwrites...
Delete Wall Design Results...
Overwrite Frame Design Procedure...

Options Menu Commands

- Preferences...
 - Dimensions/Tolerances...
 - Output Decimals...
 - Steel Frame Design...
 - Concrete Frame Design...
 - Composite Beam Design...
 - Shear Wall Design...
 - Reinforcement Bar Sizes...
 - Live Load Reduction...
- Colors
 - Display...
 - Output...
- Windows
 - One
 - Two Tiled Vertically
 - Two Tiled Horizontally
 - Three
 - Four
- Show Tips at Startup
- Show Bounding Plane
- Moment Diagrams on Tension Side
- Sound
- Lock Model
- Show Aerial View Window
- Show Floating Property Window
- Show Crosshairs

Help Menu Commands

- Search for Help on ...
- About ETABS...

A1



Index

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

\$set file, 3-5, 4-12, 8-20, 8-29, 20-4
2D view, 10-5, 10-6
3D view, 10-2, 10-8

A

acceleration loads, 33-11
active degrees of freedom, 15-1
add to model from template, 9-7
additional mass
 area, 14-56
 line, 14-38
 point, 14-14
aerial view window, 4-8, 18-23
align points, lines and edges, 9-20, 9-29
Analyze menu commands
 Set Analysis Options, 15-1
 Run Analysis, 15-8
 Run Static Nonlinear Analysis, 15-9
analysis, types
 eigenvector, 33-3
 linear elastic static, 33-2
 linear time history, 33-14

modal, 33-2
nonlinear time history, 33-14, 33-17
nonlinear static, 33-22
p-delta, 33-18
periodic time history (linear), 33-14
response spectrum (linear), 33-12
ritz-vector, 33-8
analysis log file, 43-1
analysis sections, 45-1, 45-13, 46-2, 46-10, 47-1,
 47-12, 47-13
angle drawing constraint, 12-22
area object, 23-1
 assignments to, 14-48, 23-5
drawing, 12-9
right click information, 23-6
type, 23-2
Assign menu commands
Joint/Point
 Rigid Diaphragm, 14-2
 Panel Zone, 14-3
 Restraints (Supports), 14-9
 Point Springs, 14-10
 Link Properties, 14-13
 Additional Point Mass, 14-14

ETABS User's Manual – Volume 2

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

I Frame/Line

- Frame Section, 14-22
- Frame Releases/Partial Fixity, 14-23
- Frame Rigid Offsets, 14-24
- Frame Output Stations, 14-28
- Local Axes, 14-29
- Frame Property Modifiers, 14-31
- Link Properties, 14-32
- Frame Nonlinear Hinges, 14-32
- Pier Label, 14-34
- Spandrel Label, 14-35
- Line Springs, 14-36
- Additional Line Mass, 14-38
- Automatic Frame Mesh/No Mesh, 14-39

Shell/Area

- Wall/Slab/Deck Section, 14-48
- Opening, 14-49
- Rigid Diaphragm, 14-49
- Local Axes, 14-50
- Shell Stiffness Modifiers, 14-51
- Pier Label, 14-52
- Spandrel Label, 14-53
- Area Springs, 14-54
- Additional Area Mass, 14-56
- Automatic Membrane Floor Mesh/No Mesh, 14-57

Joint/Point Loads

- Force, 14-16
- Ground Displacement, 14-18
- Temperature, 14-20

Frame/Line Loads

- Point, 14-40
- Distributed, 14-42
- Temperature, 14-46

Shell/Area Loads

- Temperature, 14-60
- Uniform Surface, 14-58

Group Names, 14-63

- Clear Display of Assigns, 14-64
- auto merge tolerance (preference), 18-2
- auto relabel all, 23-5
- auto select list, 11-11, 24-1
- auto zoom step (preference), 18-5

