

ETABS[®]

Integrated Building Design Software

Welcome to ETABS



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

Version 8
February 2003

Copyright

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

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

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

Phone: (510) 845-2177

FAX: (510) 845-4096

e-mail: info@csiberkeley.com (for general questions)

e-mail: support@csiberkeley.com (for technical support questions)

web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2003.

The CSI Logo is a registered trademark of Computers and Structures, Inc.

ETABS is a registered trademark of Computers and Structures, Inc.

“Watch & Learn” is a trademark of Computers and Structures, Inc.

Windows is a registered trademark of Microsoft Corporation.

Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

DISCLAIMER

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

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

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	1-5
	Analysis Features	1-6
2	Getting Started	
	Installing ETABS	2-1
	If You Are Upgrading	2-1
	About the Manuals	2-2
	“Watch & Learn”™ Movies	2-2
	Technical Support	2-2
	Help Us Help You	2-3
	Telephone and Facsimile Support	2-4
	Online Support	2-4
3	The ETABS System	
	Overview of the Modeling Process	3-1
	Physical Modeling Terminology	3-2

Story Definition	3-3
Units	3-4
Coordinate Systems and Grids	3-4
Structural Objects	3-5
Groups	3-7
Properties	3-7
Static Load Cases	3-8
Vertical Loads	3-8
Temperature Loads	3-9
Automated Lateral Loads	3-9
Functions	3-10
Load Combinations	3-11
Design Settings	3-12
Output and Display Options	3-13
More Information	3-14
4 ETABS Modeling Techniques	
Auto-Select Properties	4-1
Vertical Load Transfer	4-2
Wind and Seismic Lateral Loads	4-3
Panel Zone Modeling	4-4
Live Load Reduction	4-5
Rigid and Semi-Rigid Floor Models	4-5
Line Constraints	4-6
Modifiers	4-7
Construction Sequence Loading	4-8
Steel Frame Design and Drift Optimization	4-8
More Information	4-9

5 ETABS Analysis Techniques

Linear Static Analysis	5-1
Modal Analysis	5-2
Mass Source	5-2
Eigenvector Analysis	5-3
Ritz-Vector Analysis	5-3
Response Spectrum Analysis	5-4
Time History Analysis	5-5
Nonlinear Time History	5-6
Initial P-Delta Analysis	5-6
Nonlinear Static Analysis	5-7
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 Version 8 features an intuitive and powerful graphical interface coupled with unmatched modeling, analytical, and design 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, making it the tool of choice for structural engineers in the building industry.

History and Advantages of ETABS

Dating back more than 30 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, providing 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 Version 8 looks radically different from its predecessors of 30 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 numerically to reduce computational effort.
- 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
- Staggered truss buildings
- Buildings with steel, concrete, composite or joist floor framing
- 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
- P-Delta analysis with static or dynamic analysis
- Explicit panel-zone deformations
- 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
- 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 and design 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 maintained, and no data is transferred between programs or modules. The effects on one part of the structure from changes in another part are instantaneous and automatic. The integrated modules include:

- Drafting module for model generation.
- Seismic and wind load generation module.
- Gravity load distribution module for the distribution of vertical loads to columns and beams when plate bending floor elements are not provided as a part of the floor system.
- Finite element-based linear static and dynamic analysis module.
- Finite element-based nonlinear static and dynamic analysis module (available in ETABS Nonlinear version only).
- Output display and report generation module.
- Steel frame design module (column, beam and brace).
- Concrete frame design module (column and beam).

- Composite beam design module
- Steel joist design module
- Shear wall design module.

ETABS Version 8 is available in two versions:

- **ETABS Plus.** Includes all of the capabilities except for nonlinear static and dynamic analysis. The steel frame design, concrete frame design, composite beam design, steel joist design, and shear wall design modules are all included.
- **ETABS Nonlinear.** Includes all of the features of ETABS Plus, with the added capability of nonlinear static and dynamic analysis.

Modeling Features

The ETABS building is idealized as an assemblage of area, line and point objects. Those objects are used to represent wall, floor, column, beam, brace 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, plate bending elements with out-of-plane stiffness only or full shell-type elements, which combine both in-plane and out-of-plane stiffness. Floor and wall objects may have uniform load patterns in-plane or out-of-plane, and they may have temperature loads. The column, beam, brace, floor and wall objects are all compatible with one another.

Analysis Features

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



Note:

The ETABS Software Verification Manual documents analysis using ETABS.

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 conditions 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, sent to a printer, 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.

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 a previous 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 approach, 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, an *Introductory User's Guide* and a *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 13 chapters of the *Introductory User's Guide* provide an introduction to the steps and menu items used to create, analyze and design a model. The *Tutorial* describes the model creation, analysis, and design processes for an example model. The *Software Verification Manual* describes analysis of a set of simple building systems using ETABS.

