SAFE™

Integrated Analysis and Design of Slab Systems

Welcome to SAFE



Computers and Structures, Inc.
Berkeley, California, USA

Version 8.0.0
August 2004

Copyright

The computer program SAFE 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., 1978-2004.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
SAFE and CSiDETAILER are trademarks 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 SAFE. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

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



Contents

Welcome to SAFE

1 Introduction

History and Advantages of SAFE 1-1

What SAFE Can Do! 1-2

An Integrated Approach 1-4

Modeling Features 1-4

Analysis Features 1-5

Design Features 1-5

Detailing Features 1-6

2 Getting Started

Installing SAFE 2-1

If You are Upgrading 2-1

About the Manuals 2-2

"Watch & Learn™ Movies" 2-2
Technical Support 2-3
Help Us Help You 2-3
Telephone and Facsimile Support 2-4
On-Line Support 2-4

3 The SAFE System

Overview of the Modeling Process 3-1
Physical Modeling Terminology 3-2
Structural Objects 3-3
Properties 3-4
Supports 3-4
Static Load Cases 3-5
Vertical Loads 3-5
Lateral Load Effects 3-6
Load Combinations 3-6
Design Procedures 3-6
More Information 3-7

4 SAFE Modeling Techniques

Overview of Slab Types 4-1
Two-Way Slabs 4-1
Flat Slabs 4-2
Ribbed Slabs 4-2
Waffle Slabs 4-3
Slabs with Discontinuities 4-3
Mat Foundations and Footings 4-3
Beam Types 4-3

Models Exported from ETABS 4-4
More Information 4-4

5 SAFE Analysis Techniques

Overview of the Analysis Process 5-1
The Analysis Model 5-2
Linear Static Analysis 5-4
Nonlinear Analysis for Uplift 5-4
Nonlinear Analysis for Cracking 5-5
More Information 5-6

6 SAFE Design Techniques

Overview of the Design Process 6-1
Slab Flexural Design 6-2
Design Strips 6-2
Integration of Moments – Nodal 6-3
Integration of Moments – Wood-Armer 6-3
Slab Punching Shear Check 6-5
Beam Flexural and Shear Design 6-5
Output Display Options 6-6
More Information 6-6

7 CSiDETAILER Detailing Options

Overview of the Detailing Process 7-1
The Detailing Model 7-2
Drawings 7-2
Views 7-3
Tables and Schedules 7-4
More Information 7-4



Chapter 1

Introduction

SAFE is a sophisticated, yet easy to use, special purpose analysis and design program developed specifically for concrete slab systems. SAFE couples powerful object-based modeling tools with an intuitive graphical interface, allowing the user to quickly and efficiently model slabs of regular or arbitrary geometry with openings, drop panels, ribs, edge beams, and slip joints supported by columns, walls or soil. Design is seamlessly integrated with modeling and analysis, and provides comprehensive reporting of the required reinforcing calculated by the program based on the user's choice of design code. And with the optional CSiDETAILER program, detail drawings may be produced almost effortlessly for the slabs and beams designed using SAFE.

History and Advantages of SAFE

Slab systems are a very special class of structures. They are characterized by their simplicity in geometry and loading. They are typically horizontal plates supported vertically by beams, columns or walls. The loading in general is comprised of vertical point, line and surface loads. Basemats

share the same characteristics as those of elevated slabs, with the exception that basemats are supported on soil and loaded by columns and walls.

Recognition of the unique characteristics of slab systems led to the original development of the SAFE program more that two decades ago. Early releases of SAFE provided input, output and numerical solution techniques that took into consideration the modeling and analysis needs distinctive to concrete slabs, providing a tool that offered significant savings in time over general purpose finite element programs, and increased accuracy in comparison to equivalent frame methods.

As computers and computer interfaces evolved, so did SAFE, adding computationally complex analytical options such as cracked section analysis, and powerful CAD-like drawing tools in a graphical and object-based interface. Although the current version looks radically different from its predecessors of 20 years ago, its mission remains the same: to provide the profession with the most efficient and comprehensive software for the analysis and design of slab systems. SAFE automates the analysis and design process for the structural engineer, resulting in more sophisticated designs produced in fewer engineering labor hours.

Creation and modification of the slab model, execution of the analysis, checking and optimization of the design, and display of graphical results are all controlled through a single interface, and all aspects of the program are linked via a common database.

SAFE also serves up the latest developments in numerical techniques, solution algorithms, and design codes, including automatic finite element meshing of complex object configurations, very accurate plate bending elements, stress-integration options that account for diagonal cracking behavior, and the most recent American and International concrete design codes.

What SAFE Can Do!