- automatic seismic loads
 - 1994 UBC, 28-6
 - 1995 NBCC (Canadian), 28-21
 - 1996 BOCA, 28-17
 - 1997 NEHRP, 28-30
 - 1997 UBC, 28-10
 - 1997 UBC isolated, 28-14
 - IBC2000, 28-25
 - user defined, 28-36
- automatic wind loads
 - 1994 UBC, 29-6
 - 1995 NBCC (Canadian), 29-14
 - 1996 BOCA, 29-11
 - 1997 UBC, 29-8
 - ASCE 7-95, 29-16
 - user defined, 29-19
- avi file, 8-26
- axes, 10-15, 21-2

B

- beams, secondary, 8-10, 12-6
- black objects on white background, 10-18, 44-2
- bounding plane, 18-3, 18-21
- breaking (dividing, meshing) line objects, 9-35
- buckling, 33-20, 33-21

C

- Cancel button, 4-14
- center of rigidity, 41-12
- charts
 - response spectrum function, 11-29
 - response spectrum curve from time history results, 16-29
 - time history function, 11-38
 - time history trace, 16-34
- clear display of assignments, 14-64
- codes, building
 - 1992 NZS 4203 (New Zealand), 11-37
 - 1994 UBC, 11-34, 28-6, 29-6
 - 1995 NBCC (Canada), 11-35, 28-21, 29-14
 - 1996 BOCA, 11-35, 28-17, 29-11
 - 1997 NEHRP, 11-36, 28-30

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

1997 UBC, 11-34, 28-10, 28-14, 29-8
 1998 Eurocode 8, 11-37
 ASCE 7-95, 29-16
 IBC2000, 11-36, 28-25
 colors
 display colors, 18-15
 frame element steel stress ratio colors, 18-17
 object fill colors, 10-18, 18-15
 output colors, 18-17
 shell element stress contour colors, 18-17
 view model by element or group colors, 10-17
 comments, user, 8-29
 composite beam design process, 47-3
 concrete frame design process, 46-4
 constraints
 drawing, 12-21
 rigid diaphragm, 14-2, 14-49, 23-9, 23-16, 25-5
 control key (Ctrl), 4-9, 4-10
 cookie cut meshing tools, 31-3
 coordinate systems, 21-1
 copying geometry, 9-2
 create a new model, 4-11, 6-1, 8-1
 crosshairs, 18-23
 coupled springs, 14-12
 cumulative center of mass, 41-12
 current units, 20-3

D

damping
 in link elements, 11-27
 modal, 11-27, 11-50, 33-15
 database, Microsoft Access, 8-26, 42-1
 database, section properties, 11-7
 database units, 20-4
 decimal places (preference), 18-6
 deck span direction, 14-49, 32-14
 default.edb file, 6-5, 8-2
 Define menu commands
 Material Properties, 11-1
 Frame Sections, 11-6
 Wall/Slab/Deck Sections, 11-21
 Link Properties, 11-26
 Frame Nonlinear Hinge Properties, 11-27

Section Cuts, 11-27
 Response Spectrum Functions, 11-29
 Time History Functions, 11-38
 Static Load Cases, 11-46
 Response Spectrum Cases, 11-50
 Time History Cases, 11-56
 Static Nonlinear/Pushover Cases, 11-63
 Load Combinations, 11-63
 Mass Source, 11-64
 deformed shape, display, 16-7
 deleting objects, 9-7
 deleting a story level, 9-18
 deselect, 13-6
 Design menu commands
 Steel Frame Design
 Select Design Group, 45-7
 Select Design Combo, 45-8
 View/Revise Overwrites, 45-8
 Set Lateral Displacement Targets, 45-9
 Start Design/Check of Structure, 45-10
 Interactive Steel Frame Design, 45-11
 Display Design Info, 45-11
 Make Auto Select Section Null, 45-11
 Change Design Section, 45-12
 Reset Design Section to Last Analysis, 45-13
 Verify Analysis vs Design Section, 45-13
 Reset All Steel Overwrites, 45-14
 Delete Steel Design Results, 45-14
 Concrete Frame Design
 Select Design Combo, 46-7
 View/Revise Overwrites, 46-7
 Start Design/Check of Structure, 46-8
 Interactive Concrete Frame Design, 46-8
 Display Design Info, 46-9
 Change Design Section, 46-9
 Reset Design Section to Last Analysis, 46-10
 Verify Analysis vs Design Section, 46-10
 Reset All Concrete Overwrites, 46-11
 Delete Concrete Design Results, 46-11
 Composite Beam Design
 Select Design Group, 47-7
 Select Design Combo, 47-8
 View/Revise Overwrites, 47-9
 Start Design/Check of Structure, 47-10

