COMPUTERS & STRUCTURES, INC.

STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

ЅAFE® 2016

Design of Slabs, Beams and Foundations
Reinforced and Post-Tensioned Concrete

Key Features and Terminology

CSi

SAFE®

DESIGN OF SLABS, BEAMS AND FOUNDATIONIS
REINFORCED AND POST-TENSIONED CONCRETE

Key Features and Terminology

Copyright

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND TESTING OF THIS SOFTWARE. HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THIS PRODUCT.

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

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

Contents

Chapter 1
Welcome to SAFE

1.1   Introduction

SAFE is a sophisticated, yet easy to use, special purpose analysis, design, and detailing program developed specifically for concrete slab and basemat systems. SAFE couples powerful object-based modeling tools with an intuitive graphical interface, allowing the quick and efficient modeling of slabs of regular or arbitrary geometry with openings, drop panels, post-tensioning, ribs, edge beams, and slip joints, supported by columns, walls, or soil. Design is seamlessly integrated with the modeling and analysis, and provides comprehensive reporting of the required reinforcement calculated based on a chosen design code. Detailed drawings may be produced effortlessly for slabs and beams designed using SAFE. SAFE may be used as a stand-alone application, or may be used in conjunction with ETABS to complete analysis, design, and detailing of concrete floor plates created in ETABS.

1.2 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, and walls. The loading is typically comprised of vertical point, line, and surface loads along with in-plane post-tensioning. 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 three decades ago. Early releases of SAFE provided input, output, and numerical solution techniques that took into consideration the modeling and analysis needs specific to concrete slabs, providing a tool that offered significant savings in time over general purpose finite element programs and that 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 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 slab systems. SAFE automates the analysis and design process for the structural engineer, resulting in more sophisticated designs produced with less engineering labor.

Creation and modification of the slab model, execution of the analysis, checking and optimization of the design, detailing of reinforcement, 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 shell elements, sophisticated post-tensioning loads, and the most recent concrete design codes from around the world.

1.3   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 that SAFE can easily handle:

- Flat slabs

- Flat slabs with perimeter beams

- Slabs with post-tensioning tendons

- Basemats

- Two-way slabs

- Waffle slabs

- Ribbed slabs

- Rectangular or circular slabs

- Geometrically complex slabs with multiple coordinate systems

- T-beam effects

- Spread footings

- Combined footings

- Slabs subjected to any number of vertical load patterns and combinations

- Pattern live loads

- Foundation uplift

- Cracked section analysis

- Walls with out-of-plane bending stiffness

- Slab reinforcement calculated based on user-defined design strips

- User controlled placement of slab rebar

- Flexural, shear, and torsion design of beams

- Punching shear checks and punching reinforcement design

- Dynamic analysis of floor systems and foundations by exporting response spectrum analysis results from ETABS to SAFE

- Design for twisting moments

- Reinforcement contours calculated using Finite Element forces

- Automatic transfer of geometry, loading, and support distortions from ETABS

- Post-tensioning and mild reinforcement detailing

- Material quantity takeoffs

- And much, much more!

1.4   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 necessary to analyze, design, and 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 analysis, design, and detailing 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.

- ▪ Concrete slab and beam reinforcement detailing module.

1.5   Modeling Features

The SAFE slab is idealized as an assemblage of area, line, tendon, and point objects. Area objects are used to model slabs, openings, soil supports, walls, ramps, and surface loads. Line objects model beams, columns, braces, and loads. Tendon objects are used to input post-tensioning loads. Point objects are used for concentrated loads. With relatively simple modeling techniques, very complex slab systems can easily be considered.

The geometry of the slab can be unsymmetrical and arbitrary, and the thickness of the slab may vary. Locations of supports and loads may be completely random and are 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 either or both sides of the line object that represents the joint.

Columns and walls provide both vertical stiffness and rotational stiffness to give a more accurate representation of the distribution of forces in the slab.

1.6   Analysis Features

Static analyses, including the effects due to post-tensioning, can be carried out for any number of user-defined load cases, and the load cases may be combined into any number of load combinations. Forces and deflections calculated may include those for both elastic and cracked sections, and load cases may include creep and shrinkage factors for nonlinear cracked deflection analyses. Hyperstatic analysis is also available and is based on a predefined static load case.

