

# ETABS<sup>®</sup> V20

Powered by **SAPFIRE<sup>®</sup>**

Analysis and Design  
of Building Systems



**User's Guide**

A PRODUCT OF COMPUTERS & STRUCTURES, INC.  
Proudly developed in the United States of America



# User's Guide **ETABS®**

**Integrated Building Design Software**

# Copyright

Copyright © Computers and Structures, Inc., 1978-2021  
All rights reserved.

The CSI Logo<sup>®</sup>, SAP2000<sup>®</sup>, ETABS<sup>®</sup>, and SAFE<sup>®</sup> are registered trademarks of Computers and Structures, Inc. Watch & Learn<sup>™</sup> is a trademark of Computers and Structures, Inc.

The computer programs SAP2000<sup>®</sup>, ETABS<sup>®</sup>, SAFE<sup>®</sup> and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of these programs or reproduction of documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior explicit written permission of the publisher.

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

Computers and Structures, Inc.

<http://www.csiamerica.com/>

[info@csiamerica.com](mailto:info@csiamerica.com) (for general information)

[support@csiamerica.com](mailto:support@csiamerica.com) (for technical support)

# DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND TESTING OF THIS SOFTWARE. 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 THIS PRODUCT.

THIS PRODUCT IS A PRACTICAL AND POWERFUL TOOL FOR STRUCTURAL DESIGN. HOWEVER, THE USER MUST EXPLICITLY UNDERSTAND THE BASIC ASSUMPTIONS OF THE SOFTWARE MODELING, ANALYSIS, AND DESIGN ALGORITHMS AND COMPENSATE FOR THE ASPECTS THAT ARE NOT ADDRESSED.

THE INFORMATION PRODUCED BY THE SOFTWARE MUST BE CHECKED BY A QUALIFIED AND EXPERIENCED ENGINEER. THE ENGINEER MUST INDEPENDENTLY VERIFY THE RESULTS AND TAKE PROFESSIONAL RESPONSIBILITY FOR THE INFORMATION THAT IS USED.

---

## Contents

---

# User's Guide

<b>1</b>	<b>Program Description</b>	
	Objective	1-1
	This is ETABS	1-1
	Time Saving Options	1-3
	Templates and Defaults	1-3
	Basic Process	1-4
	Forms	1-5
<b>2</b>	<b>ETABS “Screen”</b>	
	Objective	2-1
	The ETABS Window	2-1
	File Operations	2-4
	Edit	2-4
	View	2-4
	Define	2-5
	Draw	2-5
	Select	2-5
	Assign	2-6
	Analyze	2-6

	Display	2-7
	Design	2-8
	Tools	2-8
	Options	2-8
	Help	2-8
<b>3</b>	<b>Basic Modes, Drawing Tools, Mouse Pointers</b>	
	Objective	3-1
	Select or Draw	3-1
<b>4</b>	<b>Begin a Model</b>	
	Objective	4-1
	Create the Basic Grid System	4-1
	Grid Dimensions (Plan) – Define a Grid System	4-4
	Story Dimensions - Define Story Data	4-5
<b>5</b>	<b>Create the Structural Model</b>	
	Objective	5-1
	Add Structural Objects Using Templates	5-1
	Define Properties	5-4
	Material Properties	5-4
	Frame Sections	5-5
	Auto Select Section List	5-8
	Add Structural Objects Manually	5-11
	Draw Columns	5-11
	Draw Beams	5-13
	Draw Secondary (Infill) Beams	5-15
	Draw the Floor	5-15

	Draw Walls	5-17
	Draw Wall Stacks	5-18
	Draw Tendons	5-19
<b>6</b>	<b>Select Structural Objects</b>	
	Objective	6-1
	Selecting	6-1
	Graphical Selection Options	6-1
	Selecting by Coordinates	6-4
	Selecting by Feature	6-4
	Deselect Command	6-5
	Invert Selection Command	6-5
	Get Previous Selection Command	6-5
	Clear Selection Command	6-6
<b>7</b>	<b>Assign/Change Properties</b>	
	Objective	7-1
	Assign	7-1
	Assign the AUTOLATBM Auto Select Section List	7-4
	Make an Assignment as the Object is Drawn	7-5
	Make an Assignment using the Model Explorer	7-6
	Check the Sections in an Auto Select Section List	7-6
<b>8</b>	<b>Load the Structural Model</b>	
	Objective	8-1
	Structural Loads	8-1

	Define the Load Patterns	8-2
	Auto Lateral Load	8-2
	Self-Weight Multiplier	8-5
	Modify an Existing Load Pattern	8-5
	Delete an Existing Load Pattern	8-6
	Define Shell Uniform Load Sets	8-6
	Assign Structural Loads	8-7
<b>9</b>	<b>Define Load Cases</b>	
	Objective	9-1
	Review/Create Load Cases	9-1
	Define an Auto Construction Sequence Case	9-4
	Define a Walking Vibration Case	9-5
<b>10</b>	<b>Edit the Model Geometry</b>	
	Objective	10-1
	Editing Options	10-1
<b>11</b>	<b>Analyze the Model</b>	
	Objective	11-1
	Set the Mesh Options	11-1
	Model Analysis	11-2
	Model Alive™ Feature	11-4
	Locking and Unlocking the Model	11-4
<b>12</b>	<b>Design</b>	
	Objective	12-1
	Design the Structure	12-1



**13 Display Results**

Objective	13-1
Obtain Basic Graphical Displays	13-1
Graphical Displays using Model Explorer	13-4
Tabular Display of Results	13-5

**14 Generate Results**

Objective	14-1
Summary Report	14-1
Print Graphics	14-2
Export Results	14-2

# Program Description

## Objective

This chapter briefly describes the program and some of the concepts involved in its use.

## This Is ETABS

ETABS is a powerful program that can greatly enhance an engineer's analysis and design capabilities for structures. Part of that power lies in an array of options and features. The other part lies in how simple it is to use.

The basic approach for using the program is very straightforward. The user establishes grid lines, places structural objects relative to the grid lines using joints, frames, links, tendons, and shells, and assigns loads and structural properties to those structural objects (for example, a frame object can be assigned section properties; a joint object can be assigned spring properties; a shell object can be assigned slab or deck properties).

Analysis and design are then performed based on the structural objects and their assignments. Results are generated in graphical or tabular form that can be printed to a printer or to a file for use in other programs.

In using the program, you manage the **File**, **Edit** the model, change the **View**, **Define** properties or load patterns and cases, **Draw** something new in the model, **Select** that something, **Assign** properties or loads, **Analyze** the model, **Display** analysis results for checking, **Design** the structure, apply various **Options** to achieve the desired outcome with optimum effort, utilize plugin **Tools** to customize the program, and seek **Help** when you need it. Those actions are the basis for the program menu structure. Thus, familiarity with the menu commands and their function is key to expanding your ability to use ETABS.

Information about the various menu items is available using the **Help menu > ETABS Help** command as well as by using the F1 key when a form is displayed on the ETABS screen. The F1 key will display context sensitive help, including descriptions of the types of input for the forms used in the program. Familiarity with the menu commands will enable the user to create models for complex Composite Floor Framing Systems with Openings and Overhangs, Steel Joist Systems, Moment Resisting Frames, complex Shear Wall Systems, Rigid and Flexible Floors, Sloped Roofs, Ramps and Parking Structures, Mezzanine Floors, Concrete Slabs with Post-Tensioning, Trussed Systems, Multiple Tower Buildings and Stepped Diaphragm Systems, and many more.

Design manuals in .pdf format are available using the **Help menu > Documentation** command. Those manuals explain how the program performs steel frame design, concrete frame design, composite beam design, composite column design, steel joist design, concrete shear wall design, and concrete slab design in accordance with applicable building codes.

### **ETABS Menu Commands:**

- *File*
- *Edit*
- *View*
- *Define*
- *Draw*
- *Select*
- *Assign*
- *Analyze*
- *Display*
- *Design*
- *Options*
- *Tools*
- *Help*

## Time Saving Options

The program also includes options that allow you to reduce the time spent creating models. Those options include the following:

- **Similar Stories.** Allows the user to make changes to multiple stories simultaneously.
- **Snap To.** Allows the user to place structural objects with accuracy.
- **Auto Select Sections.** Allows the user to define a list of sections, for example W18X35, W18X40, W21X44, W21X50 and W24X55, that can be assigned to a frame member. The program can then automatically select the most economical, adequate section from the auto select section list when it is designing the member.
- **Vertical Load Transfer.** Frees the user from the chore of calculating the load on the members supporting the floor plate, and determines the area tributary to each member for live load reduction.
- **Wall Stacks.** Allows the user to quickly generate complex wall arrangements.
- **Towers.** Allows multiple towers to exist within a single model.
- **Model Explorer.** Allows the user to rapidly create and modify models using a hierarchical tree system with drag-&-drop capability.
- **Design Strips.** Allows for accurate layout of program calculated slab reinforcement and tendon placement.

## Templates and Defaults

ETABS provides a number of templates that allow for the rapid generation of models for a wide range of common building types. Those templates serve as a good starting point because they can be modified easily.

The program includes default parameters, many of which are building code specific. Those defaults are accessed using "Overwrites" and "Pref-

erences." The possible options available for overwrites and the default values for preferences are identified in the design manuals.

By using the built-in templates and defaults, the user can create a model in a matter of minutes.

## Basic Process

The following provides a broad overview of the basic modeling, analysis, and design processes:

1. Select the Base Units and Design Codes
2. Set up Grid Lines
3. Define Story Levels
4. Define Section Properties
5. Draw Structural Objects
6. Select Objects
7. Assign Properties
8. Define Load Patterns
9. Assign Loads
10. Define Load Cases
11. Edit the Model Geometry
12. View the Model
13. Analyze the Model
14. Display Results for Checking
15. Design the Model
16. Output Results and Reports
17. Save the Model

## Forms

Various forms are used in ETABS throughout the modeling, analysis, and design processes. With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

# ETABS “Screen”

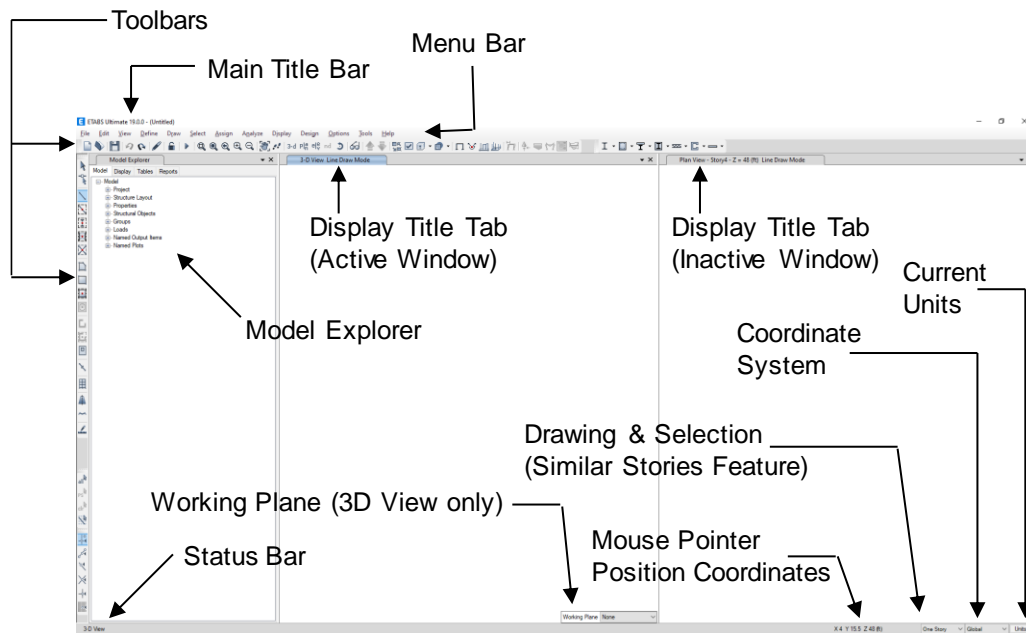
## Objective

This chapter briefly describes the ETABS “screen” or more accurately, the graphical user interface.

## The ETABS Window

The ETABS graphical user interface shown in Figure 2-1 includes the main window, main title bar, display title tabs, menu bar, toolbars, model explorer, display windows, status bar, mouse pointer position coordinates and the current units. Each of these items is described in the bulleted list that follows.

- **Main Window.** This window may be moved, resized, maximized, minimized, or closed using standard Windows operations.



**Figure 2-1: The ETABS graphical user interface**

**Main Title Bar.** The main title bar includes the program name and version and the model name. Move the main window by left clicking in the main title bar and holding down the mouse button as you drag the window around the computer screen.

- **Menu Bar.** The menu bar contains the program's menus from which various commands can be selected to perform specific actions.
- **Toolbars and Buttons.** Toolbars are made up of buttons. Buttons provide "single-click" access to commonly used commands. Holding the mouse pointer over a toolbar button for a few seconds without clicking or holding down any mouse buttons will display a short description of the button's function in a small text box.
- **Model Explorer.** The model explorer allows easy access to model definition data, including property forms, load definitions, and object forms, as well as analysis and design results in graphical, tabular, and report formats using a hierarchical tree structure. These items are grouped in four tabs in the *Model Explorer*, namely Model, Display,



Tables, and Reports. Trees may be expanded by clicking on a node, and a right click on a "leaf" in the tree will bring up a context-sensitive menu (items shown in **bold** in the menu are the default action that will occur if the user double clicks on the leaf). Sections may be assigned to a model by simply dragging the section from the tree onto an appropriate object in the model (i.e., a frame section onto a frame object). This drag-&-drop technique can significantly expedite model revisions.

- **Display Windows.** A display window shows the geometry of the model and may also include displays of properties, loading, analysis or design results. There is no limit on the number of windows that may be displayed.
- **Display Title Tab.** The display title tab is located at the top of the display window. The display title tab is highlighted when the associated display window is active. The text in the display title tab typically includes the type and location of the view in the associated display window.
- **Status Bar.** The status bar is located at the bottom of the main window. Text describing the current status of the program is displayed on the left side of the status bar.
- **Working Plane Drop-Down List.** This drop-down list appears in a 3-D View display window when a drawing command is active. Drawing objects in a 3-D view is restricted to the story (working plane) selected from this drop-down list, unless snaps are used.
- **Mouse Pointer Position Coordinates.** The mouse pointer position coordinates are displayed on the right-hand side of the status bar. 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.
- **Drawing & Selection Drop-Down List.** This drop-down list is on the right side of the status bar. The three options in the drop-down list are One Story, All Stories, and Similar Stories. With One Story, an object is created only at the story level on which it is drawn. With All Stories, an object drawn creates objects at all story levels in the model at the

same plan location. When doing an object select with All Stories, any object selected results in all other objects at the same plan location being selected at all story levels. With Similar Stories, an object drawn creates objects at all similar story levels in the model at the same plan location, and an object selected results in all other objects in the same plan location being selected at similar story levels.

- **Coordinate System Drop-Down List.** This drop-down list on the right side of the status bar allows the switching of coordinate/grid systems between the Global Coordinate System and user-defined Grid Systems. The selected system affects both the orientation of the model as well as the mouse pointer position coordinates.
- **Current Units.** The current units are displayed in a pop-up list located on the far right-hand side of the status bar. These units can be changed at any time during the model creation process.

## File Operations

File operations are used to start a new model, to bring in an existing model for display or modification, to save or export the current model for use in ETABS or another application, and to produce output. File operations are selected from the File menu.

New models can be started from scratch or from predefined templates supplied with the program.

## Edit

Editing is used to make changes to the model. Most editing operations work with one or more objects that were selected immediately before using the Edit command. Objects may be deleted, copied, pasted, moved, aligned, replicated, merged, and extruded using edit commands.

## View

View options, which affect how the structure displays, may be set for each display window and the setting may differ from window to window.

## Define

Define is used to create named entities that are not part of the geometry of the model. Those entities, accessed from the Define menu, include items such as material properties; frame, tendon, wall, and slab sections; and load patterns, cases and combinations. Definition of those entities does not require prior selection of an object, and some of those entities can be defined during the assignment operation using the Assign menu.

## Draw

Drawing is used to add new objects to the model or to modify one object at a time. Objects include beams, columns, slabs, decks, walls, links, tendons, and other joint, frame and shell objects. To draw, the program must be in **Draw Mode**, which is activated by clicking one of the draw buttons on the toolbar or using a Draw menu command.

In Draw Mode, the left mouse button is used to draw and edit objects, and the right mouse button is used to query the properties of those objects. Depending on the type of object to be drawn, a “Properties of Object” form appears that can be used to specify various structural properties, as well as the tower to which the object belongs when multiple towers are present. As frame objects are drawn, frame properties can be assigned simultaneously. Shell objects may be assigned floor properties, wall properties, or defined as openings when drawn. After an object has been drawn, the object may be selected and loads may be assigned to it, or existing assignments can be modified.

Draw Mode and Select Mode are mutually exclusive. No other operations can be performed when the program is in Draw Mode.

## Select

Selection is used to identify those objects to which the next operation will apply.

ETABS uses a “noun-verb” concept; that is, a selection is made and then an operation is performed. Certain editing, assigning, printing and displaying operations require prior selection of an object.

To select, the program must be in **Select Mode**, which is activated by clicking one of the select buttons on the toolbar. Alternatively, selecting any action from the Select menu puts the program into Select Mode. Many different types of selection are available, including selecting individual objects, drawing a window around objects, and selecting by property type.

In Select Mode, the left mouse button is used to select objects, and the right mouse button is used to query the properties of those objects.

Draw Mode and Select Mode are mutually exclusive.

## Assign

Certain assignments may be made when drawing an object, such as assigning a structural property when drawing a frame object. However, additional assignments, or changes to assignments, may be made to one or more objects that were selected immediately before using the Assign menu command. Assignment operations include properties, restraints, loads and group names.

## Analyze

After a complete structural model has been created using the preceding commands, the model can be analyzed to determine the resulting displacements, forces/stresses and reactions.

Before running an analysis, use the Set Active Degrees of Freedom command on the Analyze menu to control the active degrees of freedom and use Check Model to ensure that objects do not overlap and that objects are connected.