SAFE offers the widest assortment of analysis and design tools available for the structural engineer working on concrete slabs. The following list

represents just a portion of the types of systems and analyses SAFE can handle easily:

- Flat slabs
- Flat slabs with perimeter beams
- Basemats
- Two-way slabs
- Waffle slabs
- Ribbed slabs
- Rectangular or circular slabs
- T-beam effects
- Spread footings
- Combined footings
- Slabs subjected to any number of vertical load cases and combinations
- Pattern live loads
- Foundation uplift
- Deflections calculated using cracked section analysis
- Wall supports with out-of-plane bending stiffness
- Slab reinforcing calculated based on user-defined design strips
- Flexural and shear design of beams
- Punching shear ratios
- Design for twisting moments
- Automatic transfer of geometry, loading and slab distortions from ETABS

- And much, much more!

An Integrated Approach

SAFE is a completely integrated system. Embedded beneath the simple, intuitive user interface are very powerful numerical methods and design procedures, all working from a single comprehensive database. This integration means that only one model is created to analyze, design, and if CSiDETAILER is present, detail the entire slab.

Everything is integrated into one versatile package with a single Windows-based graphical user interface. No external modules need to be maintained, and data transfer between programs or modules is worry free. The effects on one part of the slab from changes in another part are instantaneous and automatic. The integrated modules include the following:

- Drafting module for model generation.
- Finite element based analysis module.
- Output display and table generation module.
- Concrete slab and beam design module.

Modeling Features

The SAFE slab is idealized as an assemblage of area, line and point objects. Area objects are used to model slabs, openings, soil supports, and surface loads. Line objects model beams and wall supports and loads. Point objects are used for column supports and concentrated loads. With relatively simple modeling techniques, very complex slab systems may be considered.

The geometry of the slab can be unsymmetrical, and the thickness of the slab may vary. Spacing of supports and loads may be completely arbitrary and is not limited to the uniform spans typically associated with equivalent frame techniques.

Construction or expansion joints may be modeled with or without shear transfer by assigning bending and shear releases to the line object that represents the joint.

Column and wall supports can provide both vertical stiffness and rotational stiffness to give a more accurate representation of the distribution of forces in the slab.

Analysis Features

Static analyses for any number of user-specified vertical load cases are possible, and those load cases may be combined with each other into any number of load combinations. Deflections calculated may include those for both elastic and cracked sections, and load cases may have long-term deflection multipliers for the nonlinear cracked deflection analyses.

Basemats and foundations may be subjected to overturning moments that cause uplift. The program offers a nonlinear solution that accounts for no tension in the soil springs, as well as allowing soil moduli to vary throughout the model.

To account for orthotropic effects, a thickness may be specified in both the X and Y directions for the slab properties. This can be useful to model slabs that have primarily one-way behavior, where cracking may be predominant in one direction.

Output may be viewed graphically (including soil bearing pressure maps), displayed in tabular output, sent to a printer, exported to a database file, exported to a CAD file, or saved in an ASCII file. Types of output include slab and beam forces, reactions and displacements.

Design Features

Flexural design of reinforced concrete slabs and basemats and the flexural and shear design of beams can be performed based on a variety of international design codes. Slab reinforcement location and layout is controlled using design strips. Design strip moments are obtained by integrating the finite element stresses using an algorithm that always satis-

fies equilibrium and accounts for the effects of twisting moments. Code based punching shear checks are made around column supports and point loads.

Detailing Features

Drawings showing the reinforcing detailing may be produced for both slabs and beams when CSiDETAILER is used in conjunction with SAFE. The detailing may be based on the requirements of a selected design code or on the preferences of the user. The user may prepare any number of drawings, and the drawings may contain plan views of reinforcing layouts, sections, elevations, tables and schedules. Control over reinforcing bar sizes and minimum and maximum spacing along with cut-off (curtailment) lengths is provided through a detailing preferences command. Drawings may be printed directly from CSiDETAILER or exported to a DXF or DWG file for further refinement.



Chapter 2

Getting Started

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

Installing SAFE

Please follow the installation instructions provided in the separate installation document included in your SAFE 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 SAFE, you should 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 SAFE. It provides this *Welcome to SAFE* manual, an *Introductory User's Guide*, a *Tutorial*, a *Design Manual*, a *User's Manual* for CSiDETAILER (an optional detailing program available with SAFE) and a *Verification* manual.

The next chapter of this *Welcome to SAFE* manual gives a synopsis of the terminology used in SAFE, while chapters 4 and 5 describe modeling and analysis techniques, respectively. Chapter 6 covers the design options, while chapter 7 provides an overview of the detailing features available with CSiDETAILER, the optional program available with SAFE for generating detailed drawings of SAFE models. Additional information about CSiDETAILER is available in the CSiDETAILER *User's Manual*.

