



Welcome to ETABS® 2016

Integrated Building Design Software

Copyright

Copyright © Computers & Structures, Inc., 1978-2016
All rights reserved.

The CSI Logo[®], SAP2000[®], ETABS[®], and SAFE[®] are registered trademarks of Computers & Structures, Inc. Watch & Learn[™] is a trademark of Computers & Structures, Inc. Windows[®] is a registered trademark of the Microsoft Corporation. Adobe[®] and Acrobat[®] are registered trademarks of Adobe Systems Incorporated.

The computer programs SAP2000[®] and ETABS[®] and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers & Structures, Inc. Unlicensed use of these programs or reproduction of documentation in any form, without prior written authorization from Computers & 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 & Structures, Inc.
<http://www.csiamerica.com/>

info@csiamerica.com (for general information)
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.

Welcome to ETABS

1	Introduction	
	History and Advantages of ETABS	1-1
	What ETABS Can Do!	1-3
	An Integrated Approach	1-4
	Modeling Features	1-6
	Analysis Features	1-7
	Design Features	1-8
	Detailing Features	1-9
2	Getting Started	
	Installing ETABS	2-1
	If You are Upgrading	2-1
	About the Manuals	2-2
	“Watch & Learn”™ Movies	2-2
	CSI Knowledge Base	2-3
	Technical Support	2-3
	Help Us to Help You	2-3
3	The ETABS System	
	Overview of the Modeling Process	3-1

Physical Modeling Terminology	3-2
Story Definition	3-3
Towers	3-4
Units	3-4
Coordinate Systems and Grids	3-5
Structural Objects	3-5
Groups	3-7
Properties	3-7
Nonlinear Hinges	3-8
Load Patterns	3-9
Vertical Loads	3-9
Temperature Loads	3-10
Automated Lateral Loads	3-10
Functions	3-12
P-Delta	3-12
Modal Cases	3-13
Load Cases	3-13
Load Combinations	3-15
Design Settings	3-15
Detailing	3-17
Output and Display Options	3-18
More Information	3-18
4 ETABS Modeling Techniques	
Auto-Select Properties	4-1
Vertical Load Transfer	4-2
Shell Load Sets	4-3
Wind and Seismic Lateral Loads	4-4

Performance Based Design	4-4
Panel Zone Modeling	4-5
Wall Stacks	4-5
Live Load Reduction	4-6
Rigid and Semi-Rigid Floor Models	4-6
Post-Tensioning	4-7
Edge Constraints	4-8
Cladding	4-9
Soil-Foundation Interaction	4-9
Modifiers	4-9
Construction Sequence Loading	4-10
Walking Vibrations	4-10
Design and Drift Optimization	4-11
Section Designer	4-11
More Information	4-12

5 ETABS Analysis Techniques

Linear Static Analysis	5-1
P-Delta Analysis	5-2
Nonlinear Static Analysis	5-3
Modal Analysis	5-3
Mass Source	5-4
Eigenvector Analysis	5-4
Ritz-Vector Analysis	5-5
Response Spectrum Analysis	5-6
Linear Time History Analysis	5-6
Nonlinear Time History Analysis	5-7
Buckling Analysis	5-8

Welcome to ETABS

Hyperstatic Analysis	5-8
More Information	5-8

Introduction

ETABS is a sophisticated, yet easy to use, special purpose analysis and design program developed specifically for building systems. ETABS 2016 features an intuitive and powerful graphical interface coupled with unmatched modeling, analytical, design, and detailing procedures, all integrated using a common database. Although quick and easy for simple structures, ETABS can also handle the largest and most complex building models, including a wide range of nonlinear behaviors necessary for performance based design, making it the tool of choice for structural engineers in the building industry.

History and Advantages of ETABS

Dating back more than 40 years to the original development of TABS, the predecessor of ETABS, it was clearly recognized that buildings constituted a very special class of structures. Early releases of ETABS provided input, output and numerical solution techniques that took into consideration the characteristics unique to building type structures, provid-

ing a tool that offered significant savings in time and increased accuracy over general purpose programs.

As computers and computer interfaces evolved, ETABS added computationally complex analytical options such as dynamic nonlinear behavior, and powerful CAD-like drawing tools in a graphical and object-based interface. Although ETABS 2016 looks radically different from its predecessors of 40 years ago, its mission remains the same: to provide the profession with the most efficient and comprehensive software for the analysis and design of buildings. To that end, the current release follows the same philosophical approach put forward by the original programs, namely:

- Most buildings are of straightforward geometry with horizontal beams and vertical columns. Although any building configuration is possible with ETABS, in most cases, a simple grid system defined by horizontal floors and vertical column lines can establish building geometry with minimal effort.
- Many of the floor levels in buildings are similar. This commonality can be used to dramatically reduce modeling and design time.
- The input and output conventions used correspond to common building terminology. With ETABS, the models are defined logically floor-by-floor, column-by-column, bay-by-bay and wall-by-wall and not as a stream of non-descript nodes and elements as in general purpose programs. Thus the structural definition is simple, concise and meaningful.
- In most buildings, the dimensions of the members are large in relation to the bay widths and story heights. Those dimensions have a significant effect on the stiffness of the frame. ETABS corrects for such effects in the formulation of the member stiffness, unlike most general-purpose programs that work on centerline-to-centerline dimensions.
- The results produced by the programs should be in a form directly usable by the engineer. General-purpose computer programs

produce results in a general form that may need additional processing before they are usable in structural design.

What ETABS Can Do!

ETABS offers the widest assortment of analysis and design tools available for the structural engineer working on building structures. The following list represents just a portion of the types of systems and analyses that ETABS can handle easily:

- Multi-story commercial, government and health care facilities
- Parking garages with circular and linear ramps
- Buildings with curved beams, walls and floor edges
- Buildings with steel, concrete, composite or joist floor framing
- Projects with multiple towers
- Complex shear walls and cores with arbitrary openings
- Performance based design utilizing nonlinear dynamic analyses
- Buildings based on multiple rectangular and/or cylindrical grid systems
- Flat and waffle slab concrete buildings
- Buildings subjected to any number of vertical and lateral load cases and combinations, including automated wind and seismic loads
- Multiple response spectrum load cases, with built-in input curves
- Automated transfer of vertical loads on floors to beams and walls
- Capacity check of beam-to-column and beam-to-beam steel connections
- P-Delta analysis with static or dynamic analysis

- Explicit panel-zone deformations
- Punching shear checks for concrete slabs
- Construction sequence loading analysis
- Multiple linear and nonlinear time history load cases in any direction
- Foundation/support settlement
- Large displacement analyses
- Nonlinear static pushover
- Buildings with base isolators and dampers
- Design optimization for steel and concrete frames
- Design of concrete slabs using mild reinforcement and post-tensioning
- Design capacity check of steel column base plates
- Floor modeling with rigid or semi-rigid diaphragms
- Automated vertical live load reductions

And much, much more!

An Integrated Approach

ETABS is a completely integrated system. Embedded beneath the simple, intuitive user interface are very powerful numerical methods, design procedures and international design codes, all working from a single comprehensive database. This integration means that you create only one model of the floor systems and the vertical and lateral framing systems to analyze, design, and detail the entire building.

Everything you need is integrated into one versatile analysis and design package with one Windows-based graphical user interface. No external modules are required. The effects on one part of the structure from

changes in another part are instantaneous and automatic. The integrated components include:

- Drafting for model generation
- Seismic and wind load generation
- Gravity load distribution for the distribution of vertical loads to columns and beams when plate bending floor elements are not provided as a part of the floor system
- Finite element-based linear static and dynamic analysis
- Finite element-based nonlinear static and dynamic analysis (available in ETABS Nonlinear & Ultimate versions only)
- Output display and report generation
- Steel frame design (column, beam and brace)
- Concrete frame design (column and beam)
- Concrete slab design
- Composite beam design
- Composite column design
- Steel joist design
- Shear wall design
- Steel connection design including column base plates
- Detail schematic drawing generation

ETABS 2016 is available in three different levels that all share the same graphical user interface:

- **ETABS 2016 Plus.** Includes all available capabilities except for some design and certain nonlinear and dynamic analyses (p-delta and tension/compression only frame members are provided in *all* versions). Features include unmatched solution capacity with 64-

bit optimized solvers, shear wall modeling, multiple response spectrum analyses, linear modal time histories, numerous import and export options, and comprehensive report generation. The steel frame design, concrete frame design, composite beam design, composite column design, steel joist design, shear wall design, steel connection design and steel base plate design components are all present.