Basemats and foundations may be subjected to overturning moments that cause uplift. SAFE offers a nonlinear solution that accounts for zero ten-

sion in soil springs, as well as allowing soil moduli to vary throughout the model.

To account for orthotropic effects, different thicknesses may be specified in the local 1 and 2 directions when defining slab properties. This can be useful to model slabs that have primarily one-way behavior, where cracking may be predominant in one direction.

The analysis output may be viewed graphically, displayed in tabular output, sent to a printer, exported to a database or spreadsheet file, exported to a CAD file, or saved in an ASCII file. Types of output include slab, beam, and wall forces, reactions, displacements, and design reinforcement.

SAFE also provides dynamic analysis capabilities through modal frequency analysis and handling of response spectrum results exported from ETABS into SAFE. These capabilities allow for investigation of things such as floor vibrations and machine foundations.

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

1.7   Design Features

Flexural and shear design of reinforced and post-tensioned concrete slabs and basemats and the flexural, shear, and torsion design of beams can be performed based on a variety of international design codes. Design accounts not only for the stresses in the members due to post-tensioning, but also incorporates the tendons as reinforcement. Slab reinforcement location and layout is controlled using design strips that can be user-defined such that they may be non-orthogonal. Associated with these design strips are widths that can be automated or user-defined, and both column and middle strips may be specified. Design strip moments are obtained by integrating the finite element stresses using an algorithm that accounts for the effects of twisting moments. Users may also draw

their own slab reinforcement, and then ask the program to report where additional reinforcement is required. Code-based punching shear checks and if necessary, punching shear reinforcement design, are performed at columns, supports, and point loads.

1.8   Detailing Features

Drawings showing detailed reinforcement may be produced for both slabs and beams. The detailing may be based on program defaults, which represent general detailing based on the designed reinforcement, or on user-defined preferences. Any number of drawings may be prepared, containing plan views of reinforcement and tendon layouts, sections, elevations, tables, and schedules. Control over reinforcement bar sizes, minimum and maximum spacing, along with cut-off (curtailment) lengths is provided through detailing preferences. Drawings may be printed directly from SAFE or exported to DXF or DWG files for further refinement.

1.9   An Intuitive Process

The basic approach for using SAFE is very straightforward. The user establishes grid lines, places structural objects relative to the grid lines using columns, beams, tendons, walls, and slabs, and assigns loads and structural properties to those structural objects. Analysis, design, and detailing are then performed based on the structural elements and their assignments. Results are generated in graphical or tabular form that can be printed to a printer or saved to a file for use in other programs.

In using SAFE, you manage the **File**, **Edit** the model, change the **View**, **Define** properties or load patterns, **Draw** new objects in the model, **Select** existing objects, **Assign** properties or loads, **Run** the analysis and **Design** of the model, **Display** analysis results for checking, detail the structure by running the D**etailing**, and apply various **Options** to achieve the desired outcome with minimum effort. These actions are the basis for the program menu structure. Thus, familiarity with the menu commands and their function is vital to expanding your ability to use SAFE.

Subsequent chapters of this manual and the *Defining the Work Flow* manual describe many of the menu commands in greater detail. Familiarity with the sub-menus will enable creation of models for complex Flat Slabs with Openings, Slabs with Beams, Footings, and Mats.

The SAFE design manuals explain how SAFE performs concrete beam and slab design with both post-tensioning and mild reinforcement, in accordance with applicable design codes.

Information regarding the creation of detailed drawings may be found in the *Defining the Work Flow* manual.

SAFE Menu Commands:

- *File*
- *Edit*
- *View*
- *Define*
- *Draw*
- *Select*
- *Assign*
- *Design*
- *Run*
- *Display*
- *Detailing*
- *Options*
- *Help*

1.10   Work Flow

In the manual entitled, *Defining the Work Flow,* a basic work flow is defined that covers the typical steps necessary for the majority of projects. Each of the items listed below is discussed in detail and provides a broad overview of the basic modeling process:

1.   Set the Units

2.   Begin a Model

3.   Define Materials

4.   Define Properties

5.   Draw Objects

6.   Select Objects

7.   Assign Properties to the Model

8.   Load the Model

9.   Define Load Cases

10.   View & Edit Model Geometry

11.   Analysis and Design

12. Reinforcement Detailing

13. Display Results

14. Output Results and Reports

Chapter 2
Getting Started

2.1   Installing SAFE

Please follow the installation instructions provided in the separate instal-
lation document included in the SAFE package or ask your system ad-
ministrator to install the program and provide you access to it.