The 14 chapters of the SAFE *Introductory User's Guide* provide an introduction to the menu items used to create, analyze, and design a model. The SAFE *Tutorial* takes the user through the creation, analysis and design of an example model. Further information about design in accordance with various codes is provided in the *SAFE Design Manual*. The SAFE *Verification* manual provides comparisons of SAFE output with published results.

It is strongly recommended that you read this volume and view the tutorial movies available through CSI's web site (see "Watch & Learn™ Movies") before attempting to complete a project using SAFE.

Additional information can be found in the on-line Help facility available within the SAFE graphical user interface, including Technical Notes in Adobe Acrobat .PDF format that describe various design techniques.

"Watch & Learn™ Movies"

One of the best resources available for learning about the SAFE program is the "Watch & Learn™ Movies" series, which may be accessed on the SAFE CD or through 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 vary in length from 3 to 15 minutes and include narration.

Technical Support

Free technical support is available from Computers and Structures, Inc. (CSI) via telephone, facsimile, and e-mail for 90 days after the software has been purchased. After 90 days, technical support is available in accordance with 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 software, 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 SAFE CD or at CSI's web site at *http://www.csiberkeley.com.*

If you cannot find an answer, then contact us as described in the sections that follow.

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 version number that you are using. This can be obtained from inside the program using the **Help menu > About SAFE** 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 telephone 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.



Chapter 3

The SAFE System

SAFE analyzes and designs a concrete slab system for the model created using the graphical user interface. The key to successfully implementing SAFE is to understand the unique and powerful approach the program takes in modeling slabs. This chapter will provide an overview of some of the special features 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:

- The program is optimized for modeling slab systems because the modeling procedures and design capabilities are all tailored to concrete slabs and beams.
- The program's models are object-based and consist of point, line and area objects to which assignments are made to define structural members, such as slabs, beams, columns, and supports, as well as to define loads.

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

- Define slab and beam properties (sections and materials) and supports using the Define menu options.
- Draw a series of point, line and area objects that represent your slab using the various drawing tools available within the graphical interface.
- Assign structural properties, supports and loads to objects using the Assign menu options. Note that the assignment of structural properties and supports may be completed concurrently with the drawing of the object using the Properties of Object form that appears when a Draw command is used.

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 that is used for the analysis—this is known as the analysis model. The analysis model consists of joints, frame elements, plate elements and springs 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. In addition, object meshing is performed automatically, with control over the maximum mesh dimension available on the Analysis menu.

Physical Modeling Terminology

In SAFE, 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 SAFE interface. Objects are intended to be an accurate representation of the physical members; users typically need not concern themselves with the meshing of these objects into the elements required for the mathematical, or analysis, model. For example, a single area object can model an entire slab, regardless of the number of spans and variety of loadings. With SAFE, both model creation, as well as the reporting of results, is achieved 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 SAFE, the objects, or physical members drawn by the user, are typically meshed 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 both to create the model and to interpret the results.

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 SAFE. 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.

Structural Objects

As stated previously, SAFE 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 slab structure.

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

- **Point objects** are automatically created at the corners or ends of all other types of objects and also can be added anywhere in the model. Point objects are used to model column supports, point loads and point displacements.
- **Line objects** are used to model beams, wall supports and line loads.
- **Area objects** are used to model slabs, drop panels, column geometry, openings, soil supports and surface loads.

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 reduces the chances of input error.

Properties

Properties are "assigned" to each object to define the structural behavior of that object in the model. Some properties, such as slab and beam properties that contain both material and section definitions, are named entities that must be specified before assigning them to objects. If a property is assigned to an object, for example a beam property to a line object, any changes to the definition of the property will automatically apply to the object. A named property has no effect on the model unless it is assigned to an object.

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

Supports

Supports may be assigned to point, line and area objects, and similar to properties, they are named entities that must be specified before being assigned. The three types of supports are as follows:

- Column supports for assignment to point objects.
- Wall supports for assignment to line objects.
- Soil supports for assignment to area objects.

On the basis of the object type and the associated support properties, the program generates spring elements at each mesh location.

Static Load Cases

Static loads represent actions upon the structure, and include force, pressure, and support displacement. A spatial distribution of loads upon the structure is called a load case.

Any number of static load cases can be defined. Typically you would have separate load cases for dead load, live load, pattern live load, static earthquake load, wind load, snow load, and so on. Loads that need to vary independently, either for design purposes or because of how they are applied to the slab, 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. The load values you assign to an object specify the type of load (e.g., force or displacement), its magnitude, and direction. Different loads can be assigned to different objects as part of a single load case and 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 down, or -Z direction. Point objects can accept concentrated forces or moments. Line objects may have uniform distributed forces, moments or torsions. Uniform (surface) loads can be applied to area objects. Vertical load cases may also include element self weight.