- **ETABS 2016 Nonlinear.** Includes all of the features of ETABS 2016 Plus, with additional nonlinear static and dynamic capabilities such as pushover, base isolation and dampers using Fast Nonlinear Analysis (FNA), Staged Construction, and multi-linear P-y springs.
- **ETABS 2016 Ultimate.** Includes all of the features of ETABS 2016 Nonlinear with additional features such as concrete slab design with post-tensioning, nonlinear layered shell elements, dynamic analysis utilizing nonlinear frame and wall hinges, linear and nonlinear direct integration time history analysis, buckling, and the modeling of creep and shrinkage behavior.

Modeling Features

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

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

Semi-rigid floor diaphragms may be modeled to capture the effects of in-plane floor deformations. Floor objects may span between adjacent levels

to create sloped floors (ramps), which can be useful for modeling parking garage structures.

Modeling of partial diaphragms, such as in mezzanines, setbacks, atriums and floor openings, is possible without the use of artificial (“dummy”) floors and column lines. It is also possible to model situations with multiple independent diaphragms at each level, allowing the modeling of buildings consisting of several towers rising from a common base.

The column, beam and brace elements may be non-prismatic, and they may have partial fixity at their end connections. They also may have uniform, partial uniform and trapezoidal load patterns, and they may have temperature loads. The effects of the finite dimensions of the beams and columns on the stiffness of a frame system are included using end offsets that can be automatically calculated.

The floors and walls can be modeled as membrane elements with in-plane stiffness only, or full shell-type elements, which combine both in-plane and out-of-plane stiffness. Floor and wall members may have uniform and non-uniform load patterns in-plane or out-of-plane, and they may have temperature loads. The column, beam, brace, floor and wall members are all compatible with one another.

Analysis Features

Static analyses for user specified vertical and lateral floor or story loads are possible. If floors with out-of-plane bending capability are modeled, vertical loads on the floor are transferred to the beams and columns through bending of the floor elements. Otherwise, vertical loads on the floor are automatically converted to span loads on adjoining beams, or point loads on adjacent columns, thereby automating the tedious task of transferring floor tributary loads to the floor beams without the need to explicitly model the secondary framing.

The program can automatically generate lateral wind and seismic load patterns to meet the requirements of various building codes. Three-dimensional mode shapes and frequencies, modal participation factors, direction factors and participating mass percentages are evaluated using

eigenvector or ritz-vector analysis. P-Delta effects may be included with static or dynamic analysis.

Response spectrum analysis, linear time history analysis, nonlinear time history analysis, and static nonlinear (pushover) analysis are all possible. The static nonlinear capabilities also allow you to perform incremental construction analysis so that forces that arise as a result of the construction sequence are included.

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

Output may be viewed graphically, displayed in tabular output, compiled in a report, exported to a database file, or saved in an ASCII file. Types of output include reactions and member forces, mode shapes and participation factors, static and dynamic story displacements and story shears, inter-story drifts and joint displacements, time history traces, and more.

Import and export of data may occur between third-party applications such as Revit and AutoCAD from Autodesk, or with other programs that support the CIS/2 or IFC data models.

ETABS uses the SAPFire™ analysis engine, the state-of-the-art equation solver that powers all of CSI's software. This proprietary solver exploits the latest in numerical technology to provide incredibly rapid solution times and virtually limitless model capacity.

Design Features

Design of steel frames, concrete frames, concrete slabs, concrete shear walls, composite beams, composite columns, and steel joists can be performed based on a variety of US and International design codes. Flexural, shear and deflection checks may all be performed depending upon the material and member type. Steel and concrete frame members may be optimized from autoselect lists, and concrete sections are designed using reinforcing bar sizes chosen from US or International standards. Concrete slab design may be done using either design strips, or be based on the finite element method, and may include the effects of post-

tensioning. Steel connection design automates the review of beam-beam and beam-column connections based on user specified bolt and shear plate preferences. Steel base plate design verifies the size, thickness, and anchorage of the connection.

Detailing Features

Schematic construction drawings showing floor framing, column schedules, beam elevations and sections, steel connection schedules, and concrete shear wall reinforcing may be produced. Concrete reinforcement of beams, columns, and walls may be selected based on user-defined rules. Any number of drawings may be created, containing general notes, plan views, sections, elevations, tables, and schedules. Drawings may be printed directly from ETABS or exported to DXF or DWG files for further refinement.

Getting Started

ETABS is an easy to use, yet extremely powerful, special purpose program developed expressly for building systems. This chapter will help you get started using the program.

Installing ETABS

Please follow the installation instructions provided in the separate installation document included in your ETABS Package, or ask your system administrator to install the program and give you access to it.

If You are Upgrading

If you are upgrading from an early version of ETABS, be aware that the model is now defined in terms of objects, which are automatically and internally meshed into elements during analysis.

This significant change drastically improves the capability of the program, and we recommend that you read the remainder of this manual to familiarize yourself with this and the many other new features.

About the Manuals

This volume is designed to help you quickly become productive with ETABS. It provides this *Welcome to ETABS* manual, a *User's Guide* and an *Introductory Tutorial*. The next chapter of this *Welcome to ETABS* manual provides a synopsis of the terminology used in ETABS, and Chapters 4 and 5 describe modeling and analysis techniques, respectively. The 15 chapters of the *User's Guide* provide an introduction to the steps and menu items used to create, analyze, design and detail a model. The *Introductory Tutorial* describes the model creation, analysis, and design processes for an example model.

It is strongly recommended that you read this manual and view the tutorial movies (see “Watch & Learn”™ Movies) before attempting to complete a project using ETABS.

Additional information can be found in the on-line Help facility available within the ETABS graphical user interface, including manuals that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS DVD, and can be accessed from within the program using the Help menu.

“Watch & Learn”™ Movies

One of the best resources available for learning about the ETABS program is the “Watch & Learn” movies series, which may be accessed via the CSI website at <https://www.csiamerica.com>. These movies contain a wealth of information for both the first-time user and the experienced expert, covering a wide range of topics, from basic operation to complex modeling. The movies range from a few minutes to more than a half hour in length.

CSI Knowledge Base

CSI maintains a knowledge base containing answers to frequently asked support questions as well as additional insights on program operation. This is a good first stop before contacting technical support because many of the most common, as well as some esoteric questions are answered here. This page is fully indexed and searchable, and may be found at <https://wiki.csiamerica.com>.

Technical Support

If you have questions regarding use of the software, please:

- Consult the documentation and other printed information included with your product.
- Check the on-line Help facility in the software.
- Visit the CSI Knowledge Base at <https://wiki.csiamerica.com>.

If you have a current Maintenance Agreement you may request support in one of the following ways:

- Send an email and your model file to support@csiamerica.com or your local CSI Partner.
- Visit CSI's website and Customer Support Portal at <https://www.csiamerica.com>.
- Call CSI or your local CSI Partner. Contact details are available at <https://www.csiamerica.com/contact>.

Be sure to include the necessary information listed in the *'Help Us to Help You'* section whenever you contact technical support.

Help Us to Help You

Whenever you contact us with a technical support question, please provide us with the following information to help us help you:

- The product level (Plus, Nonlinear or Ultimate) and version number that you are using. This can be obtained from inside the software using the **Help menu > About ETABS** command.
- A description of your model, including a picture, if possible.
- A description of what happened and what you were doing when the problem occurred.
- The exact wording of any error messages that appeared on your screen.
- A description of how you tried to solve the problem.
- The computer configuration (make and model, processor, operating system, hard disk size, and RAM size).
- Your name, your company's name, and how we may contact you.
- If calling, please be at your computer where you can run the software.