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 Technical Notes that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS CD, 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 on the ETABS CD or via the CSI web site at <http://www.csiberkeley.com>. Those 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 2 minutes to 13 minutes in length.

Technical Support

Free technical support is available from Computers and Structures, Inc. (CSI), via telephone, facsimile, and e-mail for 90 days after program purchase. After 90 days, technical support is available according to the terms of the Software Maintenance Agreement, which you may purchase from CSI or your dealer. Please contact CSI or your dealer to inquire about a Software Maintenance Agreement.

If you have questions regarding use of the program, we suggest the following:

- Consult this documentation and other printed information included with your product.
- Check the on-line Help facility in the program.
- Review the “Watch and Learn”™ movies provided on the ETABS CD or at CSI’s web site at <http://www.csiberkeley.com>.

If you cannot find an answer, contact us as described in the next section.

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 program level (PLUS or Nonlinear) and version number that you are using. This can be obtained from inside the program 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.

Telephone and Facsimile Support

Standard phone and fax support is available in the United States, from CSI support engineers, via a toll call between 8:30 A.M. and 5:00 P.M., Pacific time, Monday through Friday, excluding U.S. holidays.

You may:

- Contact CSI's office via phone at (510) 845-2177, or
- Send a fax with questions and information about your model (including a picture, if possible) to CSI at (510) 845-4096.

When you call, please be at your computer and have the program manuals at hand.

Online Support

Online support is available by:

- Sending an e-mail and your model file to *support@csiberkeley.com*
- Visiting CSI's web site at *http://www.csiberkeley.com* to read about frequently asked questions.

If you send us e-mail, be sure to include all of the information requested in the previous "Help Us to Help You" section.

The ETABS System

ETABS analyzes and designs 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 point, line and area 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 point, line and area objects that represent your building using the various drawing tools available within the graphical interface.
- 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 box, which appears when Draw commands are used.
- Assign meshing parameters to area objects if they are not horizontal membrane slab or deck/plank sections that the program will automatically mesh into the elements needed for the analysis model.

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, link elements and shell (membrane and plate) elements in contrast to the point, line and area objects in the user defined object-based model. 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 line 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 is accomplished at the object level.

This differs from a traditional analysis program, where the user is required to define a sub-assembly 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 in plan 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 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.

Units

ETABS works with four basic units: force, length, temperature, and time. The program offers many different compatible sets of force, length and temperature units to choose from, such as “Kip, in, F” or “N, mm, C.” Time is always measured in seconds.

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 specify a set of units. These become the “base units” for the model. Although you may provide input data and view output results in any set of units, those values are always converted to and from the base units of the model.

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).

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 (white), 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:

- **Point 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.

- **Grounded (one joint) link objects** are used to model special support behavior, such as isolators, dampers, gaps, multi-linear springs and more.
- **Line objects** of two types:
 - **Frame objects** are used to model beams, columns, braces and trusses.
 - **Connecting (two-joint) link 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.
- **Area objects** are used to model walls, slabs, decks, planks, and other thin-walled members. Area objects will be meshed automatically into the elements needed for analysis if horizontal objects with the membrane definition are included in the model; otherwise, the user should specify the meshing option to be used.

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 (except for certain Area objects; see previous bullet item) 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 called CONCRETE.
- A rectangular frame section property called RECTANGLE, and a circular frame section called CIRCULAR, both using material property CONCRETE.
- A wall/slab section property called SLAB that also uses material property CONCRETE.

If you assign frame section property RECTANGLE to a line 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. These properties can only be changed by making an-

other assignment of that same property to the object; they are not named entities and they do not exist independently of the objects.

Static Load Cases

Static 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 case.

Define as many named static load cases as needed. Typically, separate load case definitions would be used 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 cases.

After defining a static load case name, you must assign specific load values to the objects as part of the load case, or define an automated lateral load if the case is for quake 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 case, along with the automated lateral load, if so desired. Each object can be subjected to multiple load cases.

Vertical Loads

Vertical loads may be applied to point, line and area objects. Vertical loads are typically input in the gravity, or -Z direction. Point 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 loads can be applied to Area 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 case, 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 line and area 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 lines, and linearly over the object surface for areas. Although you can specify a temperature change for a point object, temperature loads act only on line and area objects.

Automated Lateral Loads

ETABS allows for the automated generation of static lateral loads for either earthquake (quake) or wind load cases based on numerous code specifications, including, but not limited to, UBC, BOCA, ASCE, NBCC, BS, JGJ, Mexican and IBC. Each automatic static lateral load that you define must be in a separate load case. You cannot have two automatic static lateral loads in the same load case. You can, however, add additional user-defined loads to a load case that includes an auto lateral load.

If you have selected quake as the load type, various auto lateral load codes are available. Upon selection of a code, the Seismic Loading 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.