ETABS User's Manual – Volume 2

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

- I Interactive Composite Beam Design, 47-10
- Display Design Info, 47-11
- Make Auto Select Section Null, 47-11
- Change Design Section, 47-11
- Reset Design Section to Last Analysis, 47-12
- Verify Analysis vs Design Section, 47-13
- Reset All Composite Beam Overwrites, 47-13
- Delete Composite Beam Design Results, 47-13
- Shear Wall Design
 - Select Design Combo, 48-11
 - View/Revise Pier Overwrites, 48-12
 - View/Revise Spandrel Overwrites, 48-15
 - Define Pier Sections for Checking, 48-17
 - Assign Pier Sections for Checking, 48-17
 - Start Design/Check of Structure, 48-17
 - Interactive Wall Design, 48-17
 - Display Design Info, 48-18
 - Reset All Pier/Spandrel Overwrites, 48-19
 - Delete Wall Design Results, 48-19
- Overwrite Frame Design Procedure, 17-2
- design postprocessors, 6-5, 6-6, 6-9, 17-1, 24-2
- design procedure, 17-2, 24-7
- design process
 - composite beam design, 47-3
 - concrete frame design, 46-4
 - shear wall design, 48-8
 - steel frame design, 45-3
- design sections, 45-1, 45-13, 46-2, 46-10, 47-1, 47-12, 47-13
- developed elevation, 6-6, 10-7, 10-9, 12-11, 12-12
- diaphragm, rigid
 - assign to area object, 14-49
 - assign to point objects, 14-2
 - display diaphragm extent, 10-29
- dimension lines, 9-40, 12-17
- dimensions, measurements, 10-15
- dimensions, preferences, 18-2
- displacement
 - ground (input static load), 14-18
 - deformed shape (static load), 16-7
- Display menu commands
 - Show Undeformed Shape, 16-1
- Show Loads
 - Joint/Point, 16-2
 - Frame/Line, 16-3
 - Shell/Area, 16-5
- Set Input Table Mode, 16-6
- Show Deformed Shape, 16-7
- Show Mode Shape, 16-12
- Show Member Forces/Stress Diagram
 - Support/Spring Reactions, 16-14
 - Frame/Pier/Spandrel Forces, 16-17
 - Shell Stresses/Forces, 16-20
 - Link Forces, 16-26
- Show Energy Diagram, 16-27
- Show Response Spectrum Curves, 16-29
- Show Time History Traces, 16-34
- Show Static Pushover Curve, 16-39
- Show Section Cut Forces, 16-39
- Set Output Table Mode, 16-40
- divide lines, 9-35
- Draw menu commands
 - Select Object, 12-1
 - Reshape Object, 9-37
 - Draw Point Objects, 12-3
 - Draw Line Objects
 - Draw Lines (Plan, Elev, 3D), 12-4
 - Create Lines in Region or at Clicks (Plan, Elev, 3D), 12-5
 - Create Columns in Region or at Clicks (Plan), 12-6
 - Create Secondary Beams in Region or at Clicks (Plan), 12-6
 - Create Braces in Region or at Clicks (Elev), 12-7
 - Draw Area Objects
 - Draw Areas (Plan, 3D), 12-9
 - Draw Rectangular Areas (Plan, Elev), 12-10
 - Create Areas at Click (Plan, Elev), 12-10
 - Draw Walls (Plan), 12-10
 - Create Walls in Region or at Click (Plan), 12-12
 - Draw Developed Elevation Definition, 12-12
 - Draw Dimension Line, 12-17
 - Snap to
 - Grid Intersections and Points, 12-18