The first time an analysis is to be run, choose Set Load Cases to Run from the Analyze menu and select which cases are to be run. Once load cases have been selected, use Run Analysis from the Analyze menu, or click

the **Run Analysis** button on the toolbar to run the analysis. Any cases that have been run already do not need to be run again. If a load case that requires results from another case is chosen, the prerequisite case will be run first if it has not been already.

The program saves the data, then checks and analyzes the model. During the checking and analysis phases, messages from the analysis engine appear in the status bar. When the analysis is complete, a deformed shape will be displayed.

No other ETABS operations may be performed while the analysis is proceeding. However, other Windows applications can be run during this time.

## Display

The Display menu commands are used to view the model and the results of the analysis. Graphical and tabular displays are available in this program. Display items may be chosen from the Display menu or accessed using toolbar buttons.

- **Graphical Displays** – Different types of graphical display may be selected for each display window. Each window may also have its own view orientation and display options. Undeformed geometry, loads, and analysis results can all be displayed. Details of the displayed results can be obtained by clicking on an object with the right mouse button.
- **Tabular Displays** – Tabular information can be displayed for the model by choosing the Tables tab on the Model Explorer. Choose a table to be viewed and then right click. If objects are selected prior to using the commands, certain tables will only be available for the selected objects. If no objects are selected, the tables produced are for the entire model. Tabular data can also be printed using the **Create Report** commands available on the File menu.

## Design

After an analysis has been completed, frames, composite beams and columns, joists, shear walls, and slabs can be designed with respect to design code requirements. Design may be performed for the given design combinations by choosing the appropriate Design menu command. Before designing, verify the selected design codes and preferences using the appropriate **View/Revise Preferences** command located on the design menus.

Graphical displays of design parameters are available. Tabular design information can also be printed using commands from the File menu.

## Tools

The Tools menu provides access to user or third-party developed plugins that allow for customization of the program.

## Options

The Options menu provides various commands that affect the overall operation of ETABS. Display units, colors, the graphics mode, tolerances, and whether multiple towers are allowed can be specified here.

## Help

The program Help is available from this menu. Documentation and verification manuals in PDF format are accessed through the Help menu as well. A link to the CSI website, as well as information about the currently installed version of ETABS and its associated license, can be found [here](#).

# Basic Modes, Drawing Tools, Mouse Pointers

## Objective
















This chapter briefly describes the two modes of user operation for the program, identifies the drawing tools, and describes how the appearance of the mouse pointer changes for various operations.

## Select or Draw
















The two distinct modes in this program are the *select* mode and the *draw* mode.

- The select mode allows objects to be selected and is used for editing operations, making assignments to objects, and viewing or printing results. By default, the program is in select mode. Chapter 6 describes the various methods for selecting points, lines, and areas in a model.
- The draw mode allows objects to be drawn.


The draw mode automatically enables when one of the following sub-menu options from the Draw menu is selected or the corresponding button on the toolbar is clicked. Note that the views in parenthesis (Plan, Elev, 3D) after the command name indicate when the button will be active; for example, the Draw Beam/Column/Brace command/button can be used in the Plan, Elevation or 3D views, but the Draw Walls command/button can be used only in Plan view. The names of the commands are assumed to explain the actions that will be accomplished. More information about the Draw tools is available by searching for “draw menu” using the **Help menu > ETABS Help** command.

- Draw Joint Objects 
- Draw Beam/Column/Brace Objects 
  -  Draw Beam/Column/Brace (Plan, Elev, 3D)
  -  Quick Draw Beams/Columns (Plan, Elev, 3D)
  -  Quick Draw Columns (Plan)
  -  Quick Draw Secondary Beams (Plan)
  -  Quick Draw Braces (Elev)
- Draw Floor/Wall Objects 
  -  Draw Floor/Wall (Plan, Elev, 3D)
  -  Draw Rectangular Floor/Wall (Plan, Elev)
  -  Quick Draw Floor/Wall (Plan, Elev)
  -  Quick Draw Area Around Point (Plan)
  -  Draw Walls (Plan)
  -  Quick Draw Walls (Plan)
  -  Draw Wall Openings (Plan, Elev, 3D)





- Draw Links 
- Draw Tendons 
- Draw Slab Rebar 
- Draw Design Strips 
- Draw Grids 
- Draw Dimension Lines 
- Draw Reference Points 
- Draw Reference Planes 
- Draw Section Cut 
- Draw Developed Elevation Definition 
- Draw Wall Stacks (Plan, Elev, 3D) 
- Auto Draw Cladding 
- Draw Line Strain Gauge 
- Draw Quad Strain Gauge 
- Snap Options 

The draw mode remains enabled until one of the following actions is taken to return to the select mode:

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

**The mouse pointer indicates which mode is enabled.** The appearance/properties of the mouse pointer are defined in the Windows Control Panel. The mouse pointer properties are Normal Select Pointer and Alternate Select pointer.

In select mode, the pointer is the Normal Select Pointer. If the default settings are being used, the mouse pointer will look like this .

In draw mode, the mouse pointer is the Alternate Select pointer. If the default settings are being used, the mouse pointer will look like this .

Note that while in draw mode, if the mouse pointer is moved over the toolbar buttons or the menus, the pointer temporarily changes to the selection pointer. If during this time one of the menus or toolbar buttons is not clicked, the mouse pointer reverts to the draw mode pointer when it is moved back into the display window.


Other mouse properties/appearances are used for various actions in the program, including Help Select, Busy, Text Select, Vertical Resize, Horizontal Resize, and Move. The appearance of the mouse pointers for those actions depends on the mouse pointer properties you specify.

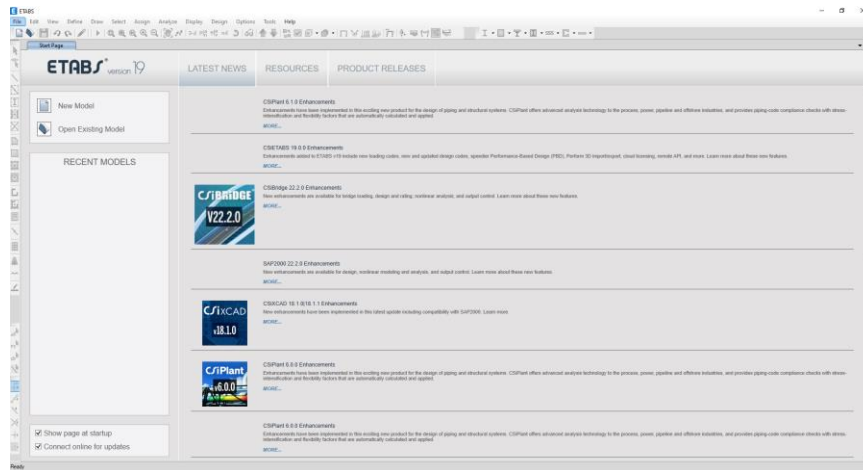
# Begin a Model

## Objective

This chapter describes how to begin a model by creating the basic grid system. Structural objects are placed relative to the grid system.

## Create the Basic Grid System

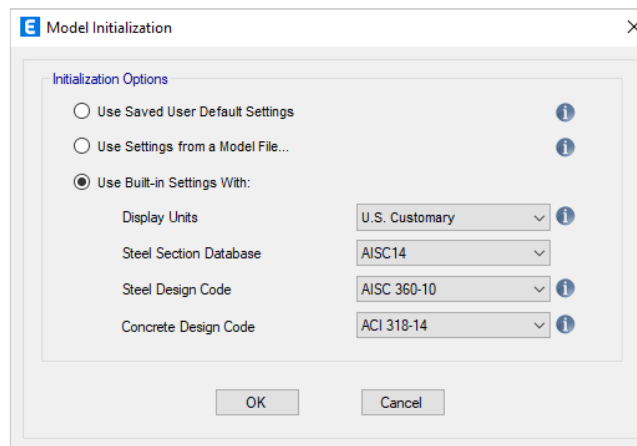
Begin creating the grid system by starting the program. The Start Page will be displayed as shown in Figure 4-1. If the program is already running with a model displayed, you can start a new model by clicking the **File menu > New Model** command or the **New Model** button .



**Figure 4-1 Start Page**


The Start Page is divided into three regions: the New Model and Open Existing Model buttons; the Recent Models area; and the Latest News/Resources/Product Releases area. The Recent Models area contains iconic buttons representing models recently created in ETABS. The Latest News/Resources/Product Releases area displays helpful links as well as recent news about CSI.

Click the **New Model** button on the Start Page to display the Model Initialization form shown in Figure 4-2.



**Figure 4-2 Model Initialization form**

There are three options on the Model Initialization form for setting the initial units, preferences, properties and definitions: User Default Settings, which can be saved using the **Options menu > Save User Default Settings** command; Settings from a Model File; or Built-in Settings with additional unit, section and code selections.

On the Model Initialization form select the Use Built-in Settings With option and then choose either *U.S. Customary*, *Metric SI* or *Metric MKS* from the Display Units drop-down list - this selection will set the defaults for the input and display units. These units determine what units are associated with each piece of input data, and what units are used to display model output. These units may be inconsistent for different items, i.e., moment diagrams may be displayed in kip-ft units while shear stresses are in lb/square inch. To review the display units hold the mouse cursor over the information icon . To change the default units, use the **Options menu > Display Units** command or click on the **Units** button located in the lower right-hand corner of the screen.

Also on the Model Initialization form are drop-down lists for selecting the steel section database, the steel design code, and the concrete design code to use when creating and designing the model.

Click the **OK** button on the Model Initialization form to display the New Model Quick Templates form shown in Figure 4-3. The New Model Quick Templates form is used to specify horizontal grid line spacing, story data, and template data. The form contains a blank button, a grid only option, four concrete building templates (Flat Slab, Flat Slab with Perimeter Beams, Waffle Slab, Two Way or Ribbed Slab), and two steel building templates (Steel Deck, Staggered Truss). Template models provide a quick, easy way of starting a model. They automatically add structural objects with appropriate properties to the model. We highly recommend that you start your models using templates whenever possible.

Figure 4-3 New Model Quick Templates form

## Grid Dimensions (Plan) - Define a Grid System

Use the Grid Dimensions (Plan) area of the form to define a grid line system. Select from two options for defining the grid line system:

- **Uniform Grid Spacing.** Specify the number of grid lines in the X and Y directions and a uniform spacing for those lines. Note that the uniform spacing in the X and Y directions can be different. This option defines a grid system for the global coordinate system only. Click the **Grid Labels** button to control how the grids are labeled. If subsequently necessary, edit the information using the **Edit menu > Edit Stories and Grid Systems** command. For more information, search for “edit grid data” using the **Help menu > ETABS Help** command. Note that the default global coordinate/grid system is a Cartesian (rectangular) coordinate system.

- **Custom Grid Spacing.** Define nonuniformly spaced grid lines in the X and Y directions for the global coordinate system. After choosing this option, click the **Edit Grid Data** button to edit the grid system. For more information, search for “grid labeling” using the **Help menu > ETABS Help** command.

The reasons for defining a grid system for the model include the following:

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

### **Story Dimensions - Define Story Data**

Use the Story Dimensions area of the form to define the number and height of stories. Select from two options for defining the story data:

- **Simple Story Data:** Enter values in the edit boxes to define the number of stories and a typical story height that is used for all story levels except for the bottom story, which is specified separately. The program provides default names for each story level (for example, Story1, Story2 and so on) and assumptions for story level similarity.
- **Custom Story Data:** After choosing this option, click the **Edit Story Data** button to access the Story Data form. Enter values in the Story Data form to define your own story names, story levels of non-uniform height and customized story similarity. Story level "similarity" can be significant, e.g., when Story2 is a Master Story, and Story1 is similar to Story2, an object drawn on Story2 typically appears in the same

plan location on Story1. The splice data identifies which stories contain steel column splices and the height of the splices - splice data is not applicable to concrete columns.

The Story Data form also appears when the **Edit menu > Edit Stories and Grid Systems** command is used followed by the **Modify/Show Story Data** button on the Edit Story and Grid System Data form. For more information about the Story Data form, refer to the Editing chapter of this manual. For more information about story level similarity, search for “similar stories drop-down list” using the **Help menu > ETABS Help** command. Story level similarity can also be significant to composite beam and steel joist design.



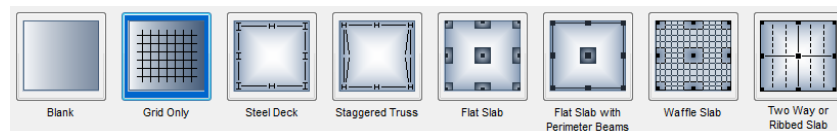
# Create the Structural Model

## Objective

This chapter describes how to create the structural model. It is assumed that you have read Chapter 4 *Begin a Model* or understand how to begin an ETABS model by defining a grid system.

## Add Structural Objects Using Templates

Use one of the six built-in templates shown on the New Model Quick Templates form to add structural objects to your model. In many cases it is the simplest, most convenient and quickest way to start a model. The New Model templates are shown below:




Note that the templates consist of two for steel buildings and four for concrete buildings, as well as a button for creating grids only and a button for starting a blank model, both of which add no structural objects to the model. Choose any of the templates by left clicking its associated button. When one of the template buttons is chosen, the Structural Geometry and Properties form will appear for that template, as shown in Figure 5-1. The Structural Geometry and Properties form typically contains areas for specifying structure data and loads.

**Note:** This form will not display if the Grid Only or Blank buttons are chosen since no structural objects are defined.

**Figure 5-1 Structural Geometry and Properties**

Once all structure and load data have been entered, click the **OK** button to close the form and return to the New Model Quick Templates form.

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

Click the **OK** button on the New Model Quick Templates form and the model appears on screen in the main ETABS window with two view windows tiled vertically, a Plan View on the left and a 3-D View on the right, as shown in Figure 5-2. The number of view windows can be changed using the **Windows List** button .

Note that the Plan View is active in Figure 5-2. When the window is active, the display title tab is highlighted. Set a view active by clicking anywhere in the view window.

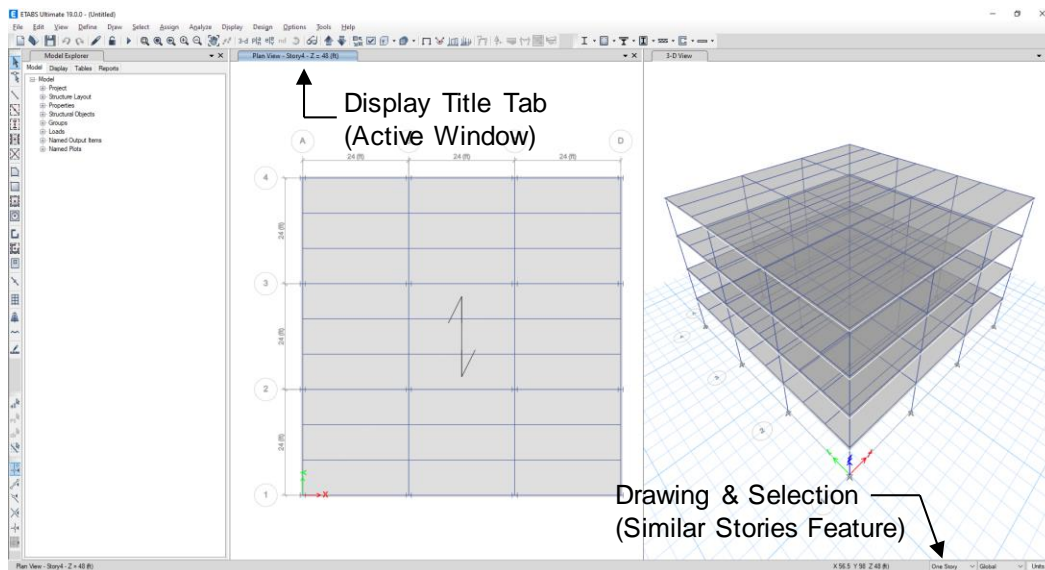



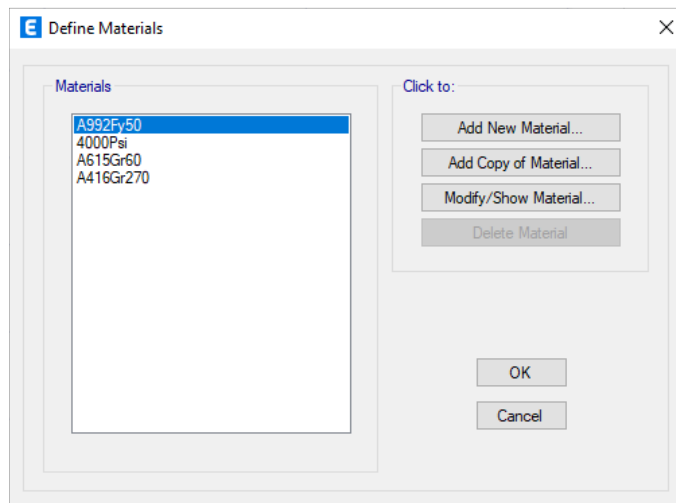
Figure 5-2 The ETABS main window

## Define Properties

Template generated models typically rely on program defined material and section properties. The following sections will show how to define additional properties or review program defaults.

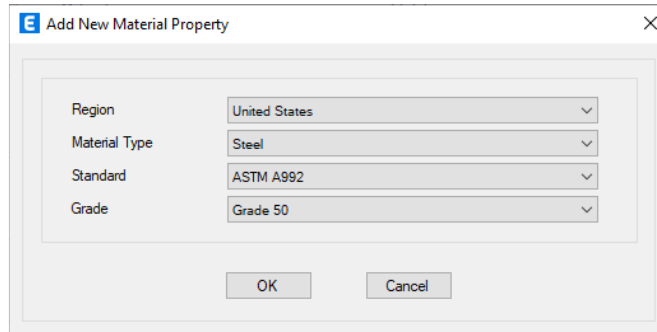
### Material Properties

Click the **Define menu > Material Properties** command to display the Define Materials form shown in Figure 5-3, or under the Model tab on the Model Explorer expand the  Properties branch and then the Materials branch to see a list of the defined material properties (a right-click on the Materials branch will display a context sensitive menu).




**Figure 5-3 Define Materials form**

The Define Materials form allows for the both the review of existing materials, as well as the definition of new properties. To add a new material, click the **Add New Material** button on the Define Materials form. When the Add New Material Property form appears as shown in Figure 5-4, select a material from the Material Type drop-down list and then a Standard and Grade from their respective drop-down lists.