Some typical vertical load cases used for slabs might include:

- Dead load
- Superimposed dead load
- Live load
- Pattern live load
- Snow load

Lateral Load Effects

Although SAFE limits the deformations in the slab to out-of-plane behavior and thus is not appropriate for diaphragm studies, other effects caused by lateral loads may be considered using load and displacement assignments. For suspended slabs, specified slab deformations (rotations and displacements) may be assigned to columns and walls to account for frame behavior under lateral loads. If the user is using ETABS to perform a 3-D building analysis, ETABS allows for the direct export of floor loads and distortions to SAFE for further analysis and design.

Overturning on basemats or foundations resulting from lateral loads may be modeled by applying moments or moment couples to walls and moments and vertical loads to columns. SAFE provides a nonlinear iterative solution to model no-tension (uplift) behavior in the soil springs.

Load Combinations

Load combinations combine the results of previously defined load cases in an additive manner. SAFE allows combinations to be named. When a load combination is defined, it applies to the results for every object in the model.

Design is always based on load combinations, not directly on load cases. A combination that contains a single load case can be created. Each design algorithm creates its own default combinations, which can be supplemented with user-defined design combinations if needed.

Design Procedures

SAFE has two integrated design postprocessors: Concrete Slab Design and Concrete Beam Design. The concrete slab design procedure is applicable to area objects with slab property definitions, and the beam design procedure is available for line objects that have previously been assigned beam properties.

Design can be affected by the following:

- The design code, e.g., ACI 318-95, EUROCODE 2-1991, or BS8110 – 85 R 1989, among others.
- The design method and strength reduction factors.
- Load combinations for which the design should be checked.
- X- and Y- design strips for the slabs, which determine the layout of the calculated reinforcing. The average moments across the strips are used to calculate the required flexural reinforcing.

More Information

This chapter presented just a brief overview of some of the basic components of the SAFE model. Additional information can be found in the on-line Help facility available within the SAFE graphical user interface as well as in the other chapters of this manual and the other books in this volume.



Chapter 4

SAFE Modeling Techniques

SAFE offers an extensive and diverse range of tools to model a wide range of slab systems. This chapter will illustrate a few of the techniques to make modeling easy and quick using SAFE.

Overview of Slab Types

SAFE can be used to model almost any type of elevated slab or basemat/ foundation system, including two-way slabs, flat slabs, ribbed slabs, and waffle slabs.

Two-Way Slabs

Two-way slabs (flat plates and flat slabs) are probably the most common concrete slab systems, and are easily modeled using SAFE. Because SAFE is finite element based, when a two-way slab of arbitrary shape is drawn, the program automatically meshes the area objects into isotropic or orthotropic thin plate bending elements. Those elements are three- to four-node elements, with one vertical and two rotational degrees of free-

dom at each node, which capture out-of-plane bending behavior. Thus, two-way slab action is automatically accounted for in the SAFE model, and unlike equivalent frame techniques, requires no additional assumptions or calculations.

Flat Slabs

Flat slabs are typically two-way slabs with drop panels in the column regions to increase the shear capacity of the slab. It is usually advisable to model the geometric plan dimensions of the drops and columns with area objects so that the slabs will have the correct span lengths, remembering that moment varies directly with the square of the span length.

Both drop panels and columns are easily modeled in SAFE by drawing area objects and then assigning thickened slab properties with either a drop or column type identifier.

Ribbed Slabs

Ribbed slabs are slabs stiffened by ribs running in one predominant direction. Modeling this type of system in a conventional finite element program can be extremely difficult and time consuming because of the meshing effort involved. However, with SAFE's object oriented approach and automated meshing capability, ribbed slabs can be modeled simply by drawing the area object representing the slab, and assigning a "Ribbed" slab property type.

Ribs are defined by specifying their depth, their widths at the top and bottom, and their spacing. The program then calculates equivalent slab properties, such as the moment of inertias and self weight, using the slab and rib definitions. This calculation takes into account the T-beam behavior occurring when the slab and ribs work together, with the slab acting as a flange.

Although an equivalent slab property is used internally when performing the analysis, the actual geometry of the slab and ribs is used for design and detailing purposes.

Waffle Slabs

Waffle slabs are similar to ribbed slabs, but instead of ribs in only one direction, the waffle slab joists are laid out in an intersecting orthogonal grid. In addition, waffle slabs also typically use drop panels at the column locations. Similar to ribbed slabs, joists in a waffle slab are defined by specifying their depth, their widths at the top and bottom, and their spacing. Again, the program uses equivalent slab properties for the analysis, accounting for T-beam behavior, while maintaining the slab and joist geometries for design and detailing.