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

Line Ends and Midpoints, 12-19
 Intersections, 12-19
 Perpendicular Projections, 12-19
 Lines and Edges, 12-19
 Fine Grid, 12-19
 Constrain Drawn Line to
 None, 12-22
 Constant X, 12-21
 Constant Y, 12-21
 Constant Z, 12-21
 Constant Angle, 12-22
 draw mode, 4-12
 dynamic load participation ratio, 41-8
 dxf export, 8-24, 8-25

E

e2k file, 3-5, 4-12, 8-21, 8-22, 8-29, 20-3
 ebk file, 20-4
 edb file, 3-5, 4-12, 8-2, 8-7, 8-19, 8-20, 8-22, 8-24, 20-4
 emf file, 8-26, 44-2
 earthquake load *See automatic seismic loads*
 edge select mode, 9-31
 Edit menu commands
 Undo, 4-14
 Redo, 4-14
 Cut, 9-2
 Copy, 9-2
 Paste, 9-2
 Delete, 9-7
 Add to Model from Template
 Add 2D Frame, 9-8
 Add 3D Frame, 9-8
 Replicate, 9-9
 Edit Grid Data, 9-14
 Edit Story Data
 Edit, 9-17
 Insert Story, 9-17
 Delete Story, 9-18
 Edit Reference Planes, 9-18
 Edit Reference Lines, 9-18
 Merge Points, 9-19
 Align Points/Lines/Edges, 9-20

Move Points/Lines/Areas, 9-29
 Expand/Shrink Areas, 9-30
 Merge Areas, 9-32
 Mesh Areas, 31-2
 Join Lines, 9-33
 Divide Lines, 9-35
 Auto Relabel All, 23-5
 eigenvalue, 33-3
 eigenvector analysis, 33-3
 elevation, developed, 10-7, 12-11, 12-12
 elevation, story level, 8-6, 9-17
 elevation view, 10-6
 energy diagram, 16-27
 export options
 Save Model as ETABS7 .e2k Text File, 8-22
 Save Model as SAP2000 .s2k Text File, 8-23
 Save Story as SAFE .f2k Text File, 8-23
 Save Story as ETABS7 .edb File, 8-24
 Save Story Plan as .DXF, 8-24
 Save as 3D .DXF, 8-25
 Save Input/Output as Access Database File, 8-26
 Save Graphics as Enhanced MetaFile, 8-26
 expand area objects, 9-30
 extend line objects, 9-25

F

fast restraints, 14-10
 fax number, 3-4
 File menu commands
 New Model, 8-1
 Open, 8-19
 Save, 8-20
 Save As, 8-20
 Import
 Open ETABS7 .e2k Text File, 8-21
 Open ETABS6 Text File, 8-21
 Overwrite Story from SAFE .f2k Text File, 8-22
 Overwrite Story from ETABS7 .edb File, 8-22
 Export
 Save Model as ETABS7 .e2k Text File, 8-22
 Save Model as SAP2000 .s2k Text File, 8-23