2.2   If You are Upgrading

If you are upgrading from an earlier version of SAFE, you should be
aware that the model is now defined in terms of objects, which are auto-
matically and internally meshed into elements during analysis.

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

2.3   About the Manuals

The SAFE documentation consists of three volumes. Volume I "*Using SAFE*" in turn consists of three manuals: *Key Features and Terminology, Defining the Work Flow,* and *Tutorial*. Volume II "*SAFE Design*" contains two manuals: *Post-Tensioned Concrete Design* and *Mild Reinforced Concrete Design*. Volume III is a single document entitled "*SAFE Verification.*" Figure 2-1 provides a graphical representation of the SAFE documentation structure.
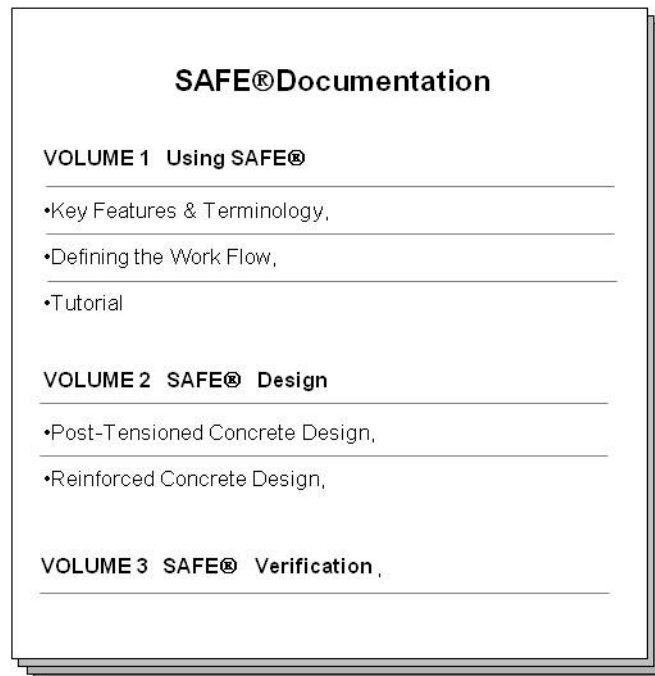
Figure 2-1 SAFE Documentation

This manual*, Key Features and Terminology,* provides overviews of the SAFE modeling, analysis, design, and detailing techniques, along with detailed descriptions of the SAFE interface. The *Defining the Work Flow* document offers an ordered description of the workflow process involved in using SAFE, from starting a model to detailing the designed structure. The *Tutorial* takes the user through the creation, analysis, and design of an example models.

Information covering the design theory and methods, in accordance with various international design codes, is provided in Volume II. *Reinforced Concrete Design* and *Post-Tensioned Concrete Design* features chapters on the various design codes included in SAFE, including the American, Australian, British, Canadian, Chinese (requires Chinese license), European, Indian, Hong Kong, New Zealand, and Singapore codes.

The *SAFE Verification* manual provides comparisons of SAFE output with independently produced or published results.

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

Additional information can be found in the Help facility that is accessible from within the SAFE graphical user interface.

2.4 "Watch & Learn™ Movies"

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

2.5 CSI Knowledge Base

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

2.6   Technical Support

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

- Consult the documentation and other printed information included with your product.

- Check the on-line Help facility in the software.

- Visit the CSI Knowledge Base at *https://wiki.csiamerica.com*.

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

- Send an email and your model file to *support@csiamerica.com* or your local CSI Partner.

- Visit CSI's website and Customer Support Portal at *https://www.csiamerica.com*.

- Call CSI or your local CSI Partner. Contact details are available at *https://www.csiamerica.com/contact*.