The ETABS System

ETABS analyzes, designs and details your building structure using a model that you create using the graphical user interface. The key to successfully implementing ETABS is to understand the unique and powerful approach the program takes in modeling building systems. This chapter will provide an overview of some of the key components and their associated terminology.

Overview of the Modeling Process

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

- This program is optimized for modeling building systems. Thus, the modeling procedures and design capabilities are all tailored to buildings.
- This program's model is object-based. It consists of joint, frame, tendon, link, and shell objects. You make assignments to those objects to define structural members such as beams, columns,

braces, floors, walls, ramps and springs. You also make assignments to those same objects to define loads.

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

- Draw a series of joint, frame, tendon, link, and shell objects that represent your building using the various drawing tools available within the graphical interface. Design strips may also be drawn to aid in adding tendons to the building model.
- Assign structural properties (sections and materials) and loads to objects using the Assign menu options. Note that the assignment of structural properties may be completed concurrently with the drawing of the object using the Properties of Object form that displays when Draw commands are used.
- Verify meshing parameters for floor (if they are not membrane slab/deck/plank sections) and wall shell objects. Maximum mesh size may be set independently for floors and walls.

When the model is complete, the analysis may be run. At that time, the program automatically converts the object-based model into an element-based model—this is known as the analysis model—that is used for the analysis. The analysis model consists of joints, frame elements, tendon elements, link elements, and shell (membrane and plate) elements that mathematically represent the structural members, i.e., columns, beams, braces, walls, floors, etc. The conversion to the analysis model is internal to the program and essentially transparent to the user.

Physical Modeling Terminology

In ETABS, we often refer to Objects, Members, and Elements. *Objects* represent the physical structural *members* in the model. *Elements*, on the other hand, refer to the finite elements used internally by the program to generate the stiffness matrices. In many cases, objects and physical members will have a one-to-one correspondence, and it is these objects that the user “draws” in the ETABS interface. Objects are intended to be an accurate representation of the physical members. Users typically do not need to concern themselves with the meshing of those objects into

the elements required for the mathematical or analysis model. For instance, a single frame object can model a complete beam, regardless of how many other members frame into it, and regardless of the loading. With ETABS, model creation and the reporting of results are accomplished at the object level.

This differs from a traditional analysis program, where the user is required to define a sub-assemblage of finite elements that comprise the larger physical members. In ETABS, the objects, or physical members drawn by the user, are typically subdivided internally into the greater number of finite elements needed for the analysis model, without user input. Because the user is working only with the physical member based objects, less time is needed to create the model and to interpret the results, with the added benefit that analysis results are generally more appropriate for the design work that follows.

The concept of objects in a structural model may be new to you. It is extremely important that you grasp this concept because it is the basis for creating a model in ETABS. After you understand the concept and have worked with it for a while, you should recognize the simplicity of physical object-based modeling, the ease with which you can create models using objects, and the power of the concept when editing and creating complex models.

Story Definition

One of the most powerful features that ETABS offers is the recognition of story levels, allowing for the input of building data in a logical and convenient manner. Users may define their models on a floor-by-floor, story-by-story basis, analogous to the way a designer works when laying out building drawings. Story levels help identify, locate and view specific areas and objects of your model; column and beam objects are easily located using their plan location and story level labels.

In ETABS terminology, a story level represents a horizontal plane cut through a building at a specified elevation, and all of the objects below this plane down to the next story level. Because ETABS inherently understands the geometry of building systems, a user can specify that an

object being drawn be replicated at all stories, or at all similar stories as identified by the user. This option works not only for repetitive floor framing, but also for columns, tendons, and walls. Story labeling, the height of each story level, as well as the ability to mark a story as similar, are all under the control of the user.

Towers

ETABS allows multiple towers to be defined, each with their own grid system and story data. This allows for the modeling of multiple buildings on a common podium. Views may be set to display one tower, all towers, or any combination thereof.

Units

ETABS works with four basic units: force, length, temperature, and time. Any units may be used at any time while working on the model, e.g., inch units for beam sections and feet units for grid layout. Time is always measured in seconds.

Angular measure always uses the following units:

- Geometry, such as axis orientation, is always measured in degrees.
- Rotational displacements are always measured in radians.
- Frequency is always measured in cycles/second (Hz).

An important distinction is made between mass and weight. Mass is used for calculating dynamic inertia and for loads caused by ground acceleration only. Weight is a force that can be applied like any other force load. Be sure to use force units when specifying weight values, and mass units (force-sec²/length) when specifying mass values.

When you start a new model, you will be asked to initialize with either "U.S." or "Metric" base units. This option sets the default units for the model and controls which units are used internally to save the model - input and output are always converted to and from the internal base units.

Coordinate Systems and Grids

All locations in the model are ultimately defined with respect to a single global coordinate system. This is a three-dimensional, right-handed, Cartesian (rectangular) coordinate system. The three axes, denoted X, Y, and Z, are mutually perpendicular, and satisfy the right-hand rule.

ETABS always considers the +Z direction as upward. By default, gravity acts in the -Z direction.

Additional coordinate systems can be defined to aid in developing and viewing the model. For each coordinate system, a three-dimensional grid system would be defined consisting of “construction” lines that are used for locating objects in the model. Each coordinate/grid system may be of Cartesian (rectangular) or cylindrical definition, and is positioned relative to the global system. When you move a grid line, specify whether the objects in the model move with it.

Drawing operations tend to “snap” to gridline intersections (default) unless you turn this feature off. Numerous other snaps are available, including snap to line ends and midpoints, snap to intersections, and so forth. Use these powerful tools whenever possible to ensure the accurate construction of your model. Not using the snaps may result in “gaps” between objects, causing errors in the model’s connectivity.

Each object in the model has its own local coordinate system used to define properties, loads, and responses. The axes of each local coordinate system are denoted 1 (red), 2 (green), and 3 (blue). Local coordinate systems do not have an associated grid.

Structural Objects

As stated previously, ETABS uses objects to represent physical structural members. When creating a model, the user starts by drawing the geometry of the object, and then assigning properties and loads to completely define the building structure.

The following object types are available, listed in order of geometrical dimension:

- **Joint objects** of two types:
 - **Joint objects** are automatically created at the corners or ends of all other types of objects, and they can be explicitly added anywhere in the model.
 - **Spring or Grounded (one joint) link objects** are used to model external springs and special support behavior, such as isolators, dampers, gaps, multi-linear springs and more.
- **Frame objects** are used to model beams, columns, braces and trusses.
- **Tendon objects** are used to model post-tensioning tendons and tendon loads. Even though they are drawn in plan similar to a beam object, they differ from beam objects in that they have a shape profile through the slab thickness.
- **Link (connecting two-joint) objects** are used to model special member behavior, such as isolators, dampers, gaps, multi-linear springs, and more. Unlike frame objects, connecting link objects can have zero length.
- **Shell objects** are used to model walls, slabs, decks, planks, and other thin-walled members. Shell objects will be meshed automatically into the elements needed for analysis. Walls and floors with both membrane and plate behavior (e.g., cast-in-place solid, waffle, & ribbed slabs) are meshed using a rectangular mesh with a maximum size set by the user. If horizontal objects with only membrane definition are included in the model (e.g., decks and planks), the objects are meshed in a manner such that the vertical loads will be properly distributed to supporting members.

As a general rule, the geometry of the object should correspond to that of the physical member. This simplifies the visualization of the model and helps with the design process.

When you run an analysis, ETABS automatically converts your object-based model into an element-based model that is used for analysis. This element-based model is called the analysis model, and it consists of traditional finite elements and joints. After running the analysis, your object-based model still has the same number of objects in it as it did before the analysis was run.

Although the majority of the object meshing is performed automatically, you do have control over how the meshing is completed, such as the degree of refinement and how to handle the connections at intersecting objects. An option is also available to manually subdivide the model, which divides an object based on a physical member into multiple objects that correspond in size and number to the analysis elements.

Groups

A group is a named collection of objects. It may contain any number of objects of any number of types. Groups have many uses, including:

- Quick selection of objects for editing and assigning.
- Defining section cuts across the model.
- Grouping objects that are to share the same design.
- Selective output.