The image shows a software dialog box titled "Add New Material Property". It contains four dropdown menus: "Region" set to "United States", "Material Type" set to "Steel", "Standard" set to "ASTM A992", and "Grade" set to "Grade 50". At the bottom are "OK" and "Cancel" buttons.


**Figure 5-4 Add New Material Property form**

Once selections have been made on the Add New Material Property form, click the **OK** button to display the Material Property Data form where data for the new material may be reviewed and edited. Click the **OK** button on the Material Property Data form to return to the Define Materials form, where additional materials may be defined or reviewed. Click the **OK** button on the Define Materials form when finished with materials.

Click the **File menu > Save** command, or the **Save** button, , to save the model.

## Frame Sections

Click the **Define menu > Section Properties > Frame Sections** command, which will display the Frame Properties form. The Frame Properties form allows for the definition of new sections as well as the review of existing sections. To make steel frame sections from property files available click the **Import New Properties** button, or to add user defined sections click the **Add New Property** button, both of which will display the Frame Property Shape Type form shown in Figure 5-5.

On the Frame Property Shape Type form, click on the **I/Wide Flange Section**  button under Steel in the Frequently Used Shape Types area, or select *Steel I/Wide Flange* from the Section Shape drop-down list in the Shape Type area and click the **OK** button. The Frame Section Property Import Data form shown in Figure 5-6 displays when importing.

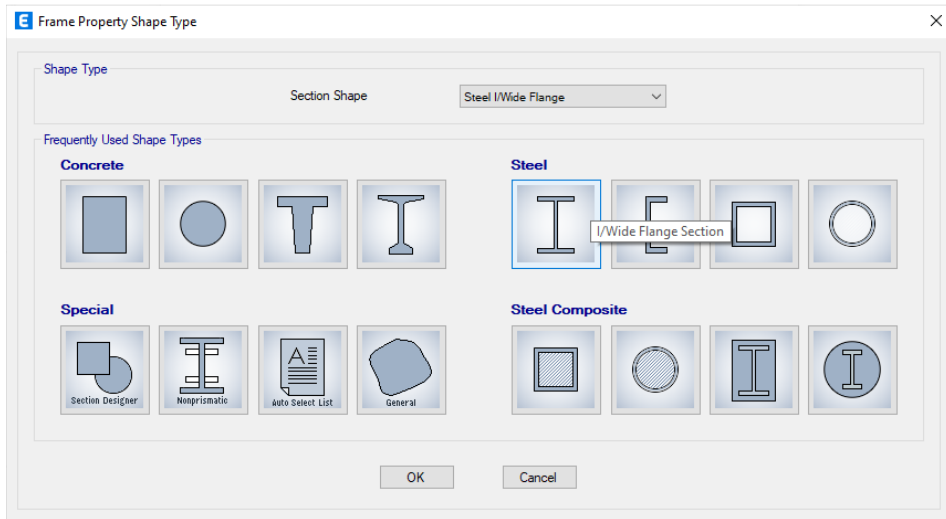


Figure 5-5 Frame Property Shape Type form

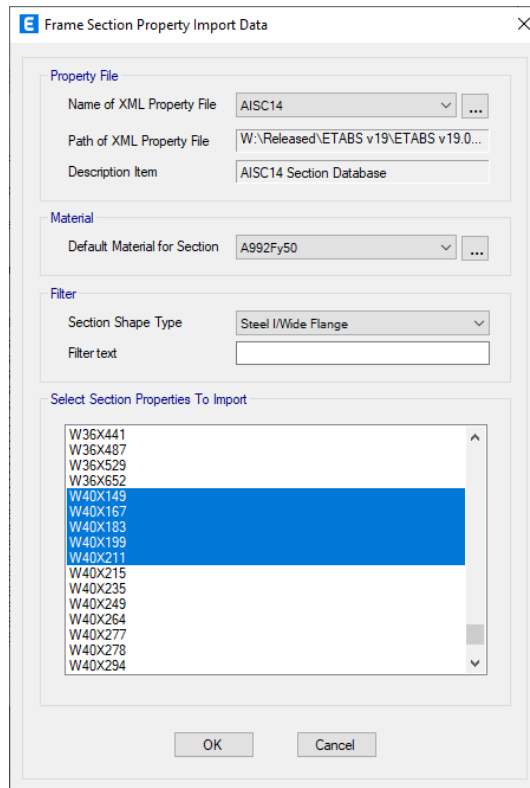
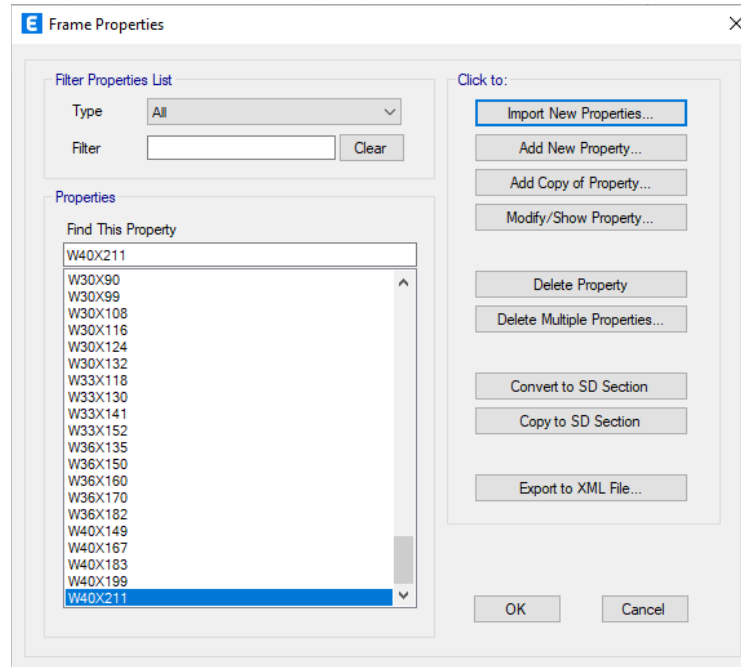


Figure 5-6 Frame Section Property Import Data form


The Frame Section Property Import Data form lists the available section properties for import into the model. Select the sections to be imported from the list (e.g., W40X149 thru W40X211) using standard Windows select techniques, i.e., holding the Shift key while selecting. Click the **OK** button to return to the Frame Properties form shown in Figure 5-7.



**Figure 5-7 Frame Properties form**

The Frame Properties form should now list the properties just selected on the Frame Section Property Import Data form. Additional sections may be added to the Properties list by using the **Import New Properties** button again, or highlighted sections may be reviewed by using the **Modify/Show Property** button.

Click the **OK** button on the Frame Properties form when finished with section definitions.

Click the **File menu > Save** command, or the **Save** button, , to save the model.

## Auto Select Section List

ETABS's Auto Select Section List feature helps to reduce the time required to develop the model as well as to enhance the design process.

An auto select section list is simply a list of sections; for example, W18X35, W18X40, W21X44, W21X50 and W24X55. Auto select section lists can be assigned to frame members. When an auto select section list is assigned to a frame member, the program can automatically select the most economical, adequate section from the auto select section list when it is designing the member.

The program has several built-in auto select section lists. However, the user can also develop a tailored list using the following steps:

1. Click the **Define menu > Section Properties > Frame Sections** command, which will display the Frame Properties form shown in Figure 5-8. The previous section explains how to import frame properties into the Properties list.

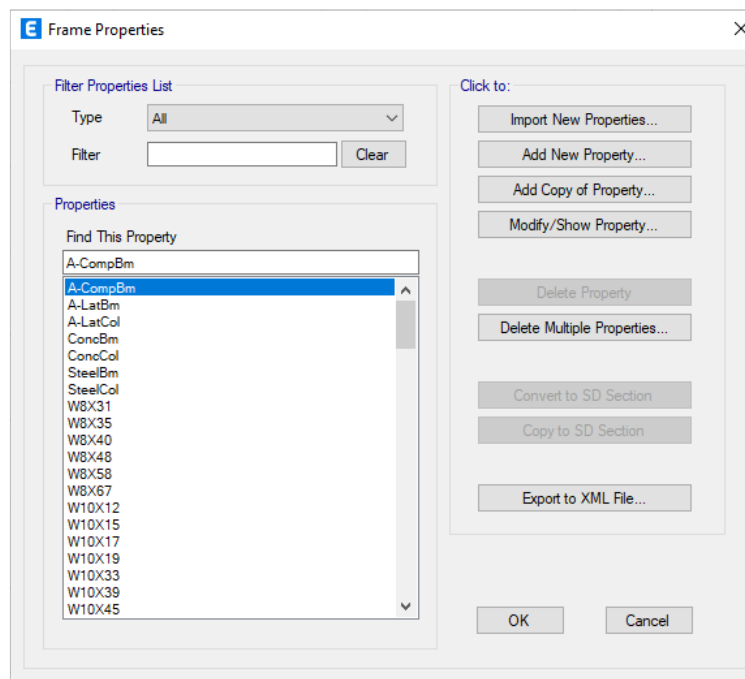



Figure 5-8 Frame Properties form



2. Click the **Add New Property** button in the Click to area of the Frame Properties form. The Frame Property Shape Type form shown in Figure 5-9 will appear.
3. Click on the **Autoselect Section List** button  under Special in the Frequently Used Shape Types area, or select *Auto Select* from the Section Shape drop-down list in the Shape Type area and then click the **OK** button. The Frame Section Property Data form shown in Figure 5-10 displays.

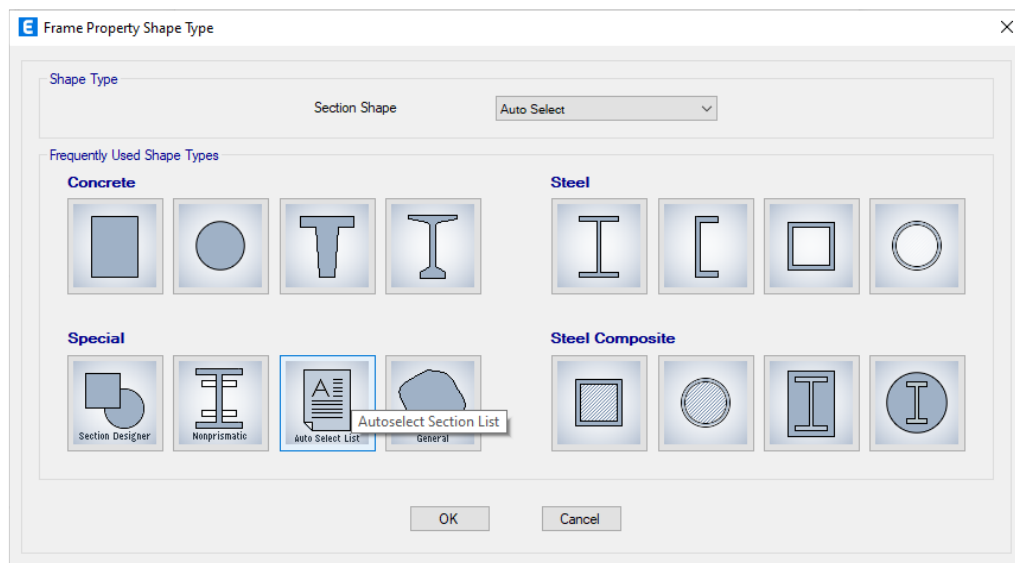


Figure 5-9 Frame Property Shape Type form

4. Type a name for the list in the Property Name edit box. Any name can be used. For the purposes of this description, the new Auto Select Section List is **AUTOLATBM**.
5. Scroll down the list of sections in the Choose Sections in Auto Select List area to find the beams to be included in the list. Click once on them to highlight them. Note that the standard Windows methods for selecting items in a list can be used (e.g., clicking on a section and

then pressing the shift key on the keyboard before selecting another section will highlight all sections between the two selected items).

6. Click the **Add** button to add the selected beams to the Auto Select List on the right side of the form.


**Figure 5-10 Frame Section Property Data form**

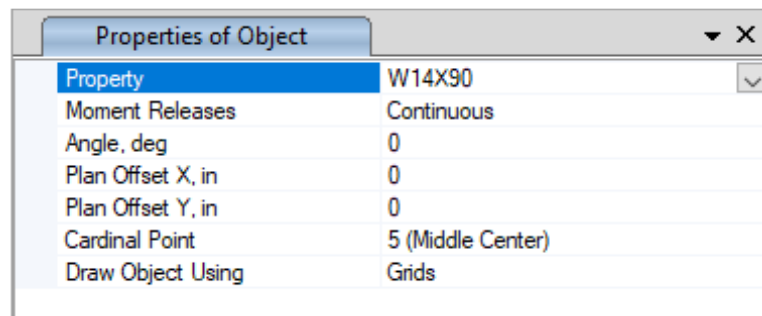
7. Click the **OK** button and then click the **OK** button in the Frame Properties form to accept the definition of a new Auto Select Section List named AUTOLATBM.

## Add Structural Objects Manually

Previously a model was generated using a template, but objects, such as columns, beams, floors, and walls, also can be drawn manually as described in the sections that follow.

### Draw Columns

Make sure that the Plan View is active. Click the **Quick Draw Columns** button, , or use the **Draw menu > Draw Beam/Column/Brace Objects > Quick Draw Columns** command. The Properties of Object box for columns shown in Figure 5-11 will display docked in the lower left-hand corner of the display. Hold the left mouse button down on the Properties of Object tab to move the box elsewhere in the display, or to dock it using the docking arrows.




**Figure 5-11 Properties of Object Box for Columns**


The Properties of Object box provides various definition parameters and drawing controls. These items differ depending on the drawing command selected. Review the parameters and controls shown in this box before drawing the column to ensure that they are what they should be. Change any entry in the box by clicking on it and making a new selection from the drop-down list or entering new information into the edit box, as appropriate.

After checking the parameters in the Properties of Object box, left click once in the Plan View at the *intersection of the grid lines* where you want

the column. An I-shaped column should appear at that point in the Plan View. Continue in this manner to place other columns.

Alternatively, draw the remaining columns in one action by "windowing" around the grid intersections. To "window," click the left mouse button above and to the left of the first grid intersection where a column is to be placed and then, while holding the left mouse button down, drag the mouse until it is below and to the right of the last grid intersection where a column is to be placed. A selection box similar to that shown in Figure 5-12 should expand around the grid line intersections as the mouse is dragged across the model. Release the left mouse button and the program will draw the column objects at the grid line intersections within the boundaries of the selection box.

To leave the Draw mode, click the **Select Object** button, .

It is a good idea to save your models often. Click the **File menu > Save** command, or the **Save** button, , to save the model.

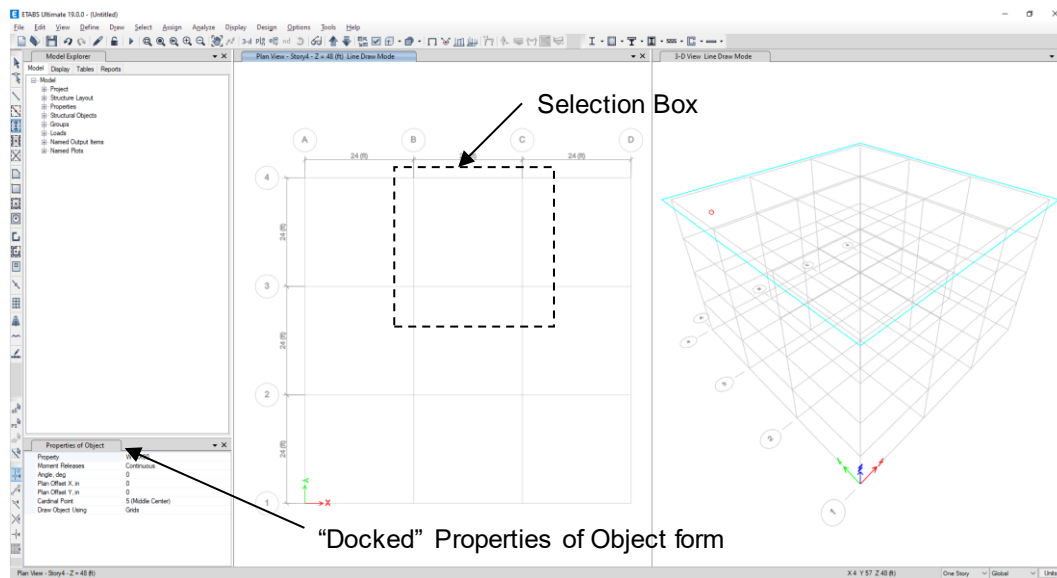

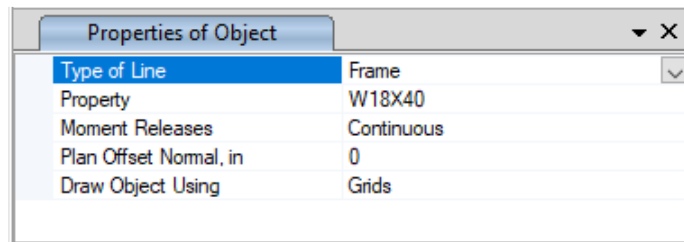


Figure 5-12 Drawing Column Objects in a Windowed Region

## Draw Beams

Make sure that the Plan View is active. Click the **Quick Draw Beams/Columns** button,  or the **Draw menu > Draw Beam/Column/Brace Objects > Quick Draw Beams/Columns** command. The Properties of Object box for frames shown in Figure 5-13 will display docked in the lower left-hand corner.




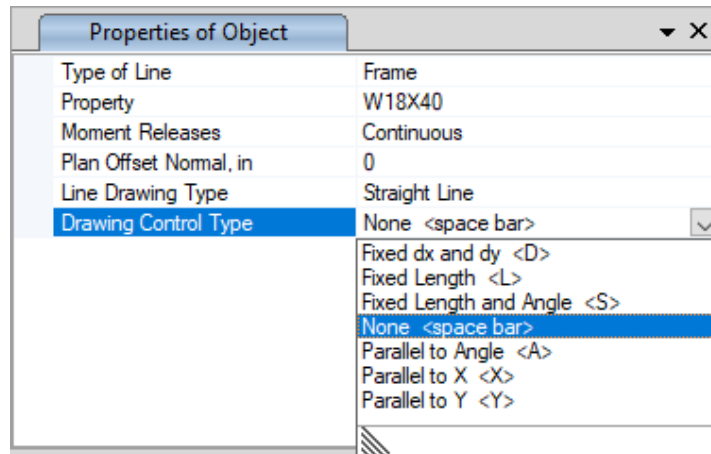
**Figure 5-13 Properties of Object Box for Frames**

As explained previously, the Properties of Object box provides various definition parameters. Change any entry in the box by clicking on it and making a new selection from the drop-down list or entering new information into the edit box, as appropriate.