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

2.7   Help Us to Help You

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

- The product level (Standard or P/T) and version number that you are using. This can be obtained from inside the software 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.

- If calling, please be at your computer where you can run the software.

Chapter 3
The SAFE System

SAFE analyzes, designs, and details concrete slab systems that are created using the various drawing tools or imported into the graphical user interface. The key to successfully implementing SAFE is to understand the unique and powerful approach the program takes in modeling, designing, and detailing slabs. This chapter provides an overview of some of the special features and their associated terminology.

3.1   Physical Modeling Terminology

In SAFE, reference is often made 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 SAFE 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 during the analysis, 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 required both to create the model and 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 as it is the basis for creating models 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.

3.2   Structural Objects

As previously stated, SAFE uses objects to represent physical structural members. When creating a model, start by drawing the geometry and then assign properties and loads to completely define the slab structure.

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

- **Slab/Area objects** are used to model slabs, drop panels, column geometry, openings, soil supports, and surface loads.

- **Wall/Ramp objects** are used to model walls and ramps.

- **Beam/Line objects** are used to model beams and line loads.

- **Column objects** are used to model columns and braces.

- **Tendon objects** are used to model post-tensioning tendons and tendon loads. Even though they are drawn in plan similar to a beam object, they differ from beam objects in that they have a shape profile through the slab thickness.

- **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 point loads and point displacements, as well as for applying point restraints and springs.

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.

3.3   Properties

Properties are assigned to each object to define the structural or support behavior of that object in the model. Properties under the **Define menu**, namely slab, beam, reinforcing bar, tendon, column, wall, soil, and spring properties, 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, any changes to the definition of the property will automatically apply to the beam objects with this property assigned. A named property has no effect on the model unless it is assigned to an object.

Soil subgrade support properties may be assigned to slab/area objects, and for these properties, SAFE generates spring elements at each mesh location.

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

3.4 Load Patterns

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

Any number of load patterns can be defined. Typically, a model will have separate load patterns for dead load, live load, post-tensioning, pattern live load, static earthquake load, wind load, snow load, temperature 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 patterns.

After defining a load pattern name, you must assign specific load values to the objects as part of the load pattern. The load values you assign to an object specify the type of load (e.g., force or displacement), the magnitude, and the direction. Different loads can be assigned to different objects as part of a single load pattern and each object can be subjected to multiple load patterns.

3.4.1 Vertical Loads

Vertical loads may be applied to point, beam, and slab objects. Vertical loads are typically input in the down, or –Z direction. Point objects can accept concentrated forces or moments. Beam objects may have uniform or trapezoidal distributed forces, moments, and torsions, as well as concentrated point loads, moments, and torsions. Uniform and non-uniform surface loads can be applied directly to slab objects, including openings. Vertical load patterns also may include element self weight.

Some typical vertical load patterns used for slabs might include:

- Dead load

- Superimposed dead load

- Live load

- Pattern live load

- Snow load

3.4.1.1 Arrangement of Live Load

It is often desirable to apply live loads in varied geometric arrangements to model how these loads are applied to the real structure. Arranging live loads in critical patterns can increase the negative and positive moments in certain panels and columns. Common arrangements of loading include geometric patterns such as live loads on two adjacent panels and live loads on alternate panels.

SAFE allows for live loads to be geometrically arranged using two different options. Loading may be arranged using patterns defined by the user or by patterns created automatically by SAFE. In either case, the first step is to create a load pattern with a type of Pattern Live or Auto Pattern Live. If the pattern type is Pattern Live, SAFE will take each Pattern Live load pattern and combine them using the "Range Add" load combination feature described later. Multiple load patterns are typically required to capture alternate and adjacent panel loading arrangements for user-defined cases.

If the pattern type is Auto Pattern Live, the loads associated with the pattern will automatically be divided up internally into smaller "single panel" load patterns based on the panel grid created by the layout of the grid lines. Results from each of these "single panel" loads are then combined automatically by SAFE using the Range Add load combination type.

Using the Auto Pattern Live option, rather than the user Pattern Live option, allows for a varied arrangement of live loads to be generated with a single load pattern versus the multiple load pattern definitions required with the user option.