Slabs with Discontinuities

The slab release capability of SAFE allows for the convenient modeling of discontinuities. Releases allow jumps in moment or shear across a specific line of discontinuity. Releases are assigned to line objects.

Mat Foundations and Footings

Similar to elevated slabs, basemats and footings can be drawn using area objects to which slab properties are assigned. In addition to the thin plate element described for elevated slabs, SAFE also offers a thick plate element that incorporates shear deformation for slabs that have a large depth-to-span ratio, as is often the case with foundations. Stiffening walls can be incorporated, along with the correct spatial distribution of loads from columns (punching shear).

Soil springs can be added quickly to the model. Any number of soil support properties may be defined, so that the subgrade modulus can vary throughout the foundation. An optional iterative process to model no-tension (uplift) in the soil springs is also available.

Beam Types

The majority of beams in concrete construction are formed in a rectangular shape. However, beams can come in other shapes, and SAFE, in addition to providing the rectangular beam, offers a number of other

geometric beam options, including T Beams, Inverted T Beams, L Beams, and a General Beam option that can be used to model beams not reflected by one of the other categories.

Models Exported from ETABS

Often times a concrete floor system defined in ETABS will require further refinement during analysis (e.g., deflections based on cracked sections), as well as design and detailing. This can be accomplished easily using the link provided between ETABS and SAFE/CSiDETAILER. ETABS allows for the export of any story level as a SAFE.f2k text file.

In addition to the geometry and property assignments of the objects, ETABS provides three additional export options:

- Export floor loads only.
- Export floor loads and loads from above.
- Export floor loads plus column and wall distortions.

The last export option allows SAFE to include the effects of lateral loads.

More Information

This chapter was intended to illustrate just some of the many techniques SAFE provides for the efficient modeling of slab systems. Much more information may be found in the on-line Help and in the “Watch & Learn Movies” series.



Chapter 5

SAFE Analysis Techniques

This chapter provides an overview of some of the analysis techniques available within SAFE. The types of analyses described are linear static analysis, nonlinear analysis for uplift, and nonlinear analysis for cracking.

In a given analysis, a nonlinear uplift analysis or a nonlinear cracking analysis can be specified, but not both in a single run. A linear elastic analysis must always precede a nonlinear cracking analysis.

Overview of the Analysis Process

At the time of analysis, SAFE converts the object-based model created by the user into a finite element model, called the analysis model. The finite element mesh used in the analysis is a rectangular mesh based on a maximum acceptable element size. However, extra mesh lines are introduced at all locations of objects, object boundaries, and gridlines. Additional mesh lines can be introduced at specified locations by adding

gridlines. SAFE also has options to modify meshing by setting the maximum mesh dimension.

After the mesh has been generated, SAFE automatically creates slab elements by subdividing all the area objects that have been assigned slab properties. SAFE also creates beam elements by subdividing all the line objects that have been assigned beam properties.

Support properties are lumped into discrete springs and are assigned to finite element mesh points. Support properties of all point objects are assigned directly to the corresponding finite element mesh point. Support properties for all line objects are discretized and applied to all of the mesh points that exist on the line based on the tributary length associated with the mesh point. Similarly, support properties of all area objects are discretized and applied to all the mesh points that exist within the area and on the boundaries of the area, based on the tributary area associated with the mesh point.

Point loads of all point objects are assigned directly to the corresponding finite element mesh point. Line loads of all line objects are discretized and applied to all the mesh points that exist on the line based on the tributary length associated with the mesh point. Similarly, surface loads of all area objects are discretized and applied to all the mesh points that exist within the area and on the boundaries of the area based on the tributary area associated with the mesh point.

All of the internal meshing and assigning performed by the program does not alter the number or size of the objects, which allows revisions and modifications to be executed at the object level.

The Analysis Model

The analysis model consists of slab, beam and support elements. Each slab element is an isotropic or orthotropic, thin or thick plate bending element. The essential features of the plate elements are as follows:

- Each of the element nodes has one vertical and two rotational degrees of freedom.

- The material properties and thickness within each element are constant.
- Optionally, to model orthotropic effects, it is possible to specify three different effective thicknesses: X-direction bending, Y-direction bending, and twisting.
- The slab system must exist in the XY plane.
- In-plane action is not allowed in the XY plane; therefore, membrane stresses in the plane of the slab do not exist.
- The calculation of the self weight of the slab element is based on the design thickness, the XY dimensions of the element, and the unit weight of the element.
- Slab element moments and shears are calculated at the mesh points of the element.