After checking the parameters in the Properties of Object box, left click once in the Plan View on a *grid line* where a beam is to be placed. A beam is drawn along the selected grid line. Continue in this manner to place other beams.

Alternatively, draw the remaining beams in one action by windowing around the grid intersections. Windowing is explained in the previous section.

To draw beams not located on a grid line, click the **Draw Beam/Column/Brace** button,  or the **Draw menu > Draw Beam/Column/Brace Objects > Draw Beam/Column/Brace** command. The Properties of Object box for frames shown in Figure 5-14 will display docked in the lower left-hand corner. This form is similar to that shown in Figure 5-13 with the addition of an option for constraining how the frame object is to be drawn, i.e., Drawing Control Type.

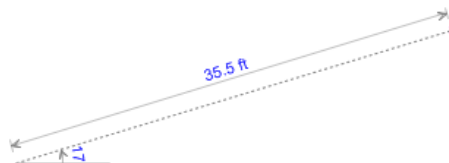


**Figure 5-14 Properties of Object Box for Frames**


The Drawing Control Type can constrain the line to be a fixed length, or parallel to an angle, or both, or parallel to coordinate axes.

After checking the parameters in the Properties of Object box, left click once in the Plan View to indicate the starting location of the beam. Select an option from the Drawing Control Type drop-down list if some type of drawing constraint is desired, and then left click to indicate the end joint of the beam. The program will start another frame object at the location of the just drawn beam's end joint unless the right button of the mouse is clicked to stop drawing.


Another aid when drawing objects is the Draw Measurement Tool shown in Figure 5-15. This tool automatically displays when in the drawing mode after the starting joint of the object is drawn. This tool displays the length and angle orientation of the frame member or edge.



**Figure 5-15 Draw Measurement Tool**

To leave the Draw mode, click the **Select Object** button, .


## Draw Secondary (Infill) Beams

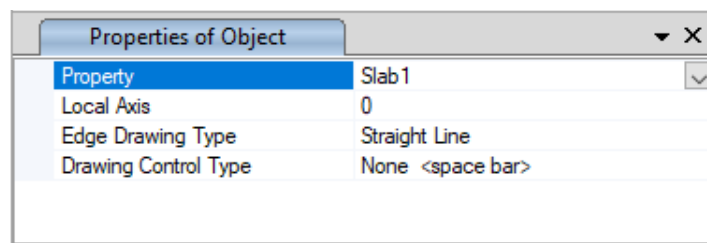
Add secondary or "infill" beams by clicking the **Quick Draw Secondary Beams** button,  or the **Draw menu > Draw Beam/Column/Brace Objects > Quick Draw Secondary Beams** command. Similar to the other drawing operations, a Properties of Object box will display docked in the lower left-hand corner that provides the opportunity to define the parameters for the secondary beams.

To place the secondary beams, left click once in the *bay* bounded by grid lines where the secondary beams are to be placed. Like the columns and the primary beams, secondary beams can be drawn by windowing over the appropriate bays. Note the *Approx. Orientation* parameter to set the span direction.

Click the **File menu > Save** command, or the **Save** button, , to save the model.

## Draw the Floor


Make sure that the Plan View is active. Click the **Draw Floor/Wall** button, , or select the **Draw menu > Draw Floor/Wall Objects > Draw Floor/Wall** command. The Properties of Object box for shells shown in Figure 5-16 will appear docked in the lower left-hand corner.




**Figure 5-16 Properties of Object Box for Shells**


Similar to columns and beams, this Properties of Object box provides the opportunity to check and change the parameters for the shell. Change any entry in the box by clicking on it and making a new selection from the

drop-down list or entering new information into the edit box, as appropriate.

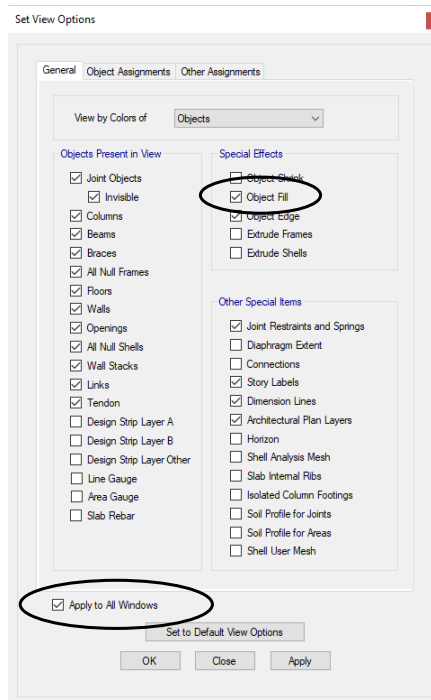
After checking the parameters in the Properties of Object box, check that the **Snap to Grid Intersections and Points** command is active. This will assist in accurately drawing the area object. This command is active when its associated button  is depressed. Alternatively, use the **Draw menu > Snap Options** command to ensure that this command is active. By default, this command is active.

Left click once at a column to begin the floor/area object at that column. Then, moving around the perimeter of the floor object, click once at other column intersections to draw the outline of the building. Press the Enter key on your keyboard to complete the floor.

If you have made a mistake while drawing this object, click the **Select Object** button, , to change the program from Draw mode to Select mode. Then click the **Edit menu > Undo Shell Add** command.

To switch the fill on or off for the floor addition, click the **Set Display Options** button . Once the Set View Options form appears, check or


**Figure 5-17 Set View Options form**




The image shows the 'Set View Options' dialog box with three tabs: 'General', 'Object Assignments', and 'Other Assignments'. The 'General' tab is active. It features a 'View by Colors of' dropdown menu set to 'Objects'. Below this, there are two main sections: 'Objects Present in View' and 'Special Effects'. The 'Objects Present in View' section contains a list of checkboxes for various objects, including 'Joint Objects', 'Invisible', 'Columns', 'Beams', 'Braces', 'All Null Frames', 'Floors', 'Walls', 'Openings', 'All Null Shells', 'Wall Stacks', 'Links', 'Tendon', 'Design Strip Layer A', 'Design Strip Layer B', 'Design Strip Layer Other', 'Line Gauge', 'Area Gauge', and 'Slab Rebar'. The 'Special Effects' section contains checkboxes for 'Object Shell', 'Object Fill', 'Object Edge', 'Extrude Frames', and 'Extrude Shells'. The 'Object Fill' checkbox is circled. At the bottom of the dialog, there is a checkbox for 'Apply to All Windows', which is also circled. Below this checkbox is a 'Set to Default View Options' button. At the very bottom are 'OK', 'Close', and 'Apply' buttons.

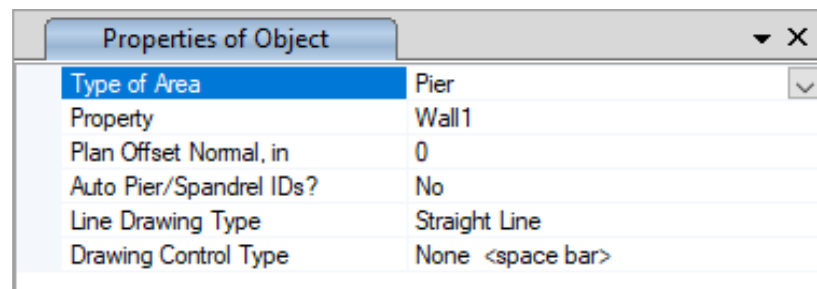


uncheck the Object Fill check box and the Apply to All Windows check box on the General tab, as shown in Figure 5-17. Click the **OK** button.

Click the **File menu > Save** command, or the **Save** button, , to save the model.

## Draw Walls


Make sure that the Plan View is active. Click the **Draw Walls** button, , or select the **Draw menu > Draw Floor/Wall Objects > Draw Walls** command. The Properties of Object box for walls shown in Figure 5-18 will appear docked in the lower left-hand corner.




**Figure 5-18 Properties of Object Box for Walls**

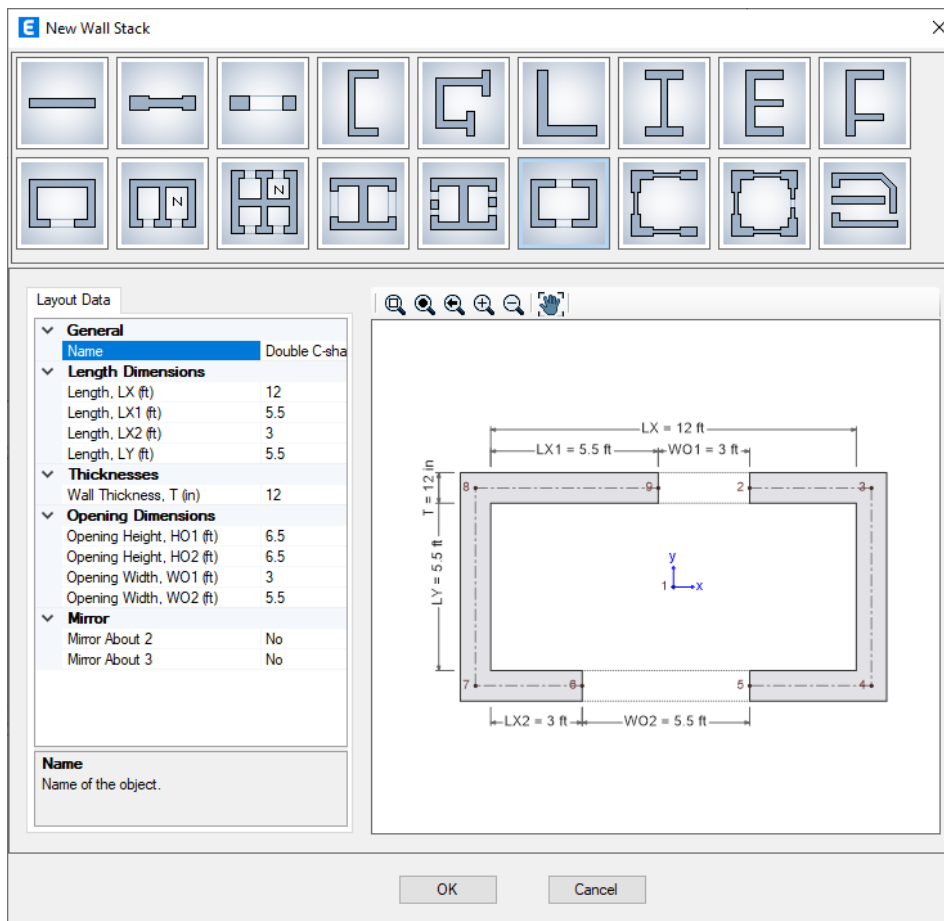
Change any entry in the Properties of Object box by clicking on it and making a new selection from the drop-down list or entering new information into the edit box, as appropriate.

To place walls, left click once at a point to begin the wall object at that point. Then, move to the end of the wall segment and left click again. Additional wall segments may be drawn by simply moving to a new point and clicking. Press the Enter key on your keyboard to complete the wall.

Click the **File menu > Save** command, or the **Save** button, , to save the model.

## Draw Wall Stacks

Make sure that the Plan View is active. Click the **Draw Wall Stacks** button, , or the **Draw menu > Draw Wall Stacks** command. The New Wall Stack form shown in Figure 5-19 will appear.



**New Wall Stack**

**Layout Data**

General	
Name	Double C-sha
Length Dimensions	
Length, LX (ft)	12
Length, LX1 (ft)	5.5
Length, LX2 (ft)	3
Length, LY (ft)	5.5
Thicknesses	
Wall Thickness, T (in)	12
Opening Dimensions	
Opening Height, HO1 (ft)	6.5
Opening Height, HO2 (ft)	6.5
Opening Width, WO1 (ft)	3
Opening Width, WO2 (ft)	5.5
Mirror	
Mirror About 2	No
Mirror About 3	No

**Name**  
Name of the object.


OK Cancel


Figure 5-19 New Wall Stack form

Select any of the predefined wall stacks by clicking on the representative icon. The lengths and thicknesses of the wall segments may be altered by entering changes into the edit boxes on the Layout Data tab. Once all wall stack parameters have been reviewed on the Layout Data tab, click


the **OK** button. The Properties of Object box for wall stacks will appear docked in the lower left-hand corner

Verify that the angle and range of stories for the wall stack are correct in the Properties of Object form, and then left click once in the Plan View where the wall stack is to be placed. A wall stack is drawn at that location for the number of stories specified. Continue in this manner to place other wall stacks.

To leave the Draw mode, click the **Select Object** button, .

Click the **File menu > Save** command, or the **Save** button, , to save the model.

## Draw Tendons

Make sure that the Plan View is active. Click the **Draw Tendons** button, , or the **Draw menu > Draw Tendons** command. The Properties of Object box for tendons shown in Figure 5-20 will appear docked in the lower left-hand corner.

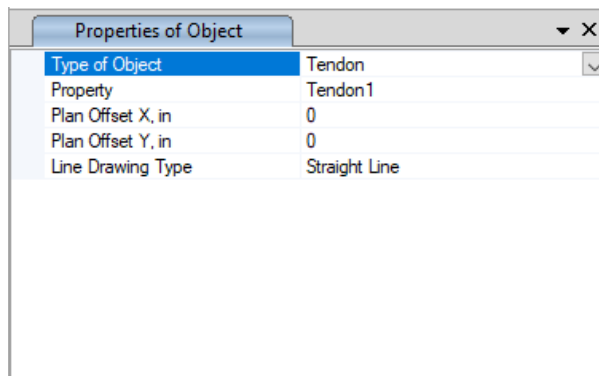



Figure 5-20 Properties of Object Box for Tendons

Change any entry in the Properties of Object box by clicking on it and making a new selection from the drop-down list or entering new information into the edit box, as appropriate.

To place tendons, left click once at a point to begin the tendon object at that point. Then, move to the end of the tendon segment and left click

again. Additional tendon segments may be drawn by simply moving to a new point and clicking. Press the Enter key on your keyboard to complete the tendon.

To expedite the addition of banded and/or distributed tendons to a slab, rather than draw individual tendons, it may be more efficient to use the **Edit menu > Add/Edit Design Strips > Add Design Strips** command to first layout design strips, and then use the **Edit menu > Add/Edit Tendons > Add Tendons in Strips** command to rapidly place multiple tendons within selected strips.

Click the **File menu > Save** command, or the **Save** button, , to save the model.

# Select Structural Objects

## Objective

This chapter describes how to select objects in the model.

## Selecting

Selecting is used to identify existing objects to which the next operation will apply. Operations that require prior selection include certain Editing, Assignment, Design, Display, and Output operations.

## Graphical Selection Options

The program has a number of techniques for graphically selecting objects:

- **Left click:** Left click on an object to select it. If multiple objects are present in the same location, one on top of the other, hold down the



Ctrl key on the keyboard and click the left mouse button on the objects. Use the form that displays to specify which object to select.

- **Window or "Windowing":** Drag a window from *left to right* to select all objects that are fully enclosed in the window. Drag a window from *right to left* to select all objects that are fully or partially enclosed in the window. To draw a window, first position the mouse pointer beyond the limits of the object; for example, above and to the left of the object(s) to be selected. Then depress and hold down the left mouse button. While keeping the left button depressed, drag the mouse to a position below and to the right of the object(s) to be selected. Release the left mouse button to complete the selection. Note the following about window selection:
  - ✓ As the mouse is dragged, a "rubber band window" appears. The rubber band window is a dashed rectangle that changes shape as the mouse is dragged. One corner of the rubber band window is at the point where the left mouse button was first depressed. The diagonally opposite corner of the rubber band window is at the current mouse pointer position. When dragging the mouse from left to right, any visible object that is completely inside the rubber band window is selected when the left mouse button is released. When dragging the mouse from right to left, any visible object that the window crosses or encloses is selected.
  - ✓ As long as the mouse pointer is beyond the limits of the object(s) to be selected, the window can start at any point.

**Note about Window Selections in Plan View:** When selecting by window in a plan view, the objects selected will be determined by the setting in the Drawing & Selection drop-down list. To select only the objects at the plan level displayed (which include the columns in the story below), the drop-down list should be set to One Story. When set to Similar Stories or All Stories, selecting in plan view may result in objects at other levels being selected, even though only one plan level is displayed.

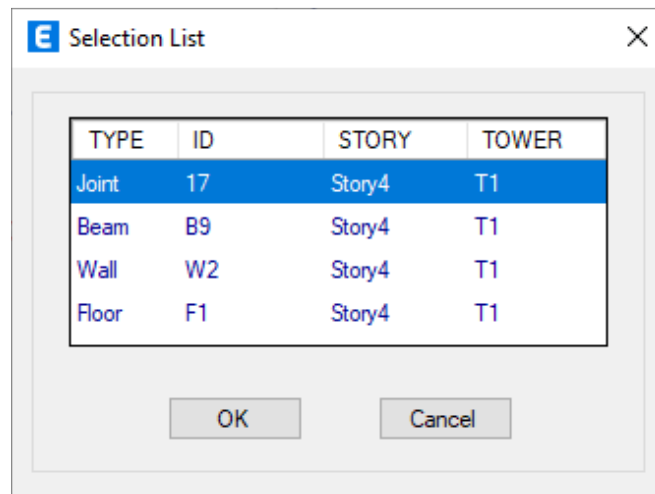
- **Poly:** Draw a polygon with any number of sides to select all objects that are fully enclosed in the polygon. To use this selection method, click the **Select menu > Select > Poly** command. Then position the

mouse pointer outside the object(s) to be selected, left click to start the polygon and then left click at each of the polygon's vertices. Hit the Enter key on the keyboard to complete the selection polygon. After using this method to make a selection, the program defaults to the window selection mode.