3.4.2 Lateral Loads

SAFE allows deformations in both the in-plane and out-of-plane directions of the slab. It can handle vertical loads as described previously or horizontal loads such as internal post-tensioning or external wind. Be-

cause in-plane deformations can occur, SAFE is appropriate for diaphragm studies. 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 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 of 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 an explicit nonlinear solution to model no-tension (uplift) behavior in the soil.

3.4.3   Dynamic Loads

If dynamic response spectrum loads are exported from ETABS along with a slab that is imported into SAFE, an additional set of load patterns will be imported. A separate load pattern will be defined for each mode, defined as type Other. These load patterns are used by an automatically generated response spectrum load case that is created during the import process. Note that response spectrum load cases can not be defined directly in SAFE.

3.4.4   Temperature Loads

SAFE allows temperature loads to be applied to slab objects, which results in thermal strains. These strains are given by the product of the Material coefficient of thermal expansion and the temperature change of the slab object. The temperature change may be specified as a gradient through the thickness of the object, or as a constant, if the top and bottom temperature changes are the same.

3.5   Load Cases

A load case defines how loads are to be applied to the structure, and how the structural response is to be calculated. SAFE automatically generates a load case for each load pattern that is defined. In addition, user-defined load cases containing multiple load patterns can be created. Any number of named load cases may be defined.

Analyses are classified in the broad sense as either linear or nonlinear, depending upon how the model responds to the loading. The results of linear analyses may be superposed, i.e., added together after analysis using combinations (see "Combinations" later in this chapter).

In addition to the typical static load cases, SAFE also provides the capability to carry out modal and hyperstatic analyses. Response spectrum load cases can also exist if imported from ETABS.

The results of nonlinear analyses normally should not be superposed. Instead, all loads acting together on the structure should be combined directly within a load case.

3.6   Load Combinations

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

The design is always based on load combinations, rather than directly on load cases or load patterns. A combination containing only a single load case can be created. The design algorithm creates its own default load combinations based on the selected design code, which can be supplemented with any number of user-defined load combinations if required.

If a load pattern with a type of Pattern Live has been created, force quantities will be combined automatically using the Range Add load combi-

nation type. The Range Add combination calculates minimum and maximum force quantities for each element.

3.7   Design Procedures

SAFE has two integrated design post-processors, namely Post-Tensioned Concrete Design (if licensed) and Reinforced Concrete Design; both design slabs and beams. The concrete slab design procedure is applicable to slab objects and the beam design procedure is available for beam objects. Punching shear checks are also carried out at column object locations, point loads, and supports. Stud rail and rebar tie design options are available when the punching shear capacity of the slab is not adequate.

Design can be affected by the following:

- The design code, e.g., ACI 318-08, Eurocode 2-2004, or BS8110-97, among others.

- The post-tensioning loads.

- The design method and strength reduction factors.

- The combinations for which the design is carried out.

- Pattern Live load patterns; these load patterns are used only in strength design.

- A and B design strips for the slabs, which determine the layout of the calculated reinforcement. The total moment at each section of the strips is used to calculate the required flexural reinforcement.

- The punching perimeter.

3.8   Detailing Procedures

SAFE includes the capability of generating detailed reinforcement drawings for slabs and beams. The detailing utilizes the analysis and design results to determine the required reinforcement quantities. For slabs, the

design strips define the orientation of the reinforcement. The detailed re-inforcement can be modified as desired and detailed component views can be compiled into drawing sheets that can be printed directly from SAFE or exported to CAD software for further manipulation.

3.9   More Information

This chapter provides a brief overview of the basic components of the SAFE system. Additional information can be found in the program Help and in the "*Watch & Learn™ Movies*" series. The other chapters of this and the other manuals provide more detailed explanation of the components introduced in this chapter.

Chapter 4
SAFE Modeling Features

SAFE offers an extensive and diverse range of tools to model a wide variety of slab systems. Slab models can be created directly in the SAFE interface, imported from ETABS, or defined using a drawing imported from a CAD application. This chapter illustrates a few of the techniques to make modeling quick and easy using SAFE.