ETABS User's Manual – Volume 2

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

Save Story as SAFE .f2k Text File, 8-23

Save Story as ETABS7 .edb File, 8-24

Save Story Plan as .DXF, 8-24

Save as 3D .DXF, 8-25

Save Input/Output as Access Database File,
8-26

Save Graphics as Enhanced MetaFile, 8-26

Create Video

 Time History Animation, 8-26

 Cyclic Animation, 8-27

Print Preview for Graphics, 8-27

Print Graphics, 8-27

Print Tables

 Input, 8-28

 Analysis Output, 8-29

 Steel Frame Design, 8-28

 Concrete Frame Design, 8-28

 Composite Beam Design, 8-28

 Shear Wall Design, 8-28

User Comments and Session Log, 8-29

Display Input/Output Text Files, 8-29

Exit, 8-30

floating property window, 12-8, 12-12

font size, 18-3, 18-4

frame element internal forces, 35-2

frame section properties, 11-6, 14-22

frequency, 33-4, 41-3

functions, response spectrum

 1992 NZS 4203 (New Zealand), 11-37

 1994 UBC, 11-34

 1995 NBCC (Canada), 11-35

 1996 BOCA, 11-35

 1997 NEHRP, 11-36

 1997 UBC, 11-34

 1998 Eurocode 8, 11-37

 from text file, 11-30

 IBC2000, 11-36

 user-defined, 11-32

functions, time history

 cosine, 11-44

 from a file, 11-38

 ramp, 11-44

 sawtooth, 11-45

 sine, 11-43

triangular, 11-46

user-defined, 11-41

user-defined periodic, 11-41

G

getting started

 creating a model, 6-1, 8-1

 installing ETABS, 2-1

global force balance, 43-3

graphical user interface, 4-1

grid line systems, 9-14, 21-1

ground displacement, 14-18

groups, 14-63, 26-1

groups, designing by, 26-3

groups, section cuts, 26-4

groups, selecting by, 13-4, 26-3

H

Help menu commands

 Search for Help on, 19-1

 About ETABS, 19-1

height, story level, 8-6, 9-17

I

import options

 Open ETABS7 .e2k Text File, 8-21

 Open ETABS6 Text File, 8-21

 Overwrite Story from SAFE .f2k Text File, 8-22

 Overwrite Story from ETABS7 .edb File, 8-22

incremental analysis, 33-22

initial p-delta analysis, 33-18

initialization of model, 8-2

inserting a story level, 9-17

installation

 hardware key device, 2-11

 network server, 2-5

 network workstation, 2-6

 sentinel driver, 2-9

 single user, 2-4

 troubleshooting, 2-14

interactive composite beam design, 47-14

I

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

interactive steel frame design, 45-14
 interactive concrete frame design, 46-11
 interactive shear wall design, 48-17
 intersecting line select mode, 13-3
 invert selection, 13-5
 invisible grid for snapping, 12-19

J

join lines, 9-33
 joint *See point object*

K

keyboard commands *See inside front cover*

L

limits, viewing, 10-10
 line objects
 assignments to, 14-22, 24-5
 drawing, 12-3
 right click information, 24-7
 type, 24-3
 line, reference, 9-18
 line thickness (preference)
 printer, 18-3
 screen, 18-3
 linear static analysis, 33-2
 link element
 assigned to line objects, 14-32
 assigned to panel zones, 14-5
 assigned to point objects (grounded), 14-13
 internal deformations, 37-4
 internal forces, 37-6
 internal nonlinear springs, 37-2
 properties, 11-26
 live load reduction, 11-47, 18-10
 loads, assignment
 area object
 temperature load, 14-60
 uniform surface load, 14-58
 line object
 distributed load, 14-42

point load, 14-40
 temperature load, 14-46
 point object
 force load, 14-16
 ground displacement, 14-18
 temperature load, 14-20
 loads, displaying on model
 joint/point, 16-2
 frame/line, 16-3
 shell/area, 16-5
 load cases
 response spectrum, 11-50, 27-4
 static, 11-46, 27-2
 static nonlinear (pushover), 11-63, 27-6
 time history, 11-56, 27-5
 load combinations, 27-6
 load transformation, 32-1
 local axes
 area object, 14-50, 23-17
 coordinate system, 9-15, 21-4
 frame element, 35-1
 line object, 14-29, 24-29
 link element, 14-13, 37-2
 panel zone, 14-8
 point object, 25-12
 shell element, 36-1
 section cut, 11-29, 26-6
 wall pier, 38-2
 wall spandrel, 38-5
 locking model, 4-13, 18-22
 log file, 43-1

M

magnifying the view *See zoom*
 major axis, 14-30
 major direction, 14-30
 mass, additional
 area, 14-56
 line, 14-38
 point, 14-14
 mass per unit volume, 11-4
 mass source, 11-64, 27-11
 material property, 11-1