Define as many groups as needed. Using groups is a powerful way to manage larger models.

Properties

Properties are “assigned” to each object to define the structural behavior of that object in the model. Some properties, such as materials and section properties, are named entities that must be specified before assigning them to objects. For example, a model may have:

- A material property named CONCRETE.

- A rectangular frame section property named RECTANGLE, and a circular frame section named CIRCULAR, both using material property CONCRETE.
- A slab section property named SLAB that also uses material property CONCRETE.

If you assign frame section property RECTANGLE to a frame object, any changes to the definition of section RECTANGLE or material CONCRETE will automatically apply to that object. A named property has no effect on the model unless it is assigned to an object.

Other properties, such as frame releases or joint restraints, are assigned directly to objects. Those properties can be changed only by making another assignment of that same property to the object; they are not named entities and they do not exist independently of the objects.

Nonlinear Hinges

Nonlinear hinges define the load-deformation (force-displacement, moment-rotation) behavior of objects post-yield. Hinges may be assigned to both frame objects and vertical wall objects. Built-in default or user-defined hinge properties may be specified at any number of locations along frame objects, and at the mid-height of wall objects. Nonlinear hinges are processed in the analysis model in one of two ways: 1) the hinge is integral (internal) within the object - these hinges can be used in static and direct integration nonlinear analyses; or 2) the hinge is modeled using an additional link element - these hinges may be used in all nonlinear analyses, including the often faster and more stable Fast Nonlinear Analysis (FNA). There may be slight differences in results between these two options due to minor variations in flexibility and inertia.

The number of hinges affects not only computation time, but also the ease in which model behavior and results may be interpreted. Therefore, it is strongly recommended that hinges be assigned only at locations where the occurrence of nonlinear behavior is highly probable.

Load Patterns

Loads represent actions upon the structure, such as force, pressure, support displacement, thermal effects, and others. A spatial distribution of loads upon the structure is called a load pattern.

As many named load patterns as needed can be defined. Typically, separate load patterns would be defined for dead load, live load, static earthquake load, wind load, snow load, thermal load, and so on. Loads that need to vary independently, for design purposes or because of how they are applied to the building, should be defined as separate load patterns.

After defining a load pattern name, you must assign specific load values to the objects as part of that load pattern, or define an automated lateral load if the case is for seismic or wind. The load values you assign to an object specify the type of load (e.g., force, displacement, temperature), its magnitude, and direction (if applicable). Different loads can be assigned to different objects as part of a single load pattern, along with the automated lateral load, if so desired. Each object can be subjected to multiple load patterns.

Vertical Loads

Vertical loads may be applied to joint, frame and shell objects. Vertical loads are typically input in the gravity, or -Z direction. Joint objects can accept concentrated forces or moments. Frame objects may have any number of point loads (forces or moments) or distributed loads (uniform or trapezoidal) applied. Uniform and non-uniform loads can be applied to Shell objects. Vertical load cases may also include element self-weight.

Some typical vertical load cases used for building structures might include:

- Dead load
- Superimposed dead load
- Live load
- Reduced live load

- Snow load

If the vertical loads applied are assigned to a reducible live load pattern, ETABS provides you with an option to reduce the live loads used in the design phase. Many different types of code-dependent load reduction formulations are available.

Temperature Loads

Temperature loads on frame and shell objects can be generated in ETABS by specifying temperature changes. Those temperature changes may be specified directly as a uniform temperature change on the object, or they may be based on previously specified point object temperature changes, or on a combination of both.

If the point object temperature change option is selected, the program assumes that the temperature change varies linearly over the object length for frames, and linearly over the object surface for shells. Although you can specify a temperature change for a joint object, temperature loads act only on frame and shell objects.

Automated Lateral Loads

ETABS allows for the automated generation of static lateral loads for earthquake (seismic) and wind load cases based on numerous code specifications, including, but not limited to, ASCE, BOCA, UBC, NBCC, AS, BS, NZS, Chinese, Italian, Mexican, Turkish and EUROCODE. Two automatic static lateral loads can not be in the same load pattern. Thus, define each automatic static lateral load in a separate load pattern. However, additional user-defined loads can be added to a load pattern that includes an auto lateral load.

When seismic has been selected as the load type, various auto lateral load codes are available. Upon selection of a code, the Seismic Load Pattern form is populated with default values and settings that may be reviewed and edited by the user. The program uses those values to generate lateral loads in the specified direction based on the weight defined by the masses assigned or calculated from the property definitions. After

ETABS has calculated a story level force for an automatic seismic load, that force is apportioned to each joint at the story level elevation in proportion to its mass. Seismic auto lateral loads should not be used with models that contain more than one tower as an incorrect distribution of lateral loads may occur.

If wind has been selected as the load type, various auto lateral load codes are available. Upon selection of a code, the Wind Load Pattern form is populated with default values and settings, which may be reviewed and edited by the user. In ETABS, automatically calculated wind loads may be applied to diaphragms (rigid or semi-rigid), or to walls and frames, including non-structural walls such as cladding that are created using shell objects, and to frames in open structures. If the rigid diaphragm option is selected, a separate load is calculated for each rigid diaphragm present at a story level. The wind loads calculated at any story level are based on the story level elevation, the story height above and below the level, the assumed exposure width for the rigid diaphragm(s) at that level and the various code-dependent wind coefficients. The load is applied to a rigid diaphragm at what ETABS calculates to be the geometric center.

Wind loads applied to semi-rigid diaphragms are calculated at each story level in a manner similar to that for the rigid diaphragms, but are then applied to every joint throughout the diaphragm. Although every joint in the diaphragm receives some load, the distribution of the forces is done in such a way that the resultant of the applied wind load passes through the diaphragm's geometric center.

If the option has been selected whereby wind loads are calculated and applied via shell objects defining walls, a wind pressure coefficient must be assigned to each shell object that has exposure, and it must be specified as windward or leeward. On the basis of the various code factors and user defined coefficients and exposures, ETABS calculates the wind loads for each shell (wall) object and applies the loads as point forces at the corners of the object. In addition, some codes allow wind loads to be generated for exposed frame members (i.e., lattice or open structures), in which case the program calculates joint forces based on the code selected and the solid to gross area ratio specified. Wind auto lateral loads should not be used with models that contain more than one tower as an incorrect distribution of lateral loads may occur.

Functions

Functions are defined to describe how a load varies as a function of period, time or frequency. Functions are only needed for certain types of analysis; they are not used for static analysis. A function is a series of digitized abscissa-ordinate data pairs.

There are two types of functions:

- **Response spectrum functions** are psuedo-spectral acceleration versus period functions for use in response spectrum analysis. In this program, the acceleration values in the function are assumed to be normalized; that is, the functions themselves are not assumed to have units. Instead, the units are associated with a scale factor that multiplies the function and is specified when you define the response spectrum case.
- **Time history functions** are loading magnitude versus time functions for use in time history analysis. The loading values in a time history function may be ground acceleration values or they may be multipliers for specified (force or displacement) load patterns.

Define as many named functions as necessary. They are not assigned to objects, but are used in the definition of Response Spectrum and Time History cases.

P-Delta

P-Delta options are set to determine the type of P-Delta to use. P-Delta refers to the nonlinear geometric effect that gravity loads have upon the lateral stiffness of buildings. There are three options:

- **None:** No P-Delta is included in the analysis.
- **Non-iterative:** An efficient (but approximate) P-Delta technique computed automatically from story mass.

- **Iterative:** A computationally robust approach based on a user specified combination of loads.

P-Delta may also be specified using a static nonlinear load case.

Modal Cases

A modal case defines the type and number of modes to be extracted from the model. An unlimited number of modal cases may be defined, although for most purposes a single case is enough. Each modal case results in a set of modes, and each mode consists of a mode shape (normalized deflected shape) and a set of modal properties, such as period and cyclic frequency. The dynamic modes of the structure are calculated using either eigenvector or Ritz-vector methods:

- **Eigenvector Analysis:** Determines the undamped free-vibration mode shapes and frequencies of the system, which provide an excellent insight into the behavior of the building.
- **Ritz-vector Analysis:** Modes are generated by taking into account the spatial distribution of the dynamic loading, which yields more accurate results than the use of the same number of natural mode shapes. Ritz-vector modes do not represent the intrinsic characteristics of the structure in the same way the natural (eigenvector) modes do.