The beam element in SAFE is based on linear elastic beam theory and can have properties based on the dimensions of the specified cross section or on the input cross-sectional properties, such as the moment of inertia. Other characteristics of beam elements are as follows:

- Beam elements are prismatic.
- In-plane action is not allowed in the XY plane; therefore, beam axial forces do not exist.
- The calculation of the self-weight of the beam is based on the actual area for rectangular or general sections and the stem area for T-, inverted T-, and L-sections, length between mesh points, and unit weight of the beam.
- The beam moments, shears, and torques are calculated at the two ends of the element, that is, at each mesh point.
- If any participation of the slab is to be included in the bending of any beam, specify the beam as a T-beam or an L-beam.

Column supports are modeled as linear elastic spring elements at the point object location. For wall and soil supports, the program generates

equivalent mesh point linear elastic springs. Optionally, you can activate an iterative process to model no-tension conditions in the soil supports. The fundamental features of the support elements are as follows:

- Column supports have three degrees of freedom: one vertical and two rotational.
- Wall supports have one vertical and one rotational (out-of-plane) degree of freedom.
- Soil supports have a single vertical degree of freedom.
- The support elements are weightless.
- The support reaction values are produced at every supported mesh point.

Linear Static Analysis

A linear static analysis is automatically performed for each static load case that is defined. The user may specify any number of load cases for analysis. The results of different static load cases can be combined with each other using load combinations. A linear static analysis is the default analysis in SAFE.

Nonlinear Analysis for Uplift

SAFE has an option to request nonlinear analysis to consider soil supports as providing compressive support only. This option requires that load combinations be defined before an analysis is requested. Nonlinear analysis is performed only for load combinations and not for individual load cases. SAFE applies an iterative procedure using the original stiffness and corrective load vectors to obtain the no-tension results.

This procedure converges asymptotically. The iterations for load combinations cease when a user-specified number of iterations has been reached or when the maximum tension pressure as a ratio of the maximum compressive pressure is below a user-specified convergence toler-

ance. When this option is activated, all program results for loading combinations are for this nonlinear analysis.

Nonlinear Analysis for Cracking

In the design of slabs, it is generally recognized that the distribution of moments obtained from an elastic analysis is appropriate for design purposes. However, an elastic analysis may significantly underestimate the true deflections.

The estimation of true deflections is a complex task. Some guidelines are available in design codes, such as the ACI 318-95 Section 9.5. For beams, that code recommends calculation of effective stiffness to obtain the cracked deflections and then application of modification factors to account for long-term deflections. SAFE uses the recommendation concerning effective stiffness for slabs and beams of floor systems to obtain cracked deflections. The procedure is as follows:

- Perform elastic (uncracked) analysis.
- Design the slab reinforcement.
- Calculate cracking moment and cracked moment of inertia for each element for each direction of moment and for top and bottom reinforcement.
- Calculate service level moments as the summation of dead and live moments.
- Calculate the ratio of effective stiffness to gross-section stiffness for both X and Y directions for the slab elements and for the beam elements.
- Reanalyze the structure using the effective stiffness calculated for each element.

The reanalysis results are saved for only the displacements and are reported as cracked deflections. All other results are for the original elastic (uncracked) analysis. The cracked deflection results should be treated as

estimates and need to be modified using appropriate factors to represent long-term deflections.

More Information

This chapter provides a general introduction to the primary analytical techniques SAFE offers for linear and nonlinear analysis of slabs. Much more information may be found in the on-line Help and in the “Watch & Learn Movies” series.



Chapter 6

SAFE Design Techniques

This chapter provides an overview of some of the design techniques available within SAFE. The types of design described are slab flexural design, slab punching shear check, and beam flexural and shear design. Please refer to the *SAFE Design Manual* for information about code specific design.

Overview of the Design Process

After the analysis has been completed, design may begin. Design loads are generated from built-in default load combinations for the specified code (users may also specify their own load combinations, if needed). Using those loads, the program designs both slabs and beams, and reports the calculated reinforcing on-screen.

For slabs, SAFE uses a simple layering system to separate the design from the analysis model. The layers define the design strips that SAFE uses to calculate the flexural slab reinforcing in accordance with a user-

selected code. SAFE also designs the flexural and shear reinforcing of any line object that has been assigned a beam type.

Slab Flexural Design

Similar to conventional design, the flexural design of slabs in SAFE involves defining sets of strips in two mutually perpendicular directions, integrating strip moments and shears (on the basis of nodal or Wood-Armer techniques), and then designing the strips in accordance with a user-selected code. Code specific design is described in the *SAFE Design Manual*.

The design of the slab reinforcement for a particular strip is completed at each transverse mesh line location along the length of the strip. Moments are integrated across elements of like property type to arrive at design moments. Controlling reinforcement is computed for each set of elements with the same property type, and then all of the reinforcements are summed to give the total reinforcing for the strip. The maximum top and bottom strip reinforcements for each side of a particular mesh line are obtained along with the corresponding controlling load combinations.