ETABS User's Manual – Volume 2

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

maximum graphic font size (preference), 18-3
measurements

angle between two lines, 10-16
perimeter and area of an area, 10-15
line length, 10-15
membrane property, 11-21, 11-22, 30-1
merge
 areas, 9-32
 points, 9-19
 tolerance, 9-19, 18-2

mesh areas
 automatic, 30-1
 manual, 31-1

mesh line objects *See divide lines*

Microsoft Access *See database, Microsoft Access*

Microsoft Excel *See spreadsheet, copying geometry to and from*

minimum graphic font size (preference), 18-4

minor axis, 14-30
minor direction, 14-30
modal analysis, 33-2
modal direction factors, 41-5
modal effective mass factors, 41-5
modal participation factors, 41-4
modal periods and frequencies, 41-3
mode shapes, 16-12, 41-3

modifiers
 frame property, 14-31
 shell stiffness, 14-51

modulus
 shear, 11-3
 Young's, 11-2

mouse, using, 4-9
moving objects, 9-29

N

| nonlinear
 frame hinge properties (pushover), 11-27
 link properties, 11-26, 37-2
 static analysis (pushover), 33-22
 time history analysis, 33-17
nsrvgx.exe, 2-13
nudging objects, 9-42

O

OK button, 4-14
openings, 14-49
Options menu commands
 Preferences
 Dimensions/Tolerances, 18-2
 Output Decimals, 18-6
 Reinforcement Bar Sizes, 18-7
 Live Load Reduction, 18-10
Colors
 Display, 18-15
 Output, 18-17
Windows, 18-20
Show Tips at Startup, 18-20
Show Bounding Plane, 18-21
Moment Diagrams on Tension Side, 18-22
Sound, 18-22
Lock Model, 18-22
Show Aerial View Window, 18-23
Show Floating Property Window, 18-23
Show Crosshairs, 18-23
output
 conventions, 34-1, 35-1, 36-1, 37-1, 38-1, 39-1,
 41-13
 decimals (preferences), 18-6
 display colors, 18-15
 displayed on screen, 16-1
 onscreen output tables, 16-6, 16-40
 printed to printer or file, 8-27
 stations, 14-28
out file, 43-3
overlapping area objects, 23-15
overturning moments, 41-13
overwrite frame design procedure, 17-2

P

pan, 10-13
pan margin (preference), 18-4
panel zone
 assignments, 14-3
 displacements, 34-3
 internal deformations, 34-4

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

internal forces, 34-5
 p-delta analysis, 33-18
 period, 41-3
 perspective view, 10-8
 phone number, 3-4
 piers
 labels, 14-34, 14-52, 48-1
 output forces, 16-17, 38-4
 plan fine grid spacing (preference), 18-2
 plan nudge value (preference), 18-3
 plane
 bounding, 18-21
 reference, 9-18
 plate bending, 11-22
 point object
 assignments to, 14-1, 25-3
 drawing, 12-3
 on top of another point object, 25-12
 output conventions, 34-1
 right click information, 25-3
 polyline, 24-31
 preferences, 18-1
 print
 graphics, 8-27
 preview, 8-27
 tables, 8-28

Q

quadrilaterals, drawing *See Draw menu commands*
 quick keys *See inside front cover*
 quitting ETABS, 8-30