A modal case is required in order to run either a response-spectrum or a modal time-history load case.

Load Cases

A load case defines how loads are to be applied to the structure, and how the structural response is to be calculated. Many types of load cases are available. Most broadly, load cases are classified as linear or nonlinear, depending on how the structure responds to the loading.

The results of linear analyses may be superposed, i.e., added together, after analysis. The following types of load cases are available:

- **Static:** The most common type of analysis. Loads are applied without dynamical effects.
- **Response-Spectrum:** Statistical calculation of the response caused by acceleration loads. Requires response-spectrum functions.
- **Time-History:** Time-varying loads are applied. Requires time-history functions. The solution may be by modal superposition or direct integration methods.
- **Buckling:** Calculation of buckling modes under the application of loads.

The results of nonlinear load cases normally should not be superposed. Instead, all loads acting together on the structure should be combined directly within the specific nonlinear load case. Nonlinear load cases may be chained together to represent complex loading sequences. The following types of nonlinear load cases are available:

- **Nonlinear Static:** Loads are applied without dynamical effects. May be used for pushover analysis.
- **Nonlinear Staged Construction:** Loads are applied without dynamical effects, with portions of the structure being added or removed. Time-dependent effects can be included, such as creep, shrinkage, and aging.
- **Nonlinear Time-History:** Time-varying loads are applied. Requires time-history functions. The solution may be by modal superposition or direct integration methods.

Any number of named load cases of any type may be defined. When the model is analyzed, the load cases to be run must be selected. Results for any load case may be selectively deleted.

Analysis results, when available, can be considered to be part of the model. They are needed to perform design.

Load Combinations

ETABS allows for the named combination of the results from one or more load cases and/or other combinations. When a combination is defined, it applies to the results for every object in the model.

The five types of combinations are as follows:

- **Linear Add:** Results from the included load cases and combinations are added.
- **Envelope:** Results from the included load cases and combinations are enveloped to find the maximum and minimum values.
- **Absolute Add:** The absolute values of the results from the included load cases and combinations are added.
- **SRSS:** The square root of the sum of the squares of the results from the included load cases and combinations is computed.
- **Range Add:** Positive values are added to the maximum and negative values are added to the minimum for the included load cases and combos.

Except for the Envelope type, combinations should usually be applied only to linear load cases, because nonlinear results are not generally superposable.

Design is always based on combinations, not directly on load cases. A combination can be created that contains only a single load case. Each design algorithm creates its own default combinations; supplement them with your own design combinations if needed.

Design Settings

ETABS offers the following integrated design postprocessors:

- Steel Frame Design
- Concrete Frame Design

- Composite Beam Design
- Composite Column Design
- Steel Joist Design
- Shear Wall Design
- Concrete Slab Design
- Steel Connection Design

The first five design procedures are applicable to frame objects, and the program determines the appropriate design procedure for a frame object when the analysis is run. The design procedure selected is based on the line object's orientation, section property, material type and connectivity.

Shear wall design is available for objects that have previously been identified as piers or spandrels, and both piers and spandrels may consist of both shell and frame objects.

Concrete slab design is applicable to slab objects and will include the effects of post-tensioning if present in the model. Punching shear checks are also carried out at column object locations, vertical point loads, and supports.

Steel connection design will identify which beam-to-beam and beam-to-column locations have adequate load transfer capacity using the standard connections specified in the connection preferences. Steel connection design also includes sizing and design capacity checks for column base plates.

For each of the first five design postprocessors, several settings can be adjusted to affect the design of the model:

- The specific design code to be used for each type of object, e.g., AISC 360-10 for steel frames, EUROCODE 2-2004 for concrete frames, and BS8110 97 for shear walls.
- Preferences for how these codes should be applied to a model.
- Combinations for which the design should be checked.

- Groups of objects that should share the same design.
- Optional “overwrite” values for each object that supersede the default coefficients and parameters used in the design code formulas selected by the program.

For steel and concrete frames, composite beam, composite column, and steel joist design, ETABS can automatically select an optimum section from a list you define. The section also can be changed manually during the design process. As a result, each frame object can have two different section properties associated with it:

- An “analysis section” used in the previous analysis
- A “design section” resulting from the current design

The design section becomes the analysis section for the next analysis, and the iterative analysis and design cycle should be continued until the two sections become the same.

Design results for the design section, when available, as well as all of the settings described herein, can be considered to be part of the model.

Detailing

ETABS offers the ability to produce schematic construction documents for buildings. Preferences may be set for the size and layout of drawings; dimensioning units and label prefixes; and reinforcing bar sizes for beams, columns and shear walls. Generated drawings, accessible on the Detailing tab of the Model Explorer window, can include:

- Cover Sheets
- General Notes
- Beam & Column Sections
- Floor Framing Plans
- Column Schedules

- Beam Schedules
- Connection Schedules
- Column Layout
- Wall Layout
- Wall Reinforcement Plans & Elevations

Output and Display Options

The ETABS model and the results of the analysis and design can be viewed and saved in many different ways, including:

- Two- and three-dimensional views of the model
- Customizable user defined reports
- Input/output data values in plain text, spreadsheet, or database format
- Function plots of analysis results
- Design sheets
- Story data for import into SAFE®
- Export to other drafting and design programs

Named definitions of display views and function plots can be saved as part of a model. Combined with the use of groups, this can significantly speed up the process of getting results as the modeling is being developed.

More Information

This chapter presents just a brief overview of some of the basic components of the ETABS model. Additional information can be found in the online Help facility available within the ETABS graphical user interface, including manuals that describe code-specific design algorithms. Those

documents are available in Adobe Acrobat PDF format on the ETABS DVD, and can be accessed from within the program using the Help menu.

ETABS Modeling Techniques

ETABS offers an extensive and diverse range of tools to help you model a wide range of building systems and behaviors. This chapter illustrates a few of the techniques that you can use with ETABS to make many mundane or complex tasks quick and easy.

Auto-Select Properties

When creating an ETABS model containing steel or concrete frame objects (frames, composite beams, and joists), determining explicit preliminary member sizes for analysis is not necessary. Instead, apply an auto-select section property to any or all of the frame objects. An auto-select property is a list of section sizes rather than a single size. The list contains all of the section sizes to be considered as possible candidates for the physical member, and multiple lists can be defined. For example, one auto-select list may be for steel columns, another list may be used for floor joists, and a third list may be used for steel beams and girders.

For the initial analysis, the program will select the median section in the auto-select list. After the analysis has been completed, run the design optimization process for a particular object where only the section sizes available in the auto-select section list will be considered, and the program will automatically select the most economical, adequate section from this list. After the design optimization phase has selected a section, the analysis model should be re-run if the design section differs from the previous analysis section. This cycle should be repeated until the analysis and design sections are identical.

Effective use of auto-select section properties can save many hours associated with establishing preliminary member sizes.

Vertical Load Transfer

ETABS offers powerful algorithms for the calculation of vertical load transfer. The vertical loads may be applied as dead loads, live loads, superimposed dead loads, and reducible live loads, with the self-weight of the objects included in the dead load pattern, if so desired.

The main issue for vertical load transfer is the distribution of surface loads that lie on the shell object representing the floor plate in the analysis model. ETABS analysis for vertical load transfer differs depending upon whether the floor-type object has membrane only behavior or plate bending behavior. The following bullets describe analysis for floor-type objects:

- **Out-of-plane load transformation for floor-type shell objects with deck or plank section properties:** In this case, in the analysis model, loads are transformed to beams along the edges of membrane elements or to the corner points of the membrane elements using a significant number of rules and conditions. The load transfer takes into account that the deck or plank spans in only **one** direction.
- **Out-of-plane load transformation for floor-type shell objects with slab section properties that have membrane behavior only:** In this case, in the analysis model, loads are transformed either to beams along the edges of membrane elements or to the

corner points of the membrane elements using a significant number of rules and conditions. The load transformation takes into account that the slab spans in **two** directions.