As an option, minimum reinforcement requirements may be included in accordance with a user-selected code. SAFE's slab design is intended to satisfy strength requirements only.

Design Strips

SAFE provides three layers for analysis and design: the Structural Layer, the X-Direction Strip Layer, and the Y-Direction Strip Layer. The Structural Layer is used to define the structural geometry, supports and loads—it is the analysis layer. The X- and Y-Direction Strip Layers are used for defining the design strips. The strips are, in fact, rectangular area objects, and each object is a design strip in the direction corresponding to the layer.

The strips can only run parallel to the X- or Y-axis of the grid system and must be perpendicular; skewed strips are not currently possible. The extent of the design strips is defined by the edges of the area objects defin-

ing the slab, and strips can overlap if needed. The locations of the design strips are usually governed by the locations of the supports. If the user does not define design strips and a design is requested, the program will provide default design strips on the basis of grid line locations.

Integration of Moments – Nodal

The total integrated cross-sectional moments and shears along the length of a design strip are obtained for design and display purposes. If nodal integration has been specified as the preference, the moments are integrated as follows:

- For a particular load combination or load case for each finite element within the design strip, the program calculates the nodal point reactive moments. Those moments are obtained by multiplying the slab element stiffness matrices by the element nodal displacement vectors.
- The nodal point moments for each set of slab elements with the same property type are then added to get a single positive and a single negative design moment for each set of element types, one mesh line at a time. If all of the elements across the strip have the same property assignment (same thickness and material), there would be only one positive and one negative moment.
- The positive and negative moments are totaled separately.
- This process is repeated for both sides of each mesh line.

The moments calculated using the nodal approach will always be in complete static equilibrium with the applied loads, irrespective of the refinement of the finite element mesh. This approach is best suited for use on a model where the mesh is poorly developed. However, the automated meshing procedure provided by SAFE makes the development of a poor mesh extremely unlikely, so generally, it is advisable to use the Wood-Armer integration scheme described in the next section.

Integration of Moments – Wood-Armer

Setting the preference to Wood-Armer for the integration of design moments results in the following steps:

- For a particular load combination or load case for each finite element within the design strip, the program calculates the design moments/unit width using the internal forces. Internal forces are converted to design moments/unit width in the following manner:

$$m_{by} = m_y + |m_{xy}|$$

$$m_{bx} = m_x + |m_{xy}|$$

$$m_{ty} = m_y - |m_{xy}|$$

$$m_{tx} = m_x - |m_{xy}|$$

  where m_x, m_y, and m_{xy} are internal moments per unit width, and m_{by} and m_{bx} are design moments per unit width for bottom reinforcement in the y and x directions, respectively, and m_{ty} and m_{tx} are the design moments for the top reinforcing in the y and x directions, respectively.

- The nodal point design moments are then calculated by taking the above design moments per unit width and multiplying them by one-half the associated element widths.

- The nodal point design moments for each set of slab elements with the same property type are then added to get a single bottom and a single top design moment for each set of element types, one mesh line at a time.

- The bottom and top moments are totaled separately.

- This process is repeated for both sides of each mesh line.

The design moments calculated using the Wood-Armer integration will most likely not completely satisfy equilibrium of the applied loads, but for a mesh that accurately captures the overall behavior of the slab, this integration scheme will provide a better design. This is the case because

the Wood-Armer integration scheme more effectively calculates design moments when cracking occurs diagonally to the design strip directions.

Slab Punching Shear Check

The distribution of stresses close to concentrated loads or reaction points in reinforced concrete slabs is quite complex. Punching shear is one particular failure mode recognized by design codes for which an elastic plate bending analysis may not provide adequate stress distribution. Most codes use empirical methods based on experimental verification to check against punching shear failures. SAFE automates this check for the more common geometries. The SAFE procedure for the punching shear check for each column for each design load combination is as follows (described with reference to ACI 318-95):

- Locate the critical section around the column or point load. The program reports whether it assumed the column to be an interior, edge, or corner column. This determination is based on minimum area of the critical section. The program does not currently account for changes in slab depth, presence of openings, beams or walls, and other discontinuities.
- Calculate the reactive force and moments at the column for the load combination.
- Calculate the distribution of shear stress around the critical section.
- Obtain the shear capacity of the critical section.
- Compare the shear stress distribution with the shear capacity. The comparison is reported as a ratio for the worst load combination. A value above 1.0 indicates failure.

The program currently does not consider any shear reinforcement in slabs or the presence of any prestressing.