R

reactions, 16-14, 34-2
 readme.txt, 2-5
 rectangles, drawing *See Draw menu commands*
 redo, 4-14
 reduction, live load, 18-10
 reference lines, 9-18
 reference planes, 9-18
 refresh view, 10-14

refresh window, 10-14
 reinforcing
 bar sizes (preference), 18-7
 beam, 11-17
 column, 11-19
 relabeling objects, 23-5
 replicating objects, 9-9
 reshaper tool, 9-37
 residual mass modes, 33-7
 response spectrum analysis, 33-12
 response spectrum analysis output
 damping and accelerations, 41-10
 modal amplitudes, 41-11
 base reactions, 41-11
 response spectrum curve
 from time history analysis results, 16-29
 input function, 11-29
 restore previous selection, 13-5
 restraints, 14-9
 right click
 on area object, 23-6
 on line object, 24-7
 on point object, 25-3
 right hand rule, 23-18
 rigid diaphragm assignment
 to area object, 14-49
 to point objects, 14-2
 ritz-vector analysis, 33-8
 rotate 3D view, 10-5
 rubber band line, 12-4, 12-11, 13-3
 rubber band window, 12-3, 12-5, 12-6, 12-7, 12-12, 13-2

S

saving model, 8-20
 screen selection tolerance (preference), 18-3
 screen snap to tolerance (preference), 18-3
 screen line thickness (preference), 18-3
 secondary beams, 12-6
 section cuts
 defining, 11-27, 26-4
 output forces, 26-7, 39-1
 section designer, 11-11, 48-20

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

seismic load *See automatic seismic loads*
select all, 13-5
select edge of area object, 9-31
Select menu commands
 Select at Pointer/in Window, 13-2
 Select using Intersecting Line, 13-3
 Select on XY Plane, 13-4
 Select on XZ Plane, 13-4
 Select on YZ Plane, 13-4
 Select by Groups, 13-4
 Select by Frame Sections, 13-4
 Select by Wall/Slab/Deck Sections, 13-4
 Select by Link Properties, 13-4
 Select by Line Object Type, 13-5
 Select by Area Object Type, 13-5
 Select by Story Level, 13-5
 Select All, 13-5
 Select Invert, 13-5
 Deselect, 13-6
 Get Previous Selection, 13-5
 Clear Selection, 13-6
self-mass, 11-4, 11-64
self-weight, 11-4, 11-24, 11-47
select mode, 4-12
sequential analysis *See incremental analysis*
shear wall design process, 48-8
shell element faces, 36-2
shell element section properties, 11-21
shell element internal forces, 36-3
shell element internal stresses, 36-8
shift key, 4-9, 4-10
show all, 10-11
show selection only, 10-11
shrink area objects, 9-30
shrink factor (preference), 18-5
similar stories feature, 12-2, 22-3
snap to options
 intersections, 12-19
 invisible grid, 12-19
 lines and edges, 12-19
 middle and ends, 12-19
 perpendicular, 12-19
 points, 12-18
sound, 18-22

spandrels
 labels, 14-35, 14-53, 48-5
 output forces, 16-17, 38-5
spreadsheet, copying geometry to and from, 9-3
spring
 forces, 16-14
 properties
 area, 14-54
 line, 14-36
 point, 14-10
starting a new model, 4-11, 6-1, 8-1
starting load vectors, 33-10
static analysis, 33-2
static load participation ratio, 41-7
static pushover curve, 16-39
status bar, 4-5
steel frame design process, 45-3
story levels, 8-6, 9-17, 22-1
story shears, 41-13
support, technical
 e-mail support, 3-5
 fax support, 3-4
 phone support, 3-4
 training, at CSI in Berkeley, 3-6
supports
 area spring, 14-54
 line spring, 14-36
 point spring, 14-10
reactions, 16-14
restraints, 14-9

T

tables displayed onscreen
input tables, 16-6
output tables, 16-40
static nonlinear analysis tables, 16-39
time history trace tables, 16-38
telephone numbers
 fax, 3-4
 voice, 3-4
templates, 8-6, 9-7, 11-42
thick plate, 11-22