- **Intersecting Poly:** Draw a polygon with any number of sides to select all objects that are fully or partially enclosed in the polygon. To use this selection method, click the **Select menu > Select > Intersecting Poly** command. Then position the mouse pointer outside the object(s) to be selected, left click to start the polygon and then left click at each of the polygon's vertices. Hit the Enter key on the keyboard to complete the selection polygon. After using this method to make a selection, the program defaults to the window selection mode.
- **Intersecting Line:** Draw a line through one or more objects to select them. To use this selection method, click the **Select menu > Select > Intersecting Line** command or the **Select using Intersecting Line** button, . Then position the mouse pointer to one side of the object(s) to be selected and click the left mouse button. Drag the mouse across the object(s) to be selected and click the left mouse button followed by the Enter key on the keyboard to complete the selection. Note the following about the intersecting line selection method:
  - ✓ As the mouse is dragged, a "rubber band line" appears. The rubber band line is a dashed line that changes length and orientation as the mouse is dragged. It extends from the point where the left mouse button is first clicked to the current mouse pointer position. Any visible object that is intersected (crossed) by the rubber band line is selected when the Enter key is pressed.
  - ✓ After using this method to make a selection, the program defaults to the window selection mode. Thus, the **Select menu > Select > Intersecting Line** command must be selected or the **Select using Intersecting Line** button  must be clicked each time this selection method is used.

- **Control and Left Click:** Hold down the Ctrl key on the keyboard and left click once on a joint, frame or shell object. A Selection List form like the one shown in Figure 6-1 pops up identifying the objects that exist at that location. Select the desired object by moving the mouse pointer over it and left clicking on it.

**Figure 6-1 Selection List Form**



The image shows a dialog box titled "Selection List" with a close button (X) in the top right corner. Inside the dialog is a table with four columns: TYPE, ID, STORY, and TOWER. The table contains four rows of data. The first row, "Joint 17 Story4 T1", is highlighted in blue. Below the table are two buttons: "OK" and "Cancel".

TYPE	ID	STORY	TOWER
Joint	17	Story4	T1
Beam	B9	Story4	T1
Wall	W2	Story4	T1
Floor	F1	Story4	T1

### Selecting by Coordinates

Using the **Select menu > Select > Coordinate Specification** command, select objects by clicking on a point in a XY, XZ, or YZ plane.

### Selecting by Feature

Using the **Select menu > Select** command, select objects by their various features, such as:

- All objects of a particular type, e.g., Columns, Beams, Braces, etc.
- All objects that have a given section or property type
- All objects that have a particular label or unique name



- All objects that belong to the same group
- All objects that belong to a particular tower or story

These selection methods operate independently of the display windows, and affect all objects having a given feature even if those objects are not being displayed.


### **Deselect Command**

Deselect objects one at a time by left clicking on the selected objects. Alternatively, use the **Select menu > Deselect** command and its subcommands for quicker and more specific deselection actions. This command provides access to subcommands similar to those described in this chapter for selection, except that executing the **Select menu > Deselect** command and an associated subcommand deselects rather than selects an object(s). For example, assume that you want to select all of the objects in your model except for columns. Do this quickly and easily by first using the **Select menu > Select > All** command and then using the **Select menu > Deselect > Object type** command and highlighting Columns.


### **Invert Selection Command**

The **Select menu > Invert Selection** command selects all objects not currently selected, and deselects those previously selected.

### **Get Previous Selection Command**

The **Select menu > Get Previous Selection** command selects the previously selected object(s). For example, assume you have selected some frame objects by clicking on them and assigned frame section properties to them. Use the **Get Previous Selection** command or the **Get Previous Selection** button  to select the same frame objects and assign something else to them, such as member end releases.

## Clear Selection Command

The **Select menu > Clear Selection** command and its associated **Clear Selection** button  clear all currently selected objects. It is an all or nothing command. It cannot selectively clear a portion of a selection.

# Assign/Change Properties

## Objective

This chapter describes how to assign or change the properties of structural objects in the model.

## Assign

In creating the model, the user draws joint, frame, shell, link, and tendon objects. To enable analysis and design, those objects must be assigned properties, such as material properties, frame sections, wall/slab/deck sections, link properties, tendon properties, and loads, among others. Note that the assign menu lists the various properties that can be assigned. Also note that the assignment of loads is explained in Chapter 8 of this guide.

As shown in Table 7-1, the types of assignments available depend on the type of object. Assignments also depend on the type of design (e.g., steel versus concrete versus composite design).

**TABLE 7-1 Possible Assignments to Objects by Object Type**

<b>Object</b>	<b>Assignment Option</b>	<b>Name of Input Form*</b>
<b>Joint</b>		Joint Assignment -
	Restraints	Restraints
	Springs	Springs
	Diaphragms	Diaphragms
	Panel Zone	Panel Zone Property
	Additional Mass	Additional Mass
	Joint Floor Meshing Options	Joint Floor Meshing Option
<b>Frame</b>		Frame Assignment -
	Section Property	Section Property
	Property Modifiers	Property Modifiers
	Releases/Partial Fixity	Releases/Partial Fixity
	End Length Offsets	End Length Offsets
	Insertion Point	Insertion Point
	Local Axes	Local Axes
	Output Stations	Output Stations
	Tension/Compression Limits	Tension/Compression Limits
	Hinges	Hinges
	Hinge Overwrites	Hinge Overwrites
	Line Springs	Line Springs
	Additional Mass	Additional Mass
	Pier Label	Pier Label
	Spandrel Label	Spandrel Label
	Frame Auto Mesh Options	Frame Auto Mesh Options
	Frame Floor Meshing Options	Frame Floor Meshing Option
	Moment Frame Beam Connection Type	Moment Frame Beam Connection Type
	Column Splice Overwrite	Column Splice Overwrite
	Nonprismatic Property Parameters	Nonprismatic Property Parameters
	Material Overwrite (not applicable to section designer, nonprismatic, auto select, encased rectangle/circle, or filled tube/pipe sections)	Material Overwrite
	Column/Brace Rebar Ratio for Creep Analysis	Rebar Ratio for Creep Analysis
	Floor Cracking	Assign Floor Cracking Option to Beam Objects

TABLE 7-1 Possible Assignments to Objects by Object Type

Object	Assignment Option	Name of Input Form*
Shell		Shell Assignment -
	Slab Section	Slab Section
	Deck Section	Deck Section
	Wall Section	Wall Section
	Openings	Openings
	Stiffness Modifiers	Stiffness Modifiers
	Thickness Overwrites	Thickness Overwrites
	Insertion Point	Insertion Point
	Diaphragms	Diaphragms
	Edge Releases	Edge Releases
	Rib Locations	Rib Locations
	Local Axes	Local Axis
	Area Springs	Area Springs
	Additional Mass	Additional Mass
	Pier Label	Pier Label
	Spandrel Label	Spandrel Label
	Wall Hinge	Hinges
	Reinforcement for Wall Hinge	Wall Hinge Reinforcement Select
	Floor Auto Mesh Options	Floor Auto Mesh Options
	Wall Auto Mesh Options	Wall Auto Mesh Options
	Auto Edge Constraint	Auto Edge Constraints
	Material Overwrite	Material Overwrite
	Wall Rebar Ratio for Creep Analysis	Rebar Ratio for Creep Analysis
	Floor Cracking	Assign Floor Cracking Option to Floor Objects
Link		Link Assignment -
	Link Properties	Link Property
	Local Axes	Local Axes
Tendon		
	Tendon Properties	Tendon Property Assign

\* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

View the assignments made to joint, frame, shell, link, and tendon objects by *right* clicking on the object. The appropriate Joint Object Information, Frame Object Information, Shell Object Information, Link Ob-

ject Information, or Tendon Object Information form will display. Click on the Assignments tab.

In each case, select an object before executing the desired assignment command (e.g., select a frame object before using the **Assign menu > Frame > Section Property** command). As explained in Chapter 6 of this guide, using the Ctrl key and left clicking on a location in the model can simplify the process of selecting objects when multiple objects may be present at the same location or if selecting objects is new to the user and seems challenging.

The availability of commands depends on the type of object selected. The input forms include object/assignment-specific input fields that enable refinement of the assignment. Modifications to the assignments can be made by accessing the input forms using the appropriate Assign menu command.

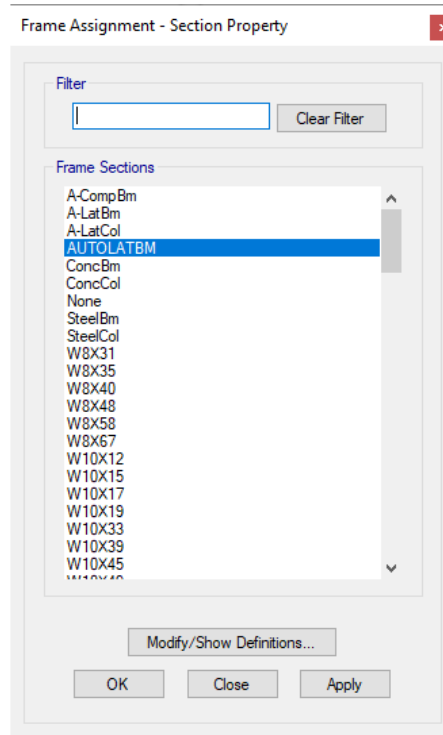
The forms typically include **OK**, **Close**, and **Apply** buttons that can be used to accept or delete changes made to the forms.

Note that the combination of the type of object, name of the command and name of the input form provides an indication of what can be achieved by using a particular command.

### Assign the AUTOLATBM Auto Select Section List

The AUTOLATBM Auto Select Section list created as described in Chapter 5 consists of various sections that can be assigned to a frame object. Thus, in making the assignment, the user should *not* select a joint or shell object in the model, or click the Joint or Shell commands on the Assign menu.

Rather, the user should select a frame object (e.g., a beam) and then click the **Assign menu > Frame > Section Property** command. This will display the Frame Assignment - Section Property form shown in Figure 7-1.





**Figure 7-1 Frame Assignment – Section Property form**

Scroll down the list of properties to locate and highlight the name of the Auto Select Section List to be assigned; AUTOLATBM in this example. Click the **Apply** button and the assignment of the Auto Select Section List named AUTOLATBM is complete. Close the Frame Assignment - Section Property form with the **Close** button.

### **Make an Assignment as the Object is Drawn**

An Auto Select Section List can also be assigned when the frame object is being drawn on the model. Using this method, select the desired Auto Select Section list by name from the Property drop-down list in the Properties of Object Box that appears when a drawing tool is selected. Use of the drawing tools is described in Chapter 5 of this guide along with figures showing the Properties of Object boxes for joint, frame, shell, and tendon objects.

## Make an Assignment using the Model Explorer

Any Frame Section or Auto Select Section List that has been defined can be assigned from the Model Explorer using "drag & drop." On the Model tab in the Model Explorer, click on the  Properties node to expand the tree and then on the  Frame Sections node to see a list of the available sections. Click on the desired section (or Auto Select List) and while holding down the left-mouse button, drag the section onto a frame object - the frame object where the section will be placed will be highlighted with a colored line. Release the mouse button to assign the section.

## Check the Sections in an Auto Select Section List

As indicated previously, several Auto Select Section Lists are built into the program. To review the sections included in any Auto Select Section Lists, whether built in or user-specified, complete the following steps:

1. Click the **Define menu > Section Properties > Frame Sections** command. The Frame Properties form will display.
2. Highlight the name of the Auto Select Section List to be checked in the Properties list.
3. Click the **Modify/Show Property** button. The Frame Section Property Data form displays; the sections included in the selected auto select section list are listed in the Auto Select List area of the form, available for review.
4. Click the **Cancel** button to close the form.



# Load the Structural Model

## Objective

This chapter describes how to define structural loads for the model.

## Structural Loads

The program allows the user to define a variety of structural loads, including dead, live, earthquake and wind loads. The user then assigns the loads to various structural objects in the model. An unlimited number of load patterns can be defined.

Note that the steel frame, concrete frame, composite beam, composite column, steel joist, concrete shear wall, and concrete slab design manuals describe design combinations in accordance with building codes.

## Define the Load Patterns

To add a load pattern, click the **Define menu > Load Patterns** command or expand the tree on the Model tab in the Model Explorer and right click on Load Patterns to access the Define Load Patterns form. Complete the following actions using that form:

1. Type the name of the load pattern in the Load edit box. The program does not allow use of duplicate names.
2. Select a load type from the Type drop-down list.
3. Type a self-weight multiplier in the Self-Weight Multiplier edit box (see the explanation about the **self-weight multiplier** that follows).
4. If the load type specified is Seismic or Wind, select an option from the Auto Lateral Load drop-down list.
5. Click the **Add New Load** button.

**Note:** If you select an automatic lateral load in the Auto Lateral Load drop-down list, click the **Modify Lateral Load** button and review or modify the parameters for the automatic lateral load in the resulting form. Then click the **OK** button to return to the Define Load Patterns form.

## AUTO LATERAL LOAD

Auto Lateral Loads automate the application of code defined seismic and wind loads. Only one code based auto lateral load may be assigned for a given load pattern. If the Type has been set to Seismic, then the Auto Lateral Load drop-down list will show an extensive list of seismic codes for determining earthquake loads. Once a code has been selected, click the **Modify Lateral Load** button to display the Seismic Loading form listing parameters for site coefficients, periods, and load directions.

**Note:** If your model has more than one tower, do not use a Seismic type Auto Lateral Load, but perform a Response Spectrum or Time History

analysis instead. Using a seismic Auto Lateral Load with multiple towers will likely result in an incorrect distribution of lateral loads.

If the Type has been set to Wind, then the Auto Lateral Load drop-down list will show a list of available codes for wind loads. Once a code has been selected, click the **Modify Lateral Load** button to display the Wind Load Pattern form, where coefficients and parameters may be input and reviewed. If the exposure is set to the *Extents of Rigid Diaphragms* option, the program will automatically calculate and apply the different code defined wind load permutations to the diaphragms. With a load pattern highlighted that has *ASCE 7-16* as the Auto Lateral Load, clicking the **Modify Lateral Load** button will display the Wind Load Pattern - ASCE 7-16 form shown in Figure 8-1.

**Wind Load Pattern - ASCE 7-16**

**Exposure and Pressure Coefficients**

- ☒ Exposure from Extents of Diaphragms
- ☐ Exposure from Frame and Shell Objects
  - ☐ Include Shell Objects
  - ☐ Include Frame Objects (Open Structure)

**Wind Pressure Coefficients**

- ☐ User Specified
- ☒ Program Determined

Windward Coefficient,  $C_{pw}$

Leeward Coefficient,  $C_{pl}$

**Wind Exposure Parameters**

Wind Direction and Exposure Width

Case (ASCE 7-16 Fig. 27.3-8)

e1 Ratio (ASCE 7-16 Fig. 27.3-8)

e2 Ratio (ASCE 7-16 Fig. 27.3-8)

**Wind Coefficients**

Wind Speed (mph)


Set	Angle	ASCE Case	e1 Ratio	e2 Ratio
1	0	1	0	0
2	90	1	0	0
3	0	2	0.15	0
4	0	2	-0.15	0
5	90	2	0.15	0
6	90	2	-0.15	0
7	0	3	0	0
8	90	3	0	0
9	0	4	0.15	0.15
10	0	4	-0.15	-0.15
11	90	4	0.15	0.15
12	90	4	-0.15	-0.15

Bottom Story  Base

☐ Include Parapet

Parapet Height  ft

**Figure 8-1 Wind Load Pattern - ASCE 7-16 form**

1. Select *Create All Sets* from the Case drop-down list.
2. Hold the mouse cursor over the information icon  to display a table listing the direction angles and ratios for the ASCE cases. The ASCE 7-16 code prescribes 12 different wind load permutations.

3. Click the **OK** button to close the form.

With a load pattern highlighted that has *EUROCODE 1 2005* as the Auto Lateral Load, clicking the **Modify Lateral Load** button will display the Wind Load Pattern - EuroCode 1 2005 form.

On the Wind Load Pattern - EuroCode 1 2005 form, click the **Modify/Show** button for Wind Directions and Exposure Widths to display the Wind Exposure Width Data form shown in Figure 8-2.

The form is titled "Wind Exposure Width Data". It features a text input field for "Direction Angles (use semicolon as separator)" containing "0; 45; 90" and a "deg" unit label. Below this is a table titled "Exposure Set 1 of 3: 0 deg". The table has six columns: Story, Diaphragm, Width ft, Depth ft, X Ordinate ft, and Y Ordinate ft. The rows represent Story4, Story3, Story2, and Story1, all with Diaphragm D1, Width 73, Depth 73, X Ordinate 36, and Y Ordinate 36. At the bottom left, there are three tabs labeled 1, 2, and 3, with tab 1 selected. Below the tabs are two radio buttons: "Exposure Set Width is Auto Calculated from Diaphragm Extents by the Program" (selected) and "Exposure Set Width is User Defined". At the bottom right are "OK" and "Cancel" buttons.

Story	Diaphragm	Width ft	Depth ft	X Ordinate ft	Y Ordinate ft
Story4	D1	73	73	36	36
Story3	D1	73	73	36	36
Story2	D1	73	73	36	36
Story1	D1	73	73	36	36

**Figure 8-2 Wind Exposure Width Data form**

1. Type **;45** after 90 in the Direction Angles edit box (make sure to precede 45 with a semicolon ;). This adds an additional wind load at a direction of 45 degrees to the previously defined angles of 0 and 90.

2. Note that the number buttons in the lower left-hand corner of the table expand from two to three - click on these buttons to display the exposure set tables for each angle setting.
3. Click the **OK** button to close the form.

### **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 (search for “material properties” using the **Help menu > ETABS Help** command for more information about material properties and the Material Properties command).

A portion of the self-weight can be applied to any load pattern. The self-weight multiplier controls what portion of the self-weight is included in a load pattern. A self-weight multiplier of 1 includes the full self-weight of the structure in the load pattern. A self-weight multiplier of 0.5 includes one-half of the self-weight of the structure in the load pattern.

**Normally a self-weight multiplier of 1 should only be specified in one load pattern, usually the dead load pattern.** All other load patterns then have self-weight multipliers of zero. Note that if a self-weight multiplier of 1 is included in two different load patterns, and then those two load patterns are combined in a load case or combination, the results are based on an analysis where double the self-weight of the building has been applied as a load.