- **Out-of-plane load transformation for floor-type shell objects with slab (solid, waffle, & ribbed) section properties that have plate bending behavior:** In this case, the floor object is automatically meshed using rectangular finite elements into an analysis model based on a user-defined maximum element size (a general mesh using quadrilaterals and triangles is also an option). Meshing lines are automatically introduced at all locations of column objects, beam objects, and slab boundaries. Meshing lines can be introduced at specified locations by adding frame objects with NONE properties and then assigning them to be included in the analysis mesh. For floor objects that have plate bending behavior, ETABS uses a bilinear interpolation function to transfer loads on the floor to the corner points of the shell/plate elements in the analysis model.

The internal meshing for both membrane only and plate bending shell sections does not alter the number or size of the objects, which allows for simple revisions and modifications to be executed at the object level.

For the load transfers described herein, the program will automatically calculate the tributary area being carried by each member so that live load reduction factors may be applied. Various code-dependent formulations are available for those calculations; however, the values can always be overwritten with user specified values.

Shell Load Sets

Occasionally it is desirable to assign loads based on the type of occupancy, e.g., hallway or office. ETABS provides Shell Uniform Load Sets to accommodate occupancy loads that consist of several different load patterns, i.e., the load set may contain loads from both dead and live patterns. Shell load sets are assigned in the same manner as any other uniform shell load, and are additive to other assigned loads. Shell objects may be assigned only one shell uniform load set.

Wind and Seismic Lateral Loads

The lateral loads can be in the form of wind or seismic loads. The loads are automatically calculated from the dimensions and properties of the structure based on built-in options for a wide variety of building codes.

For rigid diaphragm systems, the wind loads are applied at the geometric centers of each rigid floor diaphragm. For semi-rigid diaphragms, wind loads are applied to every joint in the diaphragm. For modeling multi-tower systems, more than one rigid or semi-rigid floor diaphragm may be applied at any one story.

The seismic loads are calculated from the story mass distribution over the structure using code-dependent coefficients and fundamental periods of vibration. For semi-rigid floor systems where there are numerous mass points, ETABS has a special load dependent Ritz-vector algorithm for fast automatic calculation of the predominant time periods. The seismic loads are applied at the locations where the inertia forces are generated and do not have to be at story levels only. Additionally, for semi-rigid floor systems, the inertia loads are spatially distributed across the horizontal extent of the floor in proportion to the mass distribution, thereby accurately capturing the shear forces generated across the floor diaphragms.

ETABS also has a very wide variety of Dynamic Analysis options, varying from basic response spectrum analysis to nonlinear time history analysis. Code-dependent response spectrum curves are built into the system, and transitioning to a dynamic analysis is usually trivial after the basic model has been created.

Performance Based Design

Performance based design (PBD), which assesses the demand requirements for structures when undergoing inelastic behavior during earthquakes, is an important tool for engineers in the analysis and design of buildings. The fundamental component of PBD is nonlinear dynamic analysis where building elements are allowed to yield and form plastic hinges. ETABS allows the modeling of plastic hinges in a wide-variety of types (e.g., axial, shear, P-M, fiber) for frame and wall objects, and

can perform nonlinear time history analyses using either Fast Nonlinear Analysis (FNA) or direct integration.

Panel Zone Modeling

Studies have shown that not accounting for the deformation within a beam-column panel zone in a model may cause a significant discrepancy between the analytical results and the physical behavior of the building. ETABS allows for the explicit incorporation of panel zone shear behavior any time it is believed to have an appreciable impact on the deformation at the beam-to-column connection.

Mathematically, panel zone deformation is modeled using springs attached to rigid bodies geometrically the size of the panel zone. ETABS allows the assignment of a panel zone “property” to a point object at the beam-column intersection. The properties of the panel zone may be determined in one of the following four ways:

- Automatically by the program from the elastic properties of the column.
- Automatically by the program from the elastic properties of the column in combination with any doubler plates that are present.
- User-specified spring values.
- Users-specified link properties, in which case it is possible to have inelastic panel zone behavior if performing a nonlinear time history analysis. Link properties may also be used to specify panel zone behavior for beam to brace and brace to column connections.

Wall Stacks

ETABS offers a drawing option that allows for the quick placement of predefined wall stacks. Wall stacks are assemblages of shell objects that in plan can resemble the letters C, L, I, E, or F; single- or multi-cell boxes; or user-defined general shapes. These assemblages may include openings for doors. The thickness and length of each wall section may be set

independently, and the stack may be assigned to a single story, to a range of stories, or to the entire building height.

Live Load Reduction

Certain design codes allow for live loads to be reduced based on the area supported by a particular member. ETABS allows the live loads used in the design postprocessors (not in the analysis) to be reduced for frame objects (columns, beams, braces, and so forth) and for wall-type objects (shell objects with a wall property definition). The program also allows live loads to be reduced in concrete slab design..

ETABS offers a number of options for live load reductions, and some of the methods can have their reduced live load factors (RLLF) subject to two minimums. One minimum applies to members receiving load from one story level only, while the other applies to members receiving load from multiple levels. The program provides default values for those minimums, but the user can overwrite them. It is important to note that the live loads are reduced only in the design postprocessors; live loads are never reduced in the basic analysis output.

Rigid and Semi-Rigid Floor Models

ETABS offers three basic options for modeling various types of floor systems. Floor diaphragms can be rigid or semi-rigid (flexible), or the user may specify no diaphragm at all.

In the case of rigid diaphragm models, each floor plate is assumed to translate in plan and rotate about a vertical axis as a rigid body, the basic assumption being that there are no in-plane deformations in the floor plate. The concept of rigid floor diaphragms for buildings has been in use many years as a means to lend computational efficiency to the solution process. Because of the reduced number of degrees of freedom associated with a rigid diaphragm, this technique proved to be very effective, especially for analyses involving structural dynamics. However, the disadvantage of such an approach is that the solution will not produce any information on the diaphragm shear stresses or recover any axial forces in horizontal members that lie in the plane of the floors.

These limitations can have a significant effect on the results reported for braced frame structures and buildings with diaphragm flexibility issues, among others. Under the influence of lateral loads, significant shear stresses can be generated in the floor systems, and thus it is important that the floor plates be modeled as semi-rigid diaphragms so that the diaphragm deformations are included in the analysis, and axial forces are recovered in the beams/struts supporting the floors.

Luckily, with ETABS it is an easy process to model semi-rigid diaphragm behavior, and trivial to switch between rigid and semi-rigid behavior for parametric studies. In fact, ETABS, with its efficient numerical solver techniques and physical member-based object approach, makes many of the reasons that originally justified using a rigid diaphragm no longer pertinent.

The object-based approach of ETABS allows for the automatic modeling of semi-rigid floor diaphragms, each floor plate essentially being a floor object. Objects with opening properties may be placed over floor objects to “punch” holes in the floor system. The conversion of the floor objects and their respective openings into the finite elements for the analysis model is automatic for the most common types of floor systems, namely concrete slabs, metal deck systems and concrete planks, which use in-plane membrane behavior (see the previous *Vertical Load Transfer* section). For other types of floor systems, the user may easily assign meshing parameters to the floor objects, keeping in mind that for diaphragm deformation effects to be accurately captured, the mesh does not need to be too refined.

Post-Tensioning

Post-tensioning places concrete in compression to reduce the tensile stresses due to flexure. Post-tensioning typically reduces deflections and the amount of standard reinforcement required in slabs.

The post-tensioning option in ETABS allows for prestressing loads to be applied to concrete floor slabs. Post-tensioning tendons may be drawn either explicitly by the user, in which case the tendon layout, force, and profile are specified, or the post-tensioning layout can be determined by

ETABS using design strips. The tendon distribution in design strips can be specified as banded or distributed, and acceptable ranges for both the concrete precompression level and percentage of self-weight to be balanced may be input. ETABS then iterates to determine the profile of the tendons that best satisfies the stress and balancing requirements.