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

- time history
 - analysis, 33-14
 - case, 11-56
 - function, 11-38
 - trace, 16-34
- time history types
 - linear, 33-14
 - nonlinear, 33-14
 - periodic, 33-14
- tips, showing at startup, 18-20
- tolerances, preferences, 18-2
- toolbar buttons
 - main (top) toolbar buttons
 - New Model, 8-1
 - Open .EDB File, 8-19
 - Save Model, 8-20
 - Undo, 4-14
 - Redo, 4-14
 - Refresh Window, 10-14
 - Lock/Unlock Model, 18-22
 - Run Analysis, 15-8
 - Rubber Band Zoom, 10-11
 - Restore Full View, 10-12
 - Restore Previous Zoom, 10-12
 - Zoom In One Step, 10-13
 - Zoom Out One Step, 10-13
 - Pan, 10-14
 - 3D View, 10-2, 10-5
 - Plan View, 10-5
 - Elevation View, 10-6
 - Rotate 3D View, 10-5
 - Perspective Toggle, 10-8
 - Move Up in List, 10-5
 - Move Down in List, 10-5
 - Object Shrink Toggle, 10-18
 - Set Building View Options, 10-16
 - Show Undeformed Shape, 16-1
 - Display Static Deformed Shape, 16-7
 - Display Mode Shape, 16-12
 - Display Member Force Diagram, 16-14
 - Display Output Tables, 16-40
 - side toolbar buttons
 - Pointer, 12-1
 - Reshaper, 9-37
- Draw Point Objects (displays flyout button), 12-3
- Create Points (plan, elev, 3D), 12-3
- Draw Line Objects (displays flyout buttons), 12-3
- Draw Lines (plan, elev, 3D), 12-4
- Create Lines in Region or at Clicks (all views), 12-5
- Create Columns in Region or at Clicks (plan), 12-6
- Create 2ndary Beams in Region or at Clicks (plan), 12-6
- Create Braces in Region or at Clicks (elev), 12-7
- Draw Area Objects (displays flyout buttons), 12-9
- Draw Areas (plan, 3D), 12-9
- Draw Rectangular Areas (plan, elev), 12-10
- Create Areas at Click (plan, elev), 12-10
- Draw Walls (plan), 12-10
- Create Walls in Region or at Click (plan), 12-12
- Select All, 13-5
- Restore Previous Selection, 13-5
- Clear Selection, 13-6
- Set Intersecting Line Select Mode, 13-3
- Snap to Points, 12-18
- Snap to Middle and Ends, 12-19
- Snap to Intersections, 12-19
- Snap to Perpendicular, 12-19
- Snap to Lines and Edges, 12-19
- Snap to Invisible Grid, 12-19
- training, at CSI in Berkeley, 3-6
- trim line objects, 9-25

U

- undeformed shape, display, 16-1
- undo, 4-14
- units, 20-1
- unlocking model, 4-13, 18-22
- unstable end releases, 14-24

I

Note: Page numbers are reported as X-n where X is the Chapter number and n is the page number in the chapter. Chapters 1 through 19 are in Volume 1 and Chapters 20 through 48 are in Volume 2.

V

video, create and playback, 8-26

View menu commands

Set 3D View, 10-2

Set Plan View, 10-5

Set Elevation View, 10-6

Set Building View Limits, 10-10

Set Building View Options, 10-16

Rubber Band Zoom, 10-11

Restore Full View, 10-12

Previous Zoom, 10-12

Zoom In One Step, 10-13

Zoom Out One Step, 10-13

Pan, 10-13

Measure

Line, 10-15

Area, 10-15

Angle, 10-16

Change Axes Location, 10-15

Show Selection Only, 10-11

Show All, 10-11

Save Custom View, 10-9

Show Custom View, 10-9

Refresh Window, 10-14

Refresh View, 10-14

Y

Young's modulus, 11-2

Z

zoom

in one step, 10-13

out one step, 10-13

previous zoom, 10-12

restore full view, 10-12

rubber band zoom, 10-11

W

walls

drawing *See Draw menu commands, Draw*

Area Objects

assigning pier labels to area objects, 14-52

assigning pier labels to line objects, 14-34

assigning spandrel labels to area objects, 14-53

assigning spandrel labels to line objects, 14-35

wind load *See automatic wind loads*

windows, Options menu command, 18-20

windowing, 13-2

world wide web address, 3-3

I