### **Modify an Existing Load Pattern**

Use the following procedure and the Define Load Patterns form to modify an existing load pattern. Recall that the Define Load Patterns form is accessed using the **Define menu > Load Patterns** command:

1. Highlight the existing load pattern in the Loads area of the form. Note that the data associated with that load pattern appears in the edit boxes and drop-down lists at the top of the Loads area.
2. Modify any of the data in the Loads area for the load case.

3. Click the **Modify Load** button. If necessary, click the **Modify Lateral Load** button to modify the automatic lateral load parameters.

### Delete an Existing Load Pattern

Use the following procedure to delete an existing load pattern in the Define Load Patterns form. Note that when a load pattern is deleted, all of the loads assigned in the model as a part of that load pattern are also deleted.

- Highlight the existing load pattern in the Loads area of the form. Note that the data associated with that load pattern appears in the edit boxes and drop-down lists at the top of the Loads area.
- Click the **Delete Load** button.

### Define Shell Uniform Load Sets

Shell uniform load sets define loads that consist of several different load patterns, e.g., the load set may contain loads from both dead and live patterns. To add a shell uniform load set, click the **Define menu > Shell Uniform Load Sets** command to access the Shell Uniform Load Sets form. Complete the following actions using that form:

1. Click the **Add New Load Set** button to display the Shell Uniform Load Set Data form.
2. On the Shell Uniform Load Set Data form, type the name of the shell load set in the Uniform Load Set Name edit box. The program does not allow use of duplicate names.
3. Click the **Add** button.
4. Select a load pattern from the Load Pattern drop-down list (only load patterns that have previously been defined may be selected).
5. Type a load value in the Load Value edit box.
6. Click the **OK** button to return to the Shell Uniform Load Sets form.
7. Click the **OK** button to close the Shell Uniform Load Sets form.

## Assign Structural Loads

The load patterns defined in the previous section are required in order to be able to assign loads to joints, frames, shells, and tendons. The user must first select the object before a load can be assigned to the object. Chapter 6 of this guide describes how to select structural objects.

After the object has been selected, click the **Assign menu** command to access the applicable submenu and assignment options. Table 8-1 identifies the submenus and options.

**TABLE 8-1 Load Commands on the Assign Menu**

<i>sub menus</i>	{	<b>Joint Loads</b>	<b>Frame Loads</b>	<b>Shell Loads</b>	<b>Tendon Loads</b>
		Force	Point	Uniform Load Sets	Tendon Loads
<i>assignment options</i>	{	Ground Displacement	Distributed	Uniform	Tendon Losses
		Temperature	Temperature	Non-uniform	
			Open Structure Wind Parameters	Temperature	
				Wind Pressure Coefficient	

Note that the type of object selected determines which assignment can be made. For example, a ground displacement assignment cannot be made to a frame or shell object. Thus, if a frame object (e.g., a beam) or a shell object (e.g., a wall) has been selected before clicking the **Assign menu** command, attempting to assign joint loads will result in an error message.

A form will appear after clicking the **Assign menu** command, the submenu applicable to the type of object, and the desired assignment option. Table 8-2 identifies the forms generated when the various commands are used.

**TABLE 8-2 Input Forms for Load Commands on the Assign Menu**

Command	Name of Input Form*
Joint Loads >	Joint Load Assignment -
Force	Force
Ground Displacement	Ground Displacement
Temperature	Temperature
Frame Loads >	Frame Load Assignment -
Point	Point
Distributed	Distributed
Temperature	Temperature
Open Structure Wind Parameters	Open Structure Wind Parameters
Shell Loads >	Shell Load Assignment -
Uniform Load Sets	Uniform Load Set
Uniform	Uniform
Non-uniform	Non-uniform
Temperature	Temperature
Wind Pressure Coefficient	Wind Pressure Coefficient
Tendon Loads >	
Tendon Loads	Tendon Load
Tendon Losses	Tendon Loss Options

\* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

Although the form names vary depending on the command used, each form has a drop-down list that allows the user to select the load pattern to be assigned. Logically, the available load patterns vary depending on the type of assignment. The forms also include other object/assignment-specific input fields that enable the user to refine the load assignment.



# Define Load Cases

## Objective

This chapter describes how to define load cases.

## Review/Create Load Cases

A load case defines how loads are to be applied to the structure, and how the structural response is to be calculated. Analyses are classified in the broad sense as either linear or nonlinear, depending on how the model responds to the loading. The results of linear analyses may be superposed, i.e., added together after analysis. The results of nonlinear analyses normally should not be superposed. Instead, all loads acting together on the structure should be combined directly within the nonlinear load case.

After all geometry and load input has been specified for a model, review, modify, or add load cases using the **Define menu > Load Cases** command. The Load Cases form shown in Figure 9-1 will appear. Highlight a

load case (ETABS automatically generates a load case for each load pattern defined as well as a Modal case) and click the **Modify/Show Case** button to review or modify the load case definition. Click the **Delete Case** button to delete the highlighted load case.

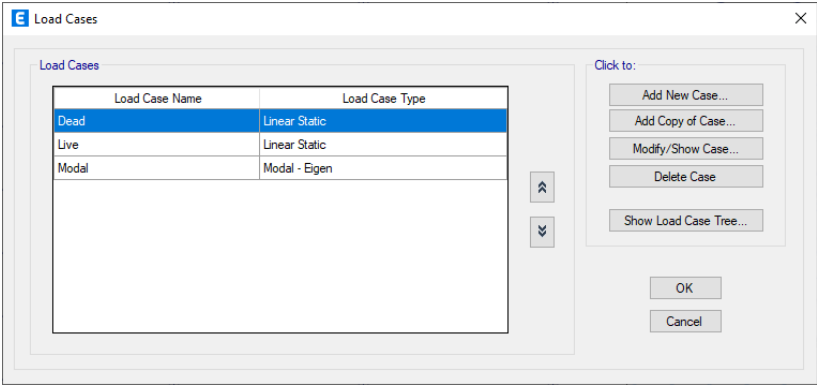


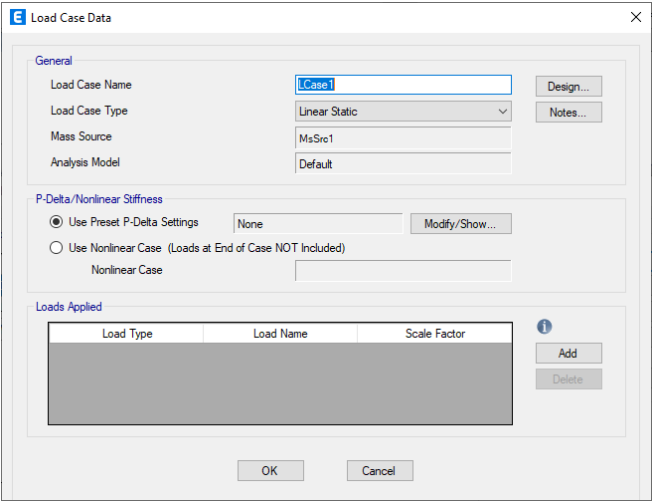
Figure 9-1 Load Cases form

To define a new load case, click the **Add New Case** button to display the Load Case Data form shown in Figure 9-2.

Use that form to specify the following information:

- The name of the load case. ETABS does not allow duplicate names.

Figure 9-2  
Load Case Data  
form



- A load case type, which can be selected from the Load Case Type drop-down list. The default setting is linear static, but nonlinear static, nonlinear staged construction, modal, response spectrum, time history, buckling, and hyperstatic are all available. A static case considers loads defined in a load pattern, modal determines the eigen- or ritz-vectors of the structure, a response spectrum performs a statistical calculation of the response caused by acceleration loads, a time history applies time-varying loads, buckling calculates the buckling modes, and hyperstatic is used in slab design. Nonlinear static may be used for pushover analysis, while nonlinear staged construction allows portions of the structure to be added or removed.
- The load case subtype when applicable, e.g., linear modal, nonlinear modal (FNA), linear direct integration, or nonlinear direct integration when the load case type is time history.
- The type of P-Delta when applicable. For a load case type of linear static, the P-Delta option may be reviewed or changed by clicking the **Modify/Show** button in the P-Delta/Nonlinear Stiffness area. The Preset P-Delta Options form shown in Figure 9-3 will display. This form may also be accessed by using the **Define menu > P-Delta Options** command.

**Figure 9-3**  
**P-Delta Options**  
**form**

The screenshot shows the 'Preset P-Delta Options' dialog box. It features a title bar with an 'E' icon and a close button. The main content area is divided into three sections. The first section, 'Automation Method', contains three radio buttons: 'None' (which is selected), 'Non-iterative - Based on Mass', and 'Iterative - Based on Loads'. The second section, 'Iterative P-Delta Load Case', contains a table with two columns, 'Load Pattern' and 'Scale Factor'. The table is currently empty. To the right of the table are three buttons: 'Add', 'Modify', and 'Delete'. Below the table is a text input field labeled 'Relative Convergence Tolerance'. At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

- The loads to be applied. For a linear static load case type, this is typically a load pattern with a scale factor.
- Other parameters when applicable. For a load case type of response spectrum, a modal load case will be required using either eigen- or ritz-vectors.

## Define an Auto Construction Sequence Case

A construction sequence load case automates the inclusion of stories in a model to account for the sequential effects of construction. The sequential application of structure and load as a building is built may result in a significantly different distribution of forces (i.e., dead load) than would occur were the loads to be applied only after the building is complete. The Auto Construction Sequence Case performs a multi-step analysis that follows the sequential construction of the building.

After all geometry and load input has been specified for a model, use the **Define menu > Auto Construction Sequence Case** command to specify a construction sequence. The Auto Construction Sequence Load Case form shown in Figure 9-4 will appear. Use that form to specify the following information:

- Whether the case is active and the name of the case.
- The number of stories to be included in each construction sequence - one is the default.
- The loads that should be applied during this case - typically these are dead loads.
- Whether this sequential case should replace dead load cases in design combinations.

**Figure 9-4**  
**Auto Construction**  
**Sequence Load**  
**Case**  
**form**

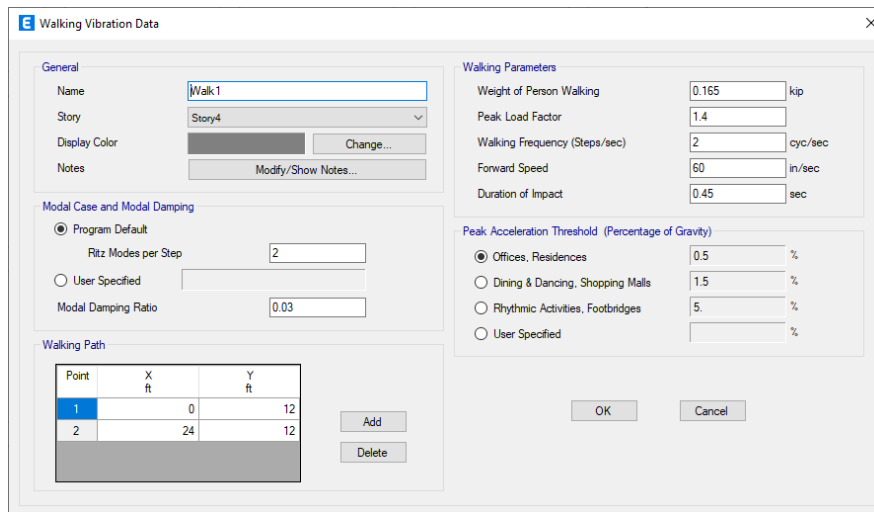
Load Pattern Name	Scale Factor
Dead	1

## Define a Walking Vibration Case

A walking vibration case allows for the automated application of vertical pulse loadings across a floor to dynamically simulate the footfall of a person walking. The output from this analysis is an acceleration response that can be compared against specified thresholds to determine whether the footfall impact will be perceptible. Use the **Define menu > Walking Vibrations** command to specify walking vibration data. On the Walking Vibrations form, click the **Add Walking Vibration** button to display the Walking Vibration Data form shown in Figure 9-5 will appear. Use that form to specify the following information:

- The name of the walking vibration case and the story level where the path is being defined.
- The walking parameters that determine the shape and size of the pulse load.

- The modes and modal damping that should be used when calculating the dynamic excitation.
- The peak acceleration threshold for comparison against the computed peak acceleration.
- The path of the person walking.



The image shows a software dialog box titled "Walking Vibration Data". It is divided into several sections for configuring walking vibration analysis parameters.

**General**

- Name: Walk1
- Story: Story4
- Display Color: [Color swatch] Change...
- Notes: [Text area] Modify/Show Notes...

**Modal Case and Modal Damping**

- ☒ Program Default
  - Ritz Modes per Step: 2
- ☐ User Specified
  - Modal Damping Ratio: 0.03

**Walking Path**

Point	X ft	Y ft
1	0	12
2	24	12

Buttons: Add, Delete

**Walking Parameters**

- Weight of Person Walking: 0.165 kip
- Peak Load Factor: 1.4
- Walking Frequency (Steps/sec): 2 cyc/sec
- Forward Speed: 60 in/sec
- Duration of Impact: 0.45 sec

**Peak Acceleration Threshold (Percentage of Gravity)**

- ☒ Offices, Residences: 0.5 %
- ☐ Dining & Dancing, Shopping Malls: 1.5 %
- ☐ Rhythmic Activities, Footbridges: 5. %
- ☐ User Specified: %

Buttons: OK, Cancel

**Figure 9-5 Walking Vibration Data form**

# Edit the Model Geometry

## Objective

This chapter describes how to edit the model quickly and easily while maintaining model integrity.

## Editing Options

During the course of creating the model, the model may require editing. Table 10-1 identifies the various edit commands available in the program. Some are familiar Windows commands.

In most cases, first select the joint, frame, tendon, or shell object, then click the appropriate menu or button. In some cases, the action will be immediate (e.g., the Undo or Redo commands). In other cases, a form will display that allows the user to specify how the object is to be edited (e.g., the **Edit menu > Align Joints/Frames/Edges** command accesses the Align Joints/Frames/Edges form, which allows the user to align *joints* to the x, y, z coordinate or to the nearest frame, or to trim or extend

*frames*). In other cases, the command is a toggle that, when enabled, will affect subsequent actions. Note that the type of commands and options available depend on the type of object being edited.

**TABLE 10-1 Edit Commands in ETABS**







Command	Action	Immediate/ Form* Used/ Toggle
<b>Undo and Redo</b>	Undo  deletes the last performed action. Redo  restores the last step that was undone.	Immediate
<b>Cut, Copy and Paste</b>	Generally similar to the standard cut, copy and paste Windows commands, with some ETABS specific behaviors. Only active in plan or plan perspective view.	Immediate
<b>Delete</b>	Delete  deletes the selected object(s) and all of its assignments (loads, properties, supports and the like).	Immediate
<b>Add to Model from Template</b>		
> Add 2D Structure	Add objects to a model using various 2D templates.	Add to Model - 2D Structure form
> Add 3D Structure	Add objects to a model using various 3D templates.	Add to Model - 3D Structure form
<b>Edit Towers, Stories and Grid Systems</b>	Edit Towers, Stories and Grid Systems  edits towers (if multiple towers have been allowed), story data, and grid systems. Resulting forms allow users to select an existing tower or define a new one; modify/show existing story data or add a new story; and add or modify/show existing grid systems.	Edit Towers, Stories and Grid Systems form and Tower, Story and Grid System Data form
<b>Add Grid at Selected Joints</b>	Adds grid lines at selected joints.	Add Grid Lines at Selected Joint form



TABLE 10-1 Edit Commands in ETABS

Command	Action	Immediate/ Form* Used/ Toggle
<b>Grid Options</b>		
> Glue Joints to Grids	"Glues" joint objects that lie directly on grid lines to those grid lines. When a joint object is glued to a grid line and the grid line is moved, the joint object moves with the grid line. Frame and shell objects that are attached to the joint object when it is moved remain attached to the joint object and move or resize as appropriate.	Toggle
> Lock OnScreen Grid System Edit	Allows users to lock out the ability to move grid lines graphically on-screen using the <b>Reshape Object</b> command.	Toggle
<b>Interactive Database</b>	Allows model definition data and certain design data to be edited using interactive database tables.	Choose Tables for Interactive Editing
<b>Replicate</b>	Replicate  replicates one or more objects and most of the object's assignments. Note that replicated objects will <b>not</b> replace objects already placed at a location.	Replicate form that accesses options form
<b>Extrude</b>		
> Extrude Joints to Frames	Creates frame objects from joints. Options are available for linear or radial extrusion. This feature is especially suited to creating beams/columns from joints.	Extrude Joints to Lines form
> Extrude Frames to Shells	Extrude Frames to Shells creates shell objects from frames. Options are available for linear or radial extrusion. This feature is especially suited to creating shell objects from beams.	Extrude Frames to Shells form
<b>Merge Joints</b>	Merge Joints  merges joints within a user-specified tolerance distance of the selected joint.	Merge Selected Joints form
<b>Align Joints/Frames/Edges</b>	Align Joints/Frames/Edges helps the user align objects in the model. Search for "edit joints frames edges" using the <b>Help menu &gt; ETABS Help</b> command for important notes about using this command.	Align Selected Frames/Edges/ Joints form

**TABLE 10-1 Edit Commands in ETABS**

Command	Action	Immediate/ Form* Used/ Toggle
<b>Move Joints/Frames/ Shells</b>	Move Joints/Frames/Shells helps the user move objects in the model. Search for "move joints frames shells" using the <b>Help menu &gt; ETABS Help</b> command for more information.	Move Joints/Frames/ Shells form
<b>Edit Frames</b>		
> Divide Frames	Divide Frames divides a frame object into multiple frame objects.	Divide Selected Frames form
> Join Frames	Join Frames joins two or more collinear frame objects with common end joints and the same type of property into a single frame object.	Immediate <i>Can use Undo</i>
> Reverse Frame Connectivity	Reverses the local I and J ends of a frame object. This reversing of ends results in a change in the orientation of the object's local axes.	Immediate <i>Can use Undo</i>
> Modify/Show Frame Type	Allows modification of the frame type to be either straight or curved.	Frame Object Type Options form
<b>Edit Shells</b>		
> Divide Shells	Divide Shells divides selected shells into additional objects using user-specified options.	Edit Shell form
> Merge Shells	Merge Shells merges two shell objects that have a common edge or overlap into one larger shell object.	Immediate <i>Can use Undo</i>
> Expand/Shrink Shells	Expand/Shrink Shells expands or shrinks a shell object using a user-specified offset value.	Expand/Shrink Areas form
> Split Shell Edges	Adds joint objects at the mid-point of each edge of a shell object.	Immediate <i>Can use Undo</i>
> Remove Joints from Shells	Removes joint objects from a shell object if they are not located at corners.	Immediate <i>Can use Undo</i>
> Chamfer Slab Corners	Allows the user to add chamfers to the corners of slabs.	Chamfer Slab Corners form
> Explode User Mesh to Shells	If a user mesh exists for the selected shell, explodes the mesh into individual shell objects.	Immediate <i>Can use Undo</i>