If you have selected wind as the load type, various auto lateral load codes are available. Upon selection of a code, the Wind Loading 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 rigid diaphragms or to walls, including non-structural walls such as cladding, that are created using area objects. 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.

If you have selected the option whereby wind loads are calculated and applied via area objects defining walls, you must assign a wind pressure coefficient to each area object that has exposure, and indicate whether it is windward or leeward. Based on the various code factors and user defined coefficients and exposures, ETABS calculates the wind loads for each area (wall) object and applies the loads as point forces at the corners of the object.

Functions

You define functions to describe how a load varies as a function of period or time. 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 pseudo-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 cases.

You may define as many named functions as you need. They are not assigned to objects, but are used in the definition of Response Spectrum and Time History cases.

Load Combinations

ETABS allows for the named combination of any previously defined load case or load combination. When a load combination is defined, it applies to the results for every object in the model.

The four types of combinations are as follows:

- **ADD (Additive):** Results from the included load cases or combos are added.
- **ENVE (Envelope):** Results from the included load cases or combos are enveloped to find the maximum and minimum values.
- **ABS (Absolute):** The absolute values of the results from the included load cases or combos are added.
- **SRSS:** The square root of the sum of the squares of the results from the included load cases or combos is computed.

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

Design is always based on load combinations, not directly on load cases. You may create a combination that contains just a single load case. Each design algorithm creates its own default combinations; supplement them with your own design combination if needed.

Design Settings

ETABS offers the following integrated design postprocessors:

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

The first four design procedures are applicable to line objects, and the program determines the appropriate design procedure for a line 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 by the user, and both piers and spandrels may consist of both area and line objects.

For each of the 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-LRFD93 for steel frames, EUROCODE 2-1992 for concrete frames, and BS8110 97 for shear walls.
- Preference settings of how these codes should be applied to your model.

- Load combinations for which the design should be checked.
- Groups of objects that should share the same design.
- For each object, optional “overwrite” values that supercede the default coefficients and parameters used in the design code formulas selected by the program.

For steel frame, composite beam, and steel joist design, ETABS can automatically select an optimum section from a list you define. You can also manually change the section during the design process. As a result, each line 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.

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
- Input/output data values in plain text, spreadsheet, or database format
- Function plots of analysis results
- Design reports
- Export to other drafting and design programs

You may save named definitions of display views, sets of tables, and function plots as part of your model. Combined with the use of groups, this can significantly speed up the process of getting results while you are developing your model.

More Information

This chapter presented just a brief overview of some of the basic components of the ETABS model. Additional information can be found in the on-line Help facility available within the ETABS graphical user interface, including Technical Notes that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS CD, 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 line 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 steel line 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 for the three most popular floor systems in building construction: concrete slabs, steel deck with concrete fill, and concrete planks. 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 case, if so desired.

The main issue for vertical load transfer is the distribution of point, line and area loads that lie on the area object representing the floor plate in the analysis model. ETABS analysis for vertical load transfer differs for floor-type objects with membrane only behavior and floor-type objects with plate bending behavior. The following bullets describe analysis for floor-type objects with membrane only type behavior:

- **Out-of-plane load transformation for floor-type area 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 (see the Technical Notes accessed using the Help menu for a detailed description). The load transfer takes into account that the deck or plank spans in only **one** direction.

- **Out-of-plane load transformation for floor-type area 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 (see the Technical Notes accessed using the Help menu for a detailed description). The load transformation takes into account that the slab spans in **two** directions.

For floor area objects that have plate bending behavior, ETABS uses a bilinear interpolation function to transfer loads applied to or on area objects to the corner points of the shell/plate elements in the analysis model.

Note that ETABS will automatically mesh floor objects with membrane properties for analysis. Typically, floors with plate bending behavior require user assigned meshing before analysis (with the exception of waffle and ribbed slabs generated using templates, which are automatically meshed even though they have plate bending behavior).

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.

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 modeling multi-tower systems, more than one 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 large deformation 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.

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 a considerable 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.

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 line objects (columns, beams, braces, and so forth) and for wall-type objects (area objects with a wall property definition). The program does not allow the reduction of live loads for floor-type area objects.

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.

Line 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 line constraints that eliminate the need for the user to worry about mesh transitions. These displacement interpolating line 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.

Modifiers

ETABS allows for modification factors to be assigned to both line and area objects. For line objects, frame property modifiers are multiplied times the specified section properties to obtain the final analysis section properties used for the frame elements. For area 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 an area 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 beneficial 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 automatic incremental story-by-story construction sequence loading of the building for a particular load case. This procedure will load the structure as it is built. Typically, you would do this for the dead load case and use the analytical results from the construction sequence loading in combination with the other load cases for the design phase.