Beam Flexural and Shear Design

In the design of concrete beams, SAFE calculates and reports the required areas of steel for flexure and shear based on the beam moments, shears and load combination factors. The beam design is executed on an element-by-element basis considering the moments and shears at each nodal point of the element. Following are some of the assumptions associated with the beam design:

- The beams are designed for flexure and shear. Design for any torsion that may exist in the beams must be investigated independently by the user.
- The reinforcement requirements reported by the program are based solely on strength considerations.
- The beam section is designed for the maximum positive and maximum negative factored moment envelopes obtained from all of the load combinations.
- Negative beam moments produce top steel. For all but the inverted T-beam, the beam is designed as a rectangular section.
- Positive beam moments produce bottom steel. In such cases, the beam can be designed as a rectangular- or T-beam.
- For the design of flexural reinforcement, the beam is first designed as a singly reinforced beam. If the beam section is not adequate, the required compression reinforcement is calculated.
- In designing the shear reinforcement for a particular element, the steps involve the determination of the factored shear force, the determination of the shear force that can be resisted by concrete, and the determination of the reinforcement steel required to carry the balance.

Output Display Options

The SAFE 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
- Design results output in text tables
- Export to other drafting and analysis programs

Named definitions of display views can be saved as part of the model. Combined with the use of groups, this can significantly speed up the process of getting results during model development.

More Information

This chapter provides a general introduction to the primary design capabilities SAFE offers for concrete slabs and beams. Much more information may be found in the *SAFE Design Manual*, the on-line Help and in the "Watch & Learn Movie" series.



Chapter 7

CSiDETAILER Detailing Options

This chapter provides an overview of some of the detailing options available within CSiDETAILER when used in conjunction with the SAFE program. More detailed information about the program is available in the CSiDETAILER *User's Manual*. Use CSiDETAILER to create a detail model and generate plans, sections and tables showing the placement and size of the reinforcing based on the user-selected code.

Overview of the Detailing Process

Detailing can be initiated after analysis and design have been completed within SAFE. Starting the CSiDETAILER program will initialize the detailing model using specified preferences (e.g., code, rebar sizes and shapes). Although default values for the preferences are provided, it is strongly recommended that the user review the values before starting CSiDETAILER using the commands on the SAFE Detailing menu.

After setting the preferences and starting the program, CSiDETAILER will generate all available drawings, and those drawing can then be

viewed and customized using the command available within the CSiDETAILER program. A CSiDETAILER project can include any number of drawings. Depending on available space on a drawing, any number of views can be added to a drawing. Views can consist of slab plans, slab sections, beam elevations, beam sections, and reinforcing tables and schedules.

The Detailing Model

The detailing model created by CSiDETAILER is an internal model used to generate the detailed objects, with input consisting of two parts:

- Data from the analysis and design of the user's model in SAFE
- Detailing options and preferences set by the user

The detailing model is initialized when the user starts the detailing process. It is strongly recommended that the user sets the detailing preferences before starting detailing to ensure that the drawings, views and reinforcement are produced with a consistent, acceptable appearance.

Output from the detailing model consists of two parts:

- Detailed area objects that represent slab, footing and mat members
- Detailed line objects that represent beam members

Detailing information that can be viewed in plans, sections, elevations, tables and schedules includes the number, the size, the shape and the location of reinforcing bars as calculated by CSiDETAILER based on the preferences. Preference settings include, among other items, code, smallest and largest rebar, minimum and maximum number of rebar, shortest and longest rebar, rebar cutoff (curtailment) rules, rebar shapes, and rebar marks.

Drawings

A drawing in CSiDETAILER refers to a sheet of a certain size on which views may be placed, and is used in a manner consistent with other CAD programs. A drawing may have any number of views showing the detailing information, such as plans, sections and tables, and the user may create any number of different drawings.

Drawings can be printed directly from CSiDETAILER, saved, or exported to a DXF or DWG file for use in CAD software. Different items on the drawings can be assigned to different layers for easy editing and display control when exported to a CAD program.

Drawings may contain detailing information about more than one CSiDETAILER model. After a drawing has been saved, it may be opened in another CSiDETAILER model and views may be added from that second model.

Views

Drawings consist of views. A drawing can have any number of views. The views available include the following:

- Slab plan
- Slab sections
- Beam elevations
- Beam sections
- Slab rebar table
- Slab rebar schedule
- Beam rebar table
- Beam rebar schedule

Views contain the actual detailing information. Views can be added, deleted and edited.

Tables and Schedules

In addition to plans, sections and elevations, tables and schedules may also be added to the drawings as views. There are rebar tables for slabs and beams, and rebar schedules for slabs and beams. Tables and schedules are similar, with schedules typically offering more information for the detailer. Tables and schedules may be deleted, moved or resized similar to any other views.

More Information

Much more information about CSiDETAILER may be found in the CSiDETAILER *User's Manual*, the on-line Help within CSiDETAILER and in the "Watch & Learn" movie series associated with the CSiDETAILER program.