TABLE 10-1 Edit Commands in ETABS

Command	Action	Immediate/ Form* Used/ Toggle
> Merge Shells to Create User Mesh	Merges selected shell objects into a single shell object with an equivalent user mesh.	Immediate <i>Can use Undo</i>
> Remove User Mesh from Shell Object	Removes the user mesh from the selected shell object.	Immediate <i>Can use Undo</i>
> Convert Current Mesh to User Mesh	Converts the current mesh of the select shell object to a user mesh.	Immediate <i>Can use Undo</i>
> Reverse Wall Local 3 Axis	Reverses the local 3 (normal) axis of wall objects.	Immediate <i>Can use Undo</i>
> Divide Walls for Openings	Divides selected walls into smaller shell objects to account for openings.	Immediate <i>Can use Undo</i>
> Modify/Show Slab Edge Type	Modify/Show Slab Edge Type allows slab edges to be modified.	Slab Shell Object Edge Type Options form
> Modify/Show Wall Curve Type	Modify/Show Wall Curve Type allows wall shapes to be modified, i.e., changing a straight wall into a wall with a curve.	Wall Shell Object Curve Type Options form
<b>Edit Links</b>		
> Reverse Link Connectivity	Reverses the local I and J ends of a link object. This reversing of ends results in a change in the orientation of the object's local axes.	Immediate <i>Can use Undo</i>
<b>Add/Edit Tendons</b>		
> Add Tendons in Strips	Add banded or distributed tendons to selected design strips.	Quick Tendon Layout form
> Edit Plan Layout (Horizontal)	Edit the plan geometry of selected tendons.	Tendon Object Type Options
> Edit Vertical Profile	Edit or review the profile of the selected tendons.	Tendon Vertical Profile
> Reset Supports and Spans to Default	Reset all support and span settings for selected tendons to the default values.	Immediate
> Copy Vertical Profile	Copy the vertical profile of the selected tendon so that it may be "pasted" to another tendon.	Immediate
> Paste Vertical Profile	Paste the previously copied vertical profile on to the selected tendons.	Immediate

**TABLE 10-1 Edit Commands in ETABS**

Command	Action	Immediate/ Form* Used/ Toggle
<b>Add/Edit Slab Rebar</b>		
> Add Slab Rebar	Add rebars to concrete slabs to supplement the reinforcing calculated during slab design.	Slab Rebar Object Data form
> Edit Slab Rebar	Edit selected rebar previously added using the Add Slab Rebar command.	Slab Rebar Object Data form
<b>Add/Edit Design Strips</b>		
> Add Design Strips	Add design strips to any floor using grid lines.	Add Design Strips form
> Edit Strip Widths	Adjust the width of design strips automatically or by inputting values.	Edit Strip Widths form
<b>Auto Relabel All</b>	Reorders all object labels based on their geometric order rather than in the sequential order drawn.	Immediate <i>Cannot use Undo</i>
<b>Nudge</b>	Works with Ctrl and arrow keys to move objects. Allows the user to select objects and move them a predefined distance. For more information, search for "nudge" using the Help menu.	Immediate

\* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

# Analyze the Model

## Objective

This chapter describes how to analyze the model.

## Set the Mesh Options

If your model contains wall objects, or floor objects that have plate bending behavior such as cast-in-place slabs, review the meshing options (e.g., maximum mesh size) before running the analysis by using the **Analyze menu > Automatic Mesh Settings for Floors** or **Analyze menu > Automatic Rectangular Mesh Settings for Walls** commands. The Automatic Mesh Options (for Floors) form is shown in Figure 11-1. Default mesh settings for floors and walls may be reviewed by using the **Assign menu > Shell > Floor Auto Mesh Options** and the **Assign menu > Shell > Wall Auto Mesh Options** commands, respectively.

The automatic meshing performed by ETABS for floor slabs may be either general or rectangular. The general mesh uses quadrilateral and tri-


angular elements, while the rectangular creates a mesh that is parallel and perpendicular to the local axes of the shell objects. Generation of the meshes are influenced by the locations of floor and wall objects, openings, beams, columns, and joints. To preview the element mesh before running the analysis and design, use the **View menu > Set Display Options** command and check the Shell Analysis Mesh option under the Other Special Items area of the General tab on the Set View Options form.

Figure 11-1 Automatic Mesh Options form

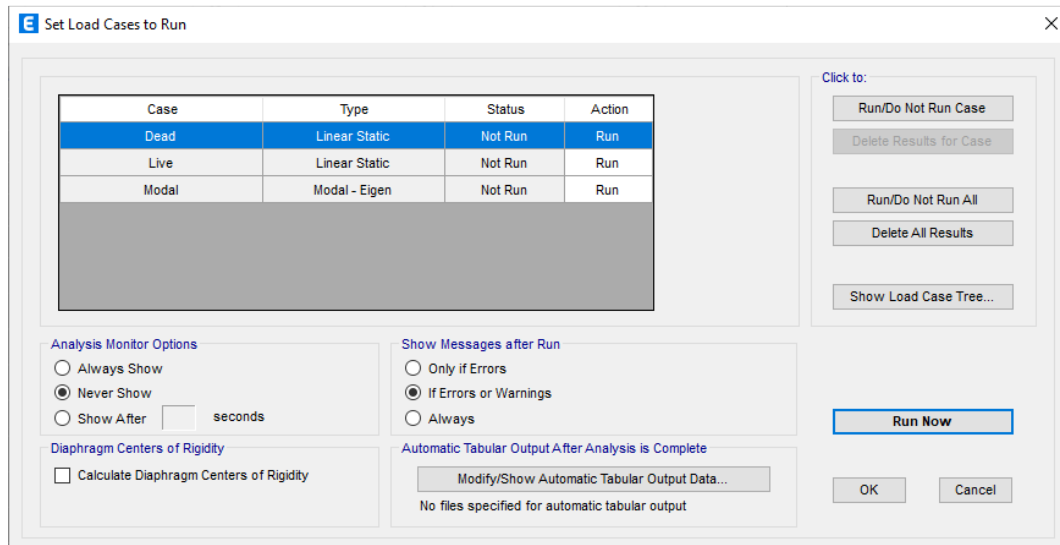
## Model Analysis

Prior to running the analysis, verify what load cases are set to run by clicking on the **Analyze menu > Set Load Cases To Run** command. The Set Load Cases to Run form shown in Figure 11-2 will appear.

To add or remove a load case from the analysis, highlight the load case in the Case column and click the **Run/Do Not Run Case** button. Both the status and action for each case are shown in their respective columns. This form also allows the user to set how the Analysis Monitor should be displayed - the default setting is that it *Never Shows*.

To run the analysis, click the **Run Now** button if the Set Load Cases to Run form is still displayed, otherwise click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, .

The program will display an "Analyzing, Please Wait" window if the Analysis Monitor has been set to "Always Show" or "Show After." Data will scroll in this window as the program runs the analysis. After the analysis has been completed, the program performs a few more "bookkeeping actions" that are evident on the status bar in the bottom left-hand corner of the ETABS window.



The dialog box titled "Set Load Cases to Run" contains a table with the following data:

Case	Type	Status	Action
Dead	Linear Static	Not Run	Run
Live	Linear Static	Not Run	Run
Modal	Modal - Eigen	Not Run	Run

Below the table, there are several sections of options:

- Analysis Monitor Options:**
  - ☐ Always Show
  - ☒ Never Show
  - ☐ Show After  seconds
- Diaphragm Centers of Rigidity:**
  - ☐ Calculate Diaphragm Centers of Rigidity
- Show Messages after Run:**
  - ☐ Only if Errors
  - ☒ If Errors or Warnings
  - ☐ Always
- Automatic Tabular Output After Analysis is Complete:**
  - 
  - No files specified for automatic tabular output

On the right side, there is a "Click to:" section with buttons:

- 
- 
- 
- 
- 

At the bottom right, there is a large  button, and at the bottom center, there are  and  buttons.


Figure 11-2 Set Load Cases to Run form

## Model Alive™ Feature

To run the analysis automatically, in a continuous manner, use the **Analyze menu > Model Alive** command. This analysis mode allows the user to make revisions to a model and have the model analysis updated automatically without first having to save the model and then execute the run command. The Model Alive feature can be especially useful on small models because it allows the user to instantly see the effects of any model revision. The Model Alive feature will not be as beneficial on larger models with longer run times.

The ETABS model is not automatically saved when running the analysis in the Model Alive mode. The **File menu > Save** command can be used at any time to save the model; otherwise, the model remains unlocked and unsaved.

## Locking and Unlocking the Model

When the entire analysis process has been completed, the model automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Lock/Unlock Model button, , appears closed. Locking the model prevents any changes to the model that would invalidate the analysis results.



# Design

## Objective

This chapter describes design using the ETABS design postprocessors.

## Design the Structure

The ETABS design postprocessors include the following:

- Steel Frame Design
- Concrete Frame Design
- Composite Beam Design
- Composite Column Design
- Steel Joist Design
- Shear Wall Design

- Concrete Slab Design

To perform the design, first run the analysis (described in Chapter 11), then click the Design menu and select the appropriate design from the drop-down menu. The type of design available depends on the type of members used in the model. That is, the user cannot complete a shear wall design if no shear walls have been included in the model, nor could they do a steel joist design if no steel joists are present.

Similarly, the commands used to execute a design depend on the type of design to be performed. However, each design has commands to address the following:

- Review and/or select overwrites.
- Review and/or select design combinations.
- Start the design or check of the structure.
- Perform interactive design.
- Display input and output design information on the model.

Generally, the sequence for using commands is indicated by their availability. In other words, some commands must be used before other commands become available. This helps the user step through the design process. (Search for “process” using the Help menu to access more information about design processes and the sequence of commands.) Table 12-1 identifies the commands that are used to start design depending on the desired design process.

**TABLE 12-1 Start Design Commands**

<b>Design Process</b>	<b>Command that Starts Design</b>
Steel Frame Design	Start Design/Check
Concrete Frame Design	Start Design/Check
Composite Beam Design	Start Design/Check

**TABLE 12-1 Start Design Commands**

<b>Design Process</b>	<b>Command that Starts Design</b>
Composite Column Design	Start Design/Check
Steel Joist Design	Start Design/Check
Shear Wall Design	Start Design/Check
Concrete Slab Design	Start Design

It is important to understand that design in ETABS is an iterative process. That is, the user should run the analysis and then perform the design and be prepared to run the analysis again and perform the design again each time sections are revised. It may be necessary to repeat this process several times before the design is complete. The objective is to have the analysis sections match the design sections. The user should repeat the analysis/start design process until the analysis and design sections match and design requirements are satisfied.

Tables 12-2 through 12-8 summarize the commands used in each type of design process.

**Note:** With a form displayed in the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

**TABLE 12-2 Steel Frame Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>View/Revise Preferences</b>	Designates which design code to use, as well as defining many other design parameters. Default values are provided for all settings.	Steel Frame Design Preferences form

**TABLE 12-2 Steel Frame Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>View/Revise Overwrites</b>	Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the frame objects to which they are specifically assigned.	Steel Frame Design Overwrites form
<b>Lateral Bracing</b>	Designates lateral bracing as either program determined or user specified.	Lateral Bracing
<b>Select Design Groups</b>	Designates that a group is to be used as a design group. Works only when auto select sections have been assigned to frame objects. All frame objects in the group will be given the same design section.	Steel Frame form
<b>Select Design Combinations</b>	Allows review of the default steel frame design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations during design.	Design Load Combinations Selection form
<b>Start Design/Check</b>	Initiates design process. If frame objects have been selected before this command is clicked, only the selected frame objects will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Interactive Design</b>	Allows the user to review the design results for any frame object and then to interactively change the design overwrites and immediately view the results.	No form; results are displayed onscreen.
<b>Display Design Info</b>	Allows review of some of the results of the steel frame design directly on the program model. Examples of results that can be displayed include design sections, unbraced lengths, effective length factors, allowable stresses, and stress ratio information.	Display Steel Frame Design Results form

TABLE 12-2 Steel Frame Design Commands

Command	Action	Form
<b>Make Auto Select Section Null</b>	Removes auto select section lists from selected frame objects. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual frame sections assigned, not auto select sections. Only works on a user-specified selection.	Warning message <i>Cannot use Undo</i>
<b>Change Design Section</b>	Allows the user to change the design section property assigned to one or more frame objects and then rerun the design without first rerunning the analysis. Only works on a user-specified selection.	Select Sections form
<b>Reset Design Section to Last Analysis</b>	Sets the design section for one or more frame objects back to the last used analysis section. Only works on a user-specified selection.	Immediate <i>Cannot use Undo</i>
<b>Verify Analysis vs Design Section</b>	Verifies that the last used analysis section and the current design section are the same for all steel frame objects in the model.	Immediate
<b>Verify All Members Passed</b>	Reports if structural members have passed the stress/capacity check. An analysis and a design/check of the structure must be completed before this command is used.	Immediate
<b>Reset All Overwrites</b>	Resets the overwrites for all frame objects with the Steel Frame design procedure to their default values.	Warning message <i>Cannot use Undo</i>
<b>Delete Design Results</b>	Deletes all of the steel frame design results but not the current design section (i.e., next analysis section).	Immediate <i>Cannot use Undo</i>

**TABLE 12-3 Concrete Frame Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>View/Revise Preferences</b>	Designates the design code to use, as well as defining other design parameters. Default values are provided for all settings.	Concrete Frame Design Preferences form
<b>View/Revise Overwrites</b>	Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the frame objects to which they are specifically assigned.	Concrete Frame Design Overwrites form
<b>Select Design Groups</b>	Designates that a group is to be used as a design group. All frame objects in the group will be given the same design section.	Concrete Design Group Selection form
<b>Select Design Combinations</b>	Allows review of the default concrete frame design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations during design.	Design Load Combinations Selection form
<b>Start Design/Check</b>	Initiates design process. If frame objects have been selected before this command is used, only the selected frame objects will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Interactive Design</b>	Allows the user to review the design results for any frame object and then to interactively change the design overwrites and immediately view the results.	No form; results are displayed onscreen.
<b>Display Beam Design</b>	Allows display of the concrete beam rebar design results directly on the model.	Beam Design form
<b>Display Design Info</b>	Allows review of some of the results of the concrete frame design directly on the program model. Examples of results that can be displayed include design sections, unbraced lengths and longitudinal reinforcing.	Display Concrete Frame Design Results form

**TABLE 12-3 Concrete Frame Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>Change Design Section</b>	Allows the user to change the design section property assigned to one or more frame objects and then rerun the design without first rerunning the analysis. Only works on a user-specified selection.	Select Sections form
<b>Reset Design Section to Last Analysis</b>	Sets the design section for one or more frame objects back to the last used analysis section. Only works on a user-specified selection.	Immediate Cannot use Undo
<b>Verify Analysis vs Design Section</b>	Verifies that the last used analysis section and the current design section are the same for all concrete frame objects in the model.	Immediate
<b>Verify All Members Passed</b>	Reports if structural members have passed the capacity check. An analysis and a design/check of the structure must be completed before this command is used.	Immediate
<b>Reset All Overwrites</b>	Resets the overwrites for all frame objects with the Concrete Frame design procedure to their default values.	Warning message Cannot use Undo
<b>Delete Design Results</b>	Deletes all of the concrete frame design results but not the current design section (i.e., next analysis section).	Immediate Cannot use Undo

**TABLE 12-4 Composite Beam Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>View/Revise Preferences</b>	Designates the design code to use, as well as defining other design parameters, such as camber. Default values are provided for all settings.	Composite Beam Design Preferences form
<b>View/Revise Overwrites</b>	Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the composite beams to which they are specifically assigned.	Composite Beam Overwrites form

**TABLE 12-4 Composite Beam Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>Select Design Groups</b>	Designates that a group is to be used as a design group. Works only when auto select sections have been assigned to frame objects. When grouped, all beams in the group are given the same beam size, but the shear connectors and camber may be different.	Composite Design Group Selection form
<b>Select Design Combinations</b>	Allows review of the default composite frame design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations during design. Note that separate design combinations are specified for construction loading, final loading considering strength, and final loading considering deflection.	Design Load Combinations Selection form
<b>Start Design/Check</b>	Initiates design process. If frame objects have been selected before this command is used, only the selected frame objects will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Interactive Design</b>	Allows the user to review the design results for any composite beam, select the best design from the various acceptable designs, and interactively change the design assumptions and immediately view the results.	Interactive Composite Beam Design and Review form
<b>Display Design Info</b>	Allows review of some of the results of the composite beam design directly on the program model. Examples of results that can be displayed include beam labels and associated design group names; design sections together with connector layout, camber and end reactions; and stress ratio information.	Display Design Results form



TABLE 12-4 Composite Beam Design Commands

Command	Action	Form
<b>Make Auto Select Section Null</b>	Removes auto select section lists from selected beams. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual beam sections assigned, not auto select sections. Only works on a user-specified selection.	Warning message Cannot use Undo
<b>Change Design Section</b>	Allows the user to change the design section property assigned to one or more beams and then rerun the design without first rerunning the analysis. Only works on a user-specified selection.	Select Sections form
<b>Copy Design</b>	Allows the user to select a template composite beam and copy its design section, camber and percentage of composite action for later use with Paste Design.	Immediate
<b>Paste Design</b>	Allows the user to select target composite beams to which the previously selected design section, camber, and percentages composite action will be assigned. The percentage of composite action of the template beam is applied to the target beams as a minimum percentage of composite action.	Immediate
<b>Reset Design Section to Last Analysis</b>	Sets the design section for one or more beams back to the last used analysis section. Only works on a user-specified selection.	Immediate Cannot use Undo
<b>Verify Analysis vs Design Section</b>	Verifies that the last used analysis section and the current design section are the same for all composite beams in the model.	Immediate
<b>Verify All Members Passed</b>	Reports if structural members have passed the stress/capacity check following a new analysis run, or a change in project composite preferences or composite beam overwrites.	Immediate
<b>Reset All Overwrites</b>	Resets the overwrites for all composite beams with the Composite Beam design procedure to their default values.	Warning message <i>Cannot use Undo</i>

**TABLE 12-4 Composite Beam Design Commands**

Command	Action	Form
<b>Delete Design Results</b>	Deletes all of the composite beam design results but not the current design section (i.e., next analysis section).	Immediate <i>Cannot use Undo</i>

**TABLE 12-5 Composite Column Design Commands**

Command	Action	Form
<b>View/Revise Preferences</b>	Designates the design code to use, as well as defining other design parameters, such as camber. Default values are provided for all settings.	Composite Column Design Preferences form
<b>View/Revise Overwrites</b>	Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the composite columns to which they are specifically assigned.	Composite Column Design Overwrites form
<b>Select Design Combinations</b>	Allows review of the default composite frame design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations during design. Note that separate design combinations are specified for construction loading, final loading considering strength, and final loading considering deflection.	Design Load Combinations Selection form
<b>Start Design/Check</b>	Initiates design process. If frame objects have been selected before this command is used, only the selected frame objects will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Interactive Design</b>	Allows the user to review the design results for any composite column and then to interactively change the design overwrites and immediately view the results.	No form; results are displayed onscreen.