Steel Frame Design and Drift Optimization

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

You can generate displays of energy diagrams that demonstrate the distribution of energy per unit volume for the members throughout the structure. 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 you specify 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.

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 Technical Notes that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS CD, 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.csiberkeley.com.

ETABS Analysis Techniques



Note:

*The ETABS
Software
Verification
Manual
documents
analysis
using ETABS.*

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

In a given analysis run, you may request an initial P-Delta analysis, a modal analysis, and multiple cases of linear static, response spectrum, and time history analyses. Multiple nonlinear static analysis cases may also be defined; these are performed separately from the other analysis cases.

Linear Static Analysis

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

Geometric and material nonlinearity are not considered in linear static analysis, except that the effect of the initial P-Delta analysis is included in every static load case. For example, if you define an initial P-Delta analysis for gravity load, deflections and moments will be increased for lateral static load cases.

Linear static load cases can still be combined when an initial P-Delta analysis has been performed, because the initial P-Delta load is the same for all static load and response spectrum cases.

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 analysis run.

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 you specify. This is the default approach.
- ETABS determines the mass from a load combination that you specify.
- ETABS determines the mass on the basis of self masses, any additional masses you assign, and any load combination that you specify, which 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 characteristics 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 re-

sults are reported as positive, actual response can be expected to vary within a range from this positive value to its corresponding negative value.

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 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. Three types of time history analyses are available:

- **Linear transient:** The structure starts with zero initial conditions or with the conditions at the end of a user-specified previous linear transient time history case. All elements are assumed to behave linearly for the duration of the analysis.
- **Periodic:** The initial conditions are adjusted to be equal to those at the end of the period of analysis. All elements are assumed to behave linearly for the duration of the analysis.
- **Nonlinear transient:** The structure starts with zero initial conditions or with the conditions at the end of a user-specified previous nonlinear transient time history case. The link elements may exhibit nonlinear behavior during the analysis. All other elements behave linearly.

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. Ritz vectors should be used when performing a nonlinear time history analysis with the nonlinear link deformation loads as starting vectors.

Nonlinear Time History Analysis

The method of nonlinear 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 or dampers. In ETABS, all nonlinearity is restricted to the nonlinear link elements.

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.

Initial P-Delta Analysis

The initial P-Delta analysis option accounts for the effect of a large compressive or tensile load upon the transverse stiffness of members in the structure. Compression reduces lateral stiffness, and tension increases it. This is a type of geometric nonlinearity 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, as required by certain design codes.

Initial P-Delta analysis in ETABS considers the P-Delta effect of a single loaded state upon the structure. Specify the load using either of the following methods:

- As a specified combination of static load cases; this is called the P-Delta load combination. For example, this may be the sum of a dead load case plus a fraction of a live load case. This approach requires an iterative solution to determine the P-Delta effect upon the structure.
- As a story-by-story load upon the structure computed automatically from the mass at each level. This approach is approximate, but does not require an iterative solution.

When you request initial P-Delta analysis, it is performed before all linear-static, modal, response spectrum, and time history analyses in the

same analysis run; initial P-Delta analysis has no effect on nonlinear-static analyses. The initial 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.

Initial P-Delta analysis may also be used to estimate buckling loads in a building by performing a series of analyses, each time increasing the magnitude of the P-Delta load combination, until buckling is detected (if the program detects that buckling has occurred, the analysis is terminated and no results are produced). The relative contributions from each static load case to the P-Delta load combination must be kept the same, increasing all load case scale factors by the same percentage between runs.

In conclusion, building codes typically recognize two types of P-Delta effects: the first caused by the overall sway of the structure and the second resulting from the deformation of a member between its ends. ETABS can model both behaviors. It is recommended that the initial P-Delta option be used in analysis for overall sway of the structure and the applicable building code moment-magnification factors be used in analysis for the deformation of a member between its ends. The design post-processors in ETABS operate in this manner.

Nonlinear Static Analysis

Nonlinear static analysis in ETABS offers a variety of capabilities, including:

- Material nonlinearity in beams and columns.
- Nonlinear gap, hook, and plasticity behavior in links.
- Geometric nonlinearity, including large deflections and P-Delta effects.
- Incremental construction analysis.
- Static pushover analysis.

Multiple 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. Loads are applied incrementally within an analysis case.

The load pattern may be applied under load or displacement control. Load control is used to apply a known magnitude of load, such as would be required for gravity load in an incremental construction analysis. Displacement control applies the load with a variable magnitude to achieve a specified displacement, as would be needed in a pushover analysis.

Nonlinear static analysis is independent of all other analysis cases, except that previously calculated mode shapes may be used to define the load pattern.

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 *Software Verification Manual* and the on-line Help facility available within the ETABS graphical user interface, including Technical Notes that describe code-specific design algorithms. Those documents are available in Adobe Acrobat PDF format on the ETABS CD, 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.csiberkeley.com.