With the load-balancing method, tendons are selected to directly counteract a percentage of the self-weight load. The force in the tendon acting eccentric to the slab centroid generates a moment that is opposite of the imposed moment. If the moments were to be perfectly balanced, the net stress in the slab would be just the axial compressive stress from the post-tensioning. In reality, imposed and post-tensioning moments do not balance, and the net stress on the section is a combination of axial stress and unbalanced bending moment stress. This net stress is used in design to determine the amount of mild reinforcement required.

Edge Constraints

Part of what makes traditional finite element modeling so time consuming is creating an appropriate mesh in the transition zones of adjacent objects whose meshes do not match. This is a very common occurrence, and almost always happens at the interface between walls and floors. General purpose programs historically have had difficulties with the meshing transitions between curved walls and floors, and between walls and sloping ramps.

However, in ETABS, element mesh compatibility between adjacent objects is enforced automatically via edge constraints that eliminate the need for the user to worry about mesh transitions. These displacement interpolating edge constraints are automatically created as part of the finite element analytical model (completed internally by the program) at intersections of objects where mismatched mesh geometries are discovered. Thus, similar to the creation of the finite elements as a whole, users of ETABS do not need to worry about mesh compatibilities.

Cladding

It is often necessary to apply wind loading to the exterior surface of a building. ETABS offers a drawing option that allows for the automated creation of exterior non-structural cladding to which wind loads may be applied. The layout of the cladding may be defined using floor outlines, perimeter beams, or perimeter columns. This cladding adds neither mass, weight, nor stiffness to the building.

Soil-Foundation Interaction

The effects of soil-foundation interaction, e.g., differential settlement, may have a significant impact on building response. ETABS offers a modeling option to automatically generate foundation springs and dashpots for isolated column footings (i.e., shallow foundations) based on footing geometry and soil profile definitions. Soil profiles may consist of any number of layers and associated properties, and there is no limit to the number of footing and soil profile combinations that may be defined.

Modifiers

ETABS allows for modification factors to be assigned to both frame and shell objects. For frame objects, frame property modifiers are multiplied times the specified section properties to obtain the final analysis section properties used for the frame elements. For shell objects, shell stiffness modifiers are multiplied times the shell element analysis stiffnesses calculated from the specified section property. Both of those modifiers affect only the analysis properties. They do not affect any design properties.

The modifiers can be used to limit the way in which the analysis elements behave. For instance, assume that you have a concrete slab supported by a steel truss, but do not want the slab to act as a flange for the truss; all flange forces should be carried by the top chord of the truss. Using a shell object modifier, you can force the concrete slab to act only in shear, thereby removing the in-plane “axial” behavior of the concrete so that it does not contribute any strength or stiffness in the vertical direction of the truss. Other examples when the use of modifiers is benefi-

cial is in modeling concrete sections where it is necessary to reduce the section properties because of cracking, or when modeling lateral diaphragm behavior so that the floor objects carry only shear, with the subsequent bending forces carried directly by the diaphragm chords.

Construction Sequence Loading

Implicit in most analysis programs is the assumption that the structure is not subjected to any load until it is completely built. This is probably a reasonable assumption for live, wind and seismic loads and other superimposed loads. However, in reality the dead load of the structure is continuously being applied as the structure is being built. In other words, the lower floors of a building are already stressed with the dead load of the lower floors before the upper floors are constructed. Engineers have long been aware of the inaccurate analytical results in the form of large unrealistic beam moments in the upper floors of buildings because of the assumption of the instantaneous appearance of the dead load after the structure is built.

In many cases, especially for taller buildings because the effect is cumulative, the analytical results of the final structure can be significantly altered by the construction sequence of the building. Situations that are sensitive to the effects of the construction sequence include, among others, buildings with differential axial deformations, transfer girders involving temporary shoring, and trussed structures where segments of the truss are built and loaded while other segments are still being installed.

ETABS has an option whereby the user can activate an automated sequential construction load case. This procedure allows the structure to be loaded as it is built, story by story. Typically, you would do this for the dead load pattern and use the analytical results from the sequential construction load case in combination with the other load cases for the design phase.

Walking Vibrations

ETABS allows for the automated application of vertical pulse loadings in a path across a floor to dynamically simulate the footfall of a person

walking. The analysis output is an acceleration response that can be compared to specified thresholds to determine whether the footfall impact will be perceptible or exceed allowable levels.

Design and Drift Optimization

The various design code algorithms for member selection, stress checking and drift optimization involve the calculation of member axial and bi-axial bending capacities, definition of code-dependent design load combinations, evaluation of K-factors, unsupported lengths and second order effects, moment magnifications, and utilization factors to determine acceptability.

Energy diagrams that demonstrate the distribution of energy per unit volume for the members throughout the structure can be generated and displayed. Those displays help in identifying the members that contribute the largest to drift resistance under the influence of lateral loads. For drift control, increasing the sizes of those members will produce the most efficient use of added material.

Along the same lines, ETABS offers an automatic member size optimization process for lateral drift control based on lateral drift targets that the user specifies for any series of points at various floors. The drift optimization is based on the energy method described herein, whereby the program increases the size of the members proportionately to the amount of energy per unit volume calculated for a particular load case.

Section Designer

Section Designer is a built-in utility for defining frame and wall sections graphically. Section Designer allows sections to be built of arbitrary geometry using a combination of steel and concrete materials. Section Designer then calculates the section properties (e.g., areas, moments of inertia, torsional constant, section moduli) for use in analysis and design. Interaction surface and moment curvature diagrams for concrete sections may be displayed from within Section Designer.

More Information

This chapter was intended to illustrate some of the many techniques ETABS provides for the efficient modeling of systems and behaviors typically associated with building structures. Additional information can be found in the on-line Help facility available within the ETABS graphical user interface, including manuals that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS DVD, and can be accessed from within the program using the Help menu. In addition, the “Watch & Learn” movie series is available from CSI’s web site at www.csiamerica.com.

ETABS Analysis Techniques

This chapter provides an overview of some of the analysis techniques available within ETABS. The types of analyses described are P-Delta analysis, linear static analysis, modal analysis, response-spectrum analysis, time-history analysis, linear buckling analysis and nonlinear analysis.

In a given analysis run, you may request a P-Delta analysis, and multiple cases of linear static, modal, response spectrum, time history, and buckling analyses. Multiple nonlinear static and time history analysis cases may also be defined.

Linear Static Analysis

A linear static load case is automatically created for each load pattern that is defined. The results of different load cases can be combined with each other and with other linear load cases, such as response spectrum analyses.

Geometric and material nonlinearity, except for the P-Delta effect, are not considered in a linear static analysis.

P-Delta Analysis

The P-Delta option accounts for the effect of a large compressive or tensile load upon the transverse stiffness of members in the structure. Compression reduces lateral stiffness, and tension increases it. This type of geometric nonlinearity is known as the P-Delta effect. This option is particularly useful for considering the effect of gravity loads upon the lateral stiffness of building structures.

The P-Delta analysis in ETABS considers the P-Delta effect of a single loaded state upon the structure. This effect can be computed in one of two ways:

- **Iterative - Based on Loads:** As a specified combination of static load patterns. For example, this may be the sum of a dead load pattern plus a fraction of a live load pattern. This approach requires an iterative solution to determine the P-Delta effect upon the structure.
- **Non-iterative - Based on Mass:** As a story-by-story load upon the structure computed automatically from the mass at each level. This approach is approximate, but does not require an iterative solution.

When you select a P-Delta option, it is performed before all linear analyses in the same analysis run. The P-Delta analysis essentially modifies the characteristics of the structure, affecting the results of all subsequent analyses performed. Because the load causing the P-Delta effect is the same for all linear analysis cases, their results may be superposed in load combinations.

Finally, building codes typically recognize two types of P-Delta effects: the first due to the overall sway of the structure and the second due to the deformation of the member between its ends. ETABS can model both of those behaviors. It is recommended that the former effect be accounted for in the analysis by using the initial P-Delta option, and that the latter effect be accounted for in design by using the applicable building code moment-magnification factors. The design components in ETABS operate in this manner.