TABLE 12-5 Composite Column Design Commands

Command	Action	Form
<b>Display Design Info</b>	Allows review of some of the results of the composite column design directly on the program model. Examples of results that can be displayed include column labels and associated design group names.	Display Composite Column Design Results form
<b>Make Auto Select Section Null</b>	Removes auto select section lists from selected columns. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual column sections assigned, not auto select sections. Only works on a user-specified selection.	Warning message Cannot use Undo
<b>Change Design Section</b>	Allows the user to change the design section property assigned to one or more columns and then rerun the design without first rerunning the analysis. Only works on a user-specified selection.	Select Sections form
<b>Reset Design Section to Last Analysis</b>	Sets the design section for one or more columns back to the last used analysis section. Only works on a user-specified selection.	Immediate Cannot use Undo
<b>Verify Analysis vs Design Section</b>	Verifies that the last used analysis section and the current design section are the same for all composite columns in the model.	Immediate
<b>Verify All Members Passed</b>	Reports if structural members have passed the stress/capacity check. An analysis and a design/check of the structure must be completed before this command is used.	Immediate
<b>Reset All Overwrites</b>	Resets the overwrites for all composite columns with the Composite Column design procedure to their default values.	Warning message Cannot use Undo
<b>Delete Design Results</b>	Deletes all of the composite column design results but not the current design section (i.e., next analysis section).	Immediate Cannot use Undo

**TABLE 12-6 Steel Joist Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>View/Revise Preferences</b>	Designates the design code to use, as well as defining other design parameters, such as camber. Default values are provided for all settings.	Steel Joist Design Preferences form
<b>View/Revise Overwrites</b>	Allows review of overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the joists to which they are specifically assigned.	Steel Joist Design Overwrites
<b>Lateral Bracing</b>	Designates lateral bracing as either program determined or user specified.	Lateral Bracing
<b>Select Design Groups</b>	Designates that a group is to be used as a design group. Works only when auto select sections have been assigned to the joists. When grouped, all objects in the group are given the same joist size.	Steel Joist Design Group Selection form
<b>Select Design Combinations</b>	Allows review of the default steel joist design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations during design.	Design Load Combinations Selection form
<b>Start Design/Check</b>	Initiates design process. If frame objects have been selected before this command is used, only the selected frame objects will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Interactive Design</b>	Allows the user to review the design results for any steel joist and then to interactively change the design overwrites and immediately view the results.	No form; results are displayed on-screen.
<b>Display Design Info</b>	Allows review of some of the results of the steel joist design directly on the program model. Examples of results that can be displayed include joist labels and associated design group names; design sections together with end reactions; and design ratio information.	Display Steel Joist Design Info form

TABLE 12-6 Steel Joist Design Commands

Command	Action	Form
<b>Make Auto Select Section Null</b>	Removes auto select section lists from selected joists. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual joist sections assigned, not auto select sections. Only works on a user-specified selection.	Warning message <i>Cannot use Undo</i>
<b>Change Design Section</b>	Allows the user to change the design section property assigned to one or more joists and then rerun the design without first rerunning the analysis. Only works on a user-specified selection.	Select Sections form
<b>Reset Design Section to Last Analysis</b>	Sets the design section for one or more joists back to the last used analysis section. Only works on a user-specified selection.	Immediate <i>Cannot use Undo</i>
<b>Verify Analysis vs Design Section</b>	Verifies that the last used analysis section and the current design section are the same for all steel joists in the model.	Immediate
<b>Verify All Members Passed</b>	Reports if all steel joists have passed the stress/capacity check. An analysis and a design/check of the structure must be completed before this command is used.	Immediate
<b>Reset All Overwrites</b>	Resets the overwrites for all steel joists with the Steel Joist design procedure to their default values.	Warning message <i>Cannot use Undo</i>
<b>Delete Design Results</b>	Deletes all of the steel joist design results but not the current design section (i.e., next analysis section).	Immediate <i>Cannot use Undo</i>

TABLE 12-7 Shear Wall Design Commands

Command	Action	Form
<b>View/Revise Preferences</b>	Designates the design code to use, as well as defining other design parameters. Default values are provided for all settings.	Shear Wall Design Preferences form

**TABLE 12-7 Shear Wall Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>Define General Pier Sections</b>	Allows the user to define a pier section using the Section Designer utility.	Pier Sections form that access other forms
<b>Assign Pier Sections</b>	Allows the user to assign a pier one of three section types.	Assign Pier Sections form
<b>View/Revise Pier Overwrites</b>	Allows review of pier overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the piers to which they are specifically assigned.	Wall Pier Design Overwrites form
<b>View/Revise Spandrel Overwrites</b>	Allows review of spandrel overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to the spandrels to which they are specifically assigned.	Wall Spandrel Design Overwrites form
<b>Select Design Combinations</b>	Allows review of the default shear wall design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations during design.	Design Load Combinations Selection form
<b>Start Design/Check</b>	Initiates design process. If piers or spandrels have been selected before this command is used, only the selected piers or spandrels will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Interactive Design</b>	Allows the user to review the design results for any piers or spandrels and then to interactively change the design overwrites and immediately view the results.	No form; results are displayed onscreen.
<b>Display Design Info</b>	Allows review of some of the results of the shear wall design directly on the program model. Examples of results that can be displayed include reinforcing requirements, capacity ratios and boundary element requirements.	Display Shear Wall Design Results form

TABLE 12-7 Shear Wall Design Commands

Command	Action	Form
<b>Make Auto Select Section Null</b>	Removes auto select section lists from selected shear walls. Typically used near the end of the iterative design process so that the final design iteration is performed using the actual shear wall sections assigned, not auto select sections. Only works on a user-specified selection.	Warning message <i>Cannot use Undo</i>
<b>Verify Analysis vs Design Section</b>	Verifies that the last used analysis section and the current design section are the same for all shear walls in the model.	Immediate
<b>Reset All Overwrites</b>	Resets the overwrites for all piers or spandrels to their default values.	Warning message <i>Cannot use Undo</i>
<b>Delete Design Results</b>	Deletes all of the shear wall results.	Immediate <i>Cannot use Undo</i>

TABLE 12-8 Concrete Slab Design Commands

Command	Action	Form
<b>View/Revise Preferences</b>	Designates the design code to use, as well as defining other design parameters. Default values are provided for all settings.	Concrete Slab Design Preferences form
<b>View/Revise Flexural Design Overwrites</b>	Allows review of flexural design overwrites, which are parameters that the user specifies to change program defaults. Overwrites may be applied to selected strip based, or FEM based design.	Slab Design Overwrites form
<b>View/Revise Punching Check Overwrites</b>	Allows review of punching check overwrites, which are parameters that the user specifies to change program defaults. Overwrites apply only to locations to which they are specifically assigned.	Punching Shear Design Overwrites form
<b>Select Design Combinations</b>	Allows review of the default concrete slab design combinations defined by the program, or designation of user-specified design combinations. Facilitates review or modification of combinations.	Design Load Combinations Selection form
<b>Select Stories for Design</b>	Allows selection of stories for which the concrete slab design should be performed.	Select Stories for Slab Design form

**TABLE 12-8 Concrete Slab Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>Start Design</b>	Initiates design process. Only the selected stories will be designed. A building analysis must precede use of this command.	Immediate, no form used
<b>Display Flexural Design</b>	Allows review of some of the results of the concrete slab design directly on the program model. Examples of results that can be displayed include flexural reinforcing requirements in the design strips as rebar area or number of bars.	Slab Design form
<b>Display Punching Check</b>	Allows review of the punching checks directly on the program model.	Immediate
<b>Display Crack Widths</b>	Allows review of crack widths directly on either the bottom or top face of the slab.	Crack Width form
<b>Reset All Flexural Design Overwrites</b>	Resets the flexural design overwrites to their default values for one of the two design methods selected.	Warning message <i>Cannot use Undo</i>
<b>Reset All Punching Overwrites</b>	Resets the punching overwrites for slab design to their default values.	Warning message <i>Cannot use Undo</i>
<b>Delete Design Results</b>	Deletes all of the concrete slab design results.	Immediate <i>Cannot use Undo</i>

In addition to the commands specific to each type of design, the **Design** menu contains several other more general commands described in table 12-9.

**TABLE 12-9 Miscellaneous Design Commands**

<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>Overwrite Frame Design Procedure</b>	Allows the user to change the default design procedure for a selected frame object, including specifying that no design should be performed.	Overwrite Frame Design Procedure form
<b>Live Load Reduction Factors</b>	Allows the user to adjust if and how much the live load should be reduced.	Live Load Reduction Factor form



TABLE 12-9 Miscellaneous Design Commands

Command	Action	Form
<b>Set Lateral Displacement Targets</b>	Specifies displacement targets, in any direction, for various load cases. This command is available only for steel frames, concrete frames, and concrete shear walls.	Lateral Displacement Targets form
<b>Set Time Period Targets</b>	Specifies time period targets for seismic analysis. This command is available only for steel frames, concrete frames, and concrete shear walls.	Time Period Targets form

There are also several commands under the **Design** menu which aid in specifying how the reinforcing calculated by ETABS is to be used in external detailing applications as described in table 12-10.

TABLE 12-10 Detailing Commands

Command	Action	Form
<b>Rebar Selection Rules</b>	Allows the user to change the default size and number of rebar, rebar spacing, and other parameters needed to detail slabs, beams, columns, piers, and spandrels.	Rebar Selection Rules form
<b>Calculate Rebars for Detailing</b>	Calculates the rebar for detailing based on the rebar selection rules specified.	Immediate, no form used
<b>Clear Rebars Calculation</b>	Clears the rebar calculated by the Calculate Rebars for Detailing command.	Immediate, no form used

# Display Results

## Objective

This chapter describes how to display analysis results graphically.

## Obtain Basic Graphical Displays

Analysis results can be displayed graphically after the analysis has been run. To display results, click the Display menu and select the type of display desired. Table 13-1 identifies the display options.

**TABLE 13-1 Display Menu Options**



<b>Command</b>	<b>Action</b>	<b>Form</b>
<b>Undeformed Shape</b>	Undeformed Shape  plots the undeformed shape onscreen.	No form; results are displayed.
<b>Load Assigns</b>		
> Joint	Displays loads assigned to Joints.	Show Joint Loads form
> Frame	Displays loads assigned to Frames.	Show Frame Loads form
> Shell	Displays loads assigned to Shells.	Show Shell Loads form
> Tendon	Displays loads assigned to Tendons.	Show Tendon Loads form
<b>Deformed Shape</b>	Deformed Shape  plots a deformed or mode shape onscreen based on user-specified loads. This plot can be animated.	Deformed Shape form
<b>Force/ Stress Diagrams</b>		
> Support/Spring Reactions	Displays support and spring reactions onscreen based on user-specified loads.	Reactions form
> Soil Pressure	Displays soil pressures onscreen based on user-specified loads.	Soil Pressure form
> Frame/Pier/ Spandrel/Link Forces	Displays column, beam, brace, pier, spandrel, and link forces onscreen based on user-specified loads.	Member Force Diagram for Frames/Piers/ Spandrels/Links form
> Shell Stresses/ Forces	Displays internal shell element forces and stresses onscreen based on user-specified loads.	Shell Forces/Stresses form
> Strip Forces	Displays forces in the design strips onscreen based on user-specified loads.	Strip Forces form
> Diaphragm Forces	Displays diaphragm forces onscreen based on user-specified loads.	Diaphragm Forces form

TABLE 13-1 Display Menu Options

Command	Action	Form
<b>Display Performance Check</b>	Display Performance Check displays demand/capacity (D/C) ratios for hinges after a nonlinear time history analysis has been run if performance checks have previously been defined.	Display Performance Check form
<b>Display Performance Check Usage Ratio Diagram</b>	Display Performance Check Usage Ratio Diagram displays a diagram of demand/capacity (D/C) ratios for hinges and other components plotted against demand sets after a nonlinear analysis has been run.	Immediate, no form used
<b>Energy/Virtual Work Diagram</b>	Energy/Virtual Work Diagram displays energy/virtual work diagrams that can be used as an aid to determine which elements should be stiffened to most efficiently control the lateral displacements of the structure. User defines forces and displacements	Energy/Virtual Work Diagram form
<b>Cumulative Energy Components</b>	Cumulative Energy Components plots all energy components in a cumulative manner after a modal time history analysis has been run.	Immediate, no form used
<b>Story Response Plots</b>	Story Response Plots displays force and displacement responses for specified stories as a new tab in a display window.	Immediate, no form used
<b>Combined Story Response Plots</b>	Combined Story Response Plots displays min/max displacements/accelerations, drifts, shears, and overturning.	Combined Story Response form
<b>Response Spectrum Curves</b>	Response Spectrum Curves plots various response spectra <b><i>after a time history analysis has been run.</i></b>	Response Spectrum Generation form
<b>Plot Functions</b>	Plot Functions plots various time history curves based on user-specified data after a time history analysis has been run.	Immediate, no form used
<b>Quick Hysteresis</b>		
> Links	Plots force-deformation hysteresis loops for link objects after a time history analysis has been run, as a new tab in a display window.	Immediate, no form used

TABLE 13-1 Display Menu Options

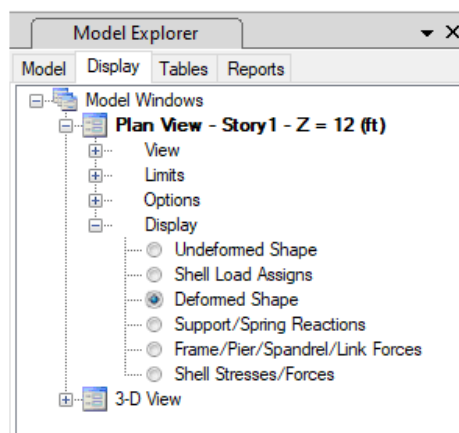
Command	Action	Form
<b>Static Pushover Curve</b>	Static Pushover Curve displays various pushover curves based on user specified data after a static nonlinear analysis has been run.	Pushover Curve form
<b>Hinge Results</b>	Hinge Results allows the user to plot rotations and deformations vs forces for a selected hinge after a static nonlinear analysis has been run.	Hinge Results form
<b>Save Named Display</b>	Save Named Display allows the user to save the display showing in the active window.	Named Displays form
<b>Show Named Display</b>	Displays a previously saved named display.	Select View form
<b>Show Tables</b>	Show Tables allows the user to select the type of information to display in table format.	Choose Tables form

\* **Note:** With a form displayed on the ETABS window, click the F1 key on your keyboard to access context-sensitive Help for the form.

## Graphical Displays using Model Explorer

A limited number of analysis results can also be shown graphically using the Display tab in the Model Explorer. Click on the View node where results are to be displayed so that the Display branch is visible. Click on the Display branch and then on the desired analysis result for display. The Model Explorer should look something like that shown in Figure 13-1. The view may be changed by clicking on the View branch and selecting from 3D, Plan or Elevation.

**Figure 13-1 Model Explorer Display tab**



## **Tabular Display of Results**

Both the analysis and design results can also be displayed in tabular format in ETABS by using the Model Explorer or the **Display menu > Show Tables** command. In the Model Explorer, select the Tables tab and then expand the tree by clicking on nodes to reveal various input, results, and design data. A right click on a branch or leaf of the tree will display an associated context sensitive menu. Click the **Show Table(s)** command on these menus to display a form with the tables.

# Generate Results

## Objective

This chapter describes how to output analysis and design results for further post-processing, presentations, or project submittals.

## Summary Report

A summary report is available at the click of a button using the **File menu > Create Report > Show Project Report** command. This summary report is automatically created by ETABS, is compatible with Microsoft Word, and can contain the following items, depending on the make-up of the model:

- Title Page
- Hyperlinked Table of Contents
- Model Definition Data

- Analysis Results
- Summary Design Results

## Print Graphics

Graphic displays also can be printed directly to a printer or captured to various file formats.

The display in the currently active window can be printed directly to the printer using the **File menu > Print Graphics** command. This will provide a print preview form that allows for adding text or graphical annotations before going to the printer. Graphic displays also may be captured to a file using the **File menu > Capture Picture** commands. There are different options for defining the region to be captured and the image file type.

## Export Results

Analysis and design results also can be exported from ETABS for further post-processing or use in other applications. The **File menu > Export** command has numerous options for exporting results.

A display showing results can be exported to a DXF/DWG file compatible with CAD applications. Tabular data can be exported to either Microsoft Excel (\*.xlsx) or Microsoft Access (\*.accdb). When exporting tabular data, choose the classes of data to export and the types of data within each class; this, in turn, determines the tables that are exported. Individual stories may be exported to CSI's SAFE® program for further analysis, design, and detailing. Models containing design results can also be sent out to Revit Structure, to a CIS/2 Step file, or to an IFC file.