Nonlinear Static Analysis

Nonlinear static analysis can be used for a wide variety of purposes, including: to analyze a building for material and geometric nonlinearity; to form the P-delta stiffness for subsequent linear analyses; to perform static pushover analysis; to investigate staged construction; and more.

Multiple nonlinear static analysis cases can be defined. Each analysis case considers a single pattern of loading, specified as a linear combination of static load cases, acceleration loads, and vibration mode shapes.

- **P-Delta:** The P-Delta analysis option accounts for the effect of a large compressive or tensile load upon the transverse stiffness of members in the structure. A nonlinear static load case subjected to gravity loads with P-delta is often an appropriate choice for determining the initial conditions for other linear and nonlinear load cases.
- **Large Displacements:** Large displacements analysis considers the equilibrium equations in the deformed configuration of the structure. This means that if the position or orientation of an element changes, its effect upon the structure is accounted for.
- **Static Pushover:** Monitors nonlinear hinge formation as building is pushed using displacement control.
- **Staged Construction:** Staged construction allows you to define a sequence of stages wherein you can add or remove portions of the structure, selectively apply load to portions of the building, and to consider time-dependent material behavior.

Modal Analysis

Modal analysis calculates vibration modes for the structure based on the stiffnesses of the elements and the masses present. Those modes can be used to investigate the behavior of a structure, and are required as a basis for subsequent response spectrum and time history analyses.

Two types of modal analysis are available: eigenvector analysis and Ritz-vector analysis. Only one type can be used in a single load case. Modal analysis is always linear. A modal load case may be based on the stiffness of the full unstressed structure, or upon the stiffness at the end of a nonlinear load case. By using the stiffness at the end of a nonlinear case, you can evaluate the modes under P-delta conditions.

Mass Source

To calculate modes of vibration, a model must contain mass. Mass may be determined and assigned in ETABS using any of the following approaches:

- ETABS determines the building mass on the basis of object self masses (defined in the properties assignment) and any additional masses that the user specifies. This is the default approach.
- ETABS determines the mass from a load pattern that the user specifies.
- ETABS determines the mass on the basis of self masses, any additional masses the user assigns, and any load pattern that the user specifies. This is a combination of the first two approaches.

Typically, masses are defined in all six degrees of freedom. However, ETABS has an option that allows only assigned translational mass in the global X and Y axes directions and assigned rotational mass moments of inertia about the global Z axis to be considered in the analysis. This option is useful when vertical dynamics are not to be considered in a model. In addition, an option exists for all lateral masses that do not occur at a story level to be lumped together at the story level above and the story level below the mass location. That approach is used primarily to eliminate the unintended dynamic out-of-plane behavior of walls spanning between story levels.

Eigenvector Analysis

Eigenvector/eigenvalue analysis determines the undamped free-vibration mode shapes and frequencies of the system. Those natural modes provide

an excellent insight into the behavior of the structure. They can also be used as the basis for response spectrum or time history analyses, although Ritz vectors are strongly recommended for those purposes.

The eigenvector modes are identified by numbers from 1 to n in the order the modes are found by the program. Specify the number of modes, N , to be found, and the program will seek the N -lowest frequency (longest period) modes.

The eigenvalue is the square of the circular frequency. The user specifies a cyclic frequency (circular frequency/ (2π)) range in which to seek the modes. Modes are found in order of increasing frequency, and although starting from the default value of zero is appropriate for most dynamic analyses, ETABS does allow the user to specify a starting “shift frequency”; this can be helpful when your building is subjected to higher frequency input, such as vibrating machinery.

ETABS also offers an option for calculating residual-mass (missing-mass) modes for eigen-analyses. In this way, ETABS tries to approximate high-frequency behavior when the mass participation ratio for a given direction of acceleration load is less than 100%.

Ritz-Vector Analysis

ETABS offers the ability to use the sophisticated Ritz-vector technique for modal analysis. Research has indicated that the natural free-vibration mode shapes are not the best basis for a mode-superposition analysis of structures subjected to dynamic loads. It has been demonstrated that dynamic analyses based on load-dependent Ritz vectors yield more accurate results than the use of the same number of eigenvalue/eigenvector mode shapes.

Ritz vectors yield excellent results because they are generated considering the spatial distribution of the dynamic loading. The direct use of the natural mode shapes neglects this important information.

Each Ritz-vector mode consists of a mode shape and frequency. When a sufficient number of Ritz-vector modes have been found, some of them may closely approximate natural mode shapes and frequencies. In general, however, Ritz-vector modes do not represent the intrinsic character-

istics of the structure in the same way the natural modes do because they are biased by the starting load vectors.

Similar to the natural modes, specify the number of Ritz modes to be found. In addition, specify the starting load vectors, which may be acceleration loads, static load cases, or nonlinear deformation loads.

Response Spectrum Analysis

For response spectrum analyses, earthquake ground acceleration in each direction is given as a digitized response spectrum curve of pseudo-spectral acceleration response versus period of the structure. This approach seeks to determine the likely maximum response rather than the full time history.

ETABS performs response spectrum analysis using mode superposition, and eigenvector or Ritz vectors may be used. Ritz vectors are typically recommended because they give more accurate results for the same number of modes.

Even though input response spectrum curves may be specified in three directions, only a single, positive result is produced for each response quantity. The response quantities may be displacements, forces, or stresses. Each computed result represents a statistical measure of the likely maximum magnitude for that response quantity. Although all results are reported as positive, actual response can be expected to vary within a range from this positive value to its corresponding negative value.

Linear Time History Analysis

Time history analysis is used to determine the dynamic response of a structure to arbitrary loading. ETABS can complete any number of linear time history cases in a single execution of the program. Each case can differ in the load applied and in the type of analysis to be performed, and the building may be subjected to a suite of time histories using time history sets. Two types of linear time history analyses are available:

- **Modal:** The standard mode superposition method of response analysis is used by the program to solve the dynamic equilibrium equations of motion for the complete structure. The modes used can be the eigenvector or the load dependent Ritz vector modes, and the damping in the structure is modeled using modal damping, also known as proportional or classical damping. The Ritz vector algorithm is faster than the eigenvector algorithm, and is therefore recommended for time history analyses.
- **Direct Integration:** This technique uses the direct integration of the full equations of motion. Although modal superposition is often more accurate and efficient than direct integration, direct integration provides better response when modes are coupled or blast/impact type loads are involved.

Nonlinear Time History Analysis

As with the linear time history analysis, two types of nonlinear analyses are available:

- **Modal:** The method of nonlinear modal time-history analysis used in ETABS is an extension of the Fast Nonlinear Analysis (FNA) method. This method is extremely efficient and is intended for use with structural systems that are primarily linear elastic, but which have a limited number of predefined nonlinear elements, such as buildings with base isolators and/or dampers.
- **Direct Integration:** Nonlinear direct integration provides the same costs and benefits as the linear procedure. Direct integration results are extremely sensitive to time-step size in a way that is not true for modal superposition..

The FNA method is highly accurate when used with appropriate Ritz vector modes, and has advantages over traditional time-stepping methods in terms of speed, and control over damping and higher mode effects.

Buckling Analysis

Linear buckling analysis seeks the instability modes of a structure due to the P-delta effect under a specified set of loads. Each eigenvalue-eigenvector pair is called a buckling mode of the structure. The eigenvalue is called the buckling factor. It is the scale factor that loads must be multiplied by to cause buckling in the given mode.

Hyperstatic Analysis

A hyperstatic analysis may be required when doing concrete slab design utilizing post-tensioning. This type of analysis is based on a previously defined linear static load case (i.e., post-tensioning), in which the reactions from this static load case are applied as loads to an unsupported structure.

More Information

This chapter provides a general introduction to the primary analytical techniques ETABS offers for linear and nonlinear analysis of buildings. Additional information can be found in the on-line Help facility available within the ETABS graphical user interface, including manuals that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS DVD, and can be accessed from within the program using the Help menu. In addition, the “Watch & Learn” movie series is available from CSI’s web site at www.csiamerica.com.