4.1   Overview of the Modeling Process

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

- SAFE is optimized for modeling slab systems through tailored modeling procedures and design capabilities for concrete slabs and beams.

- SAFE models are object-based and consist of point, line, tendon, and area objects, to which assignments are made to define structural members, such as slabs, walls, ramps, beams, tendons, columns, braces, and supports, as well as to define loads.

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

- Define materials using the **Define menu**.

- Define slab, wall (ramp), beam, column (brace), and tendon properties (sections) and supports using the **Define menu**.

- Draw a series of slab, wall, beam, column, tendon, and point objects that represent your slab system, using the various drawing tools available within the graphical interface.

- Assign structural properties, supports, and loads to objects using the **Assign menu** commands. Note that the assignment of structural properties and supports may be completed concurrently with the drawing of the objects using the floating forms that appear when the draw commands are used.

When the model is complete, the analysis may be run.

Note: When a form "floats," the form will remain visible when the cursor is moved over the active window or the mouse button is clicked. This means that changes can be made to the form without reusing the command required to bring up the form, which differs from other types of forms. Floating forms are used in the draw mode so that the type of object being drawn, the properties, and other associated attributes can be changed easily while drawing objects for the model.

4.2   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.

4.2.1   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, SAFE automatically meshes the area objects into isotropic or or-

thotropic shell bending elements. These elements are three- or four-node elements, with six degrees of freedom at each node. The shell elements capture out-of-plane bending and shear behavior, as well as in-plane deformations due to post-tensioning and other loads. Thus, two-way slab action is automatically accounted for in the SAFE model and unlike equivalent frame techniques, requires no additional assumptions or calculations.

4.2.2 Flat Slabs

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

Drop panels modeled using area objects are easily generated by SAFE using the **Draw menu > Quick Draw Areas Around Points** command. Drop panels also can be defined as part of the column property definition, in which case they are automatically generated when columns are drawn.

4.2.3 Ribbed Slabs

Ribbed slabs are 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. SAFE 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.

4.2.4 Waffle Slabs

Waffle slabs are similar to ribbed slabs, but instead of ribs in only one direction, the waffle slab ribs are laid out in an intersecting orthogonal arrangement. Waffle slabs can easily be modeled by drawing the area object representing the slab and assigning a "Waffle" slab property type. In addition, waffle slabs often use drop panels at the column locations. Similar to ribbed slabs, ribs in a waffle slab are defined by specifying their depth, their widths at the top and bottom, and their spacing. Again, SAFE uses equivalent slab properties for the analysis, accounting for T-beam behavior, while maintaining the slab and rib geometries for design and detailing.

4.3 Mat Foundations and Footings

Similar to elevated slabs, basemats and footings can be drawn using slab objects. The shell element in SAFE incorporates shear deformation, therefore handling slabs that have a large depth-to-span ratio, as is often the case with foundations. Working in 3-D allows for the accurate representation of stiffness and loading from walls and columns, including overturning moments.

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

4.4 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, to either one or both sides. Shear and moment releases also can be assigned to specific slab edges.

4.5 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, inverted L-beams, and a general beam option that can be used to model beams not reflected by one of the other categories. If the slab is to be included in the bending of any beam, it can be modeled as either a flanged or rectangular beam. If a rectangular beam is used, SAFE will automatically account for the T-beam action provided by the slab. If a flanged beam is modeled, SAFE ignores the flange of the T-beam or L-beam where it is duplicated by the slab, and uses the slab to account for the T-beam action.

4.6 Post-Tensioning

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

The post-tensioning option in SAFE allows for prestressing loads to be applied to the model. Post-tensioning tendons may be drawn either explicitly by the user, in which case the tendon layout, force and profile are specified, or the post-tensioning layout can be determined by SAFE. The tendon distribution is specified as banded or distributed, and acceptable ranges for both the concrete precompression level and percentage of self weight to be balanced are input. SAFE then iterates to determine the pro-

file of the tendons that best satisfies the stress and balancing requirements.

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

4.7   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. 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 two additional export options:

- Export floor loads only.

- Export floor loads and loads from above.

Upon importing the ETABS generated model, additional modifications can be made to the model using the SAFE tools. It should be noted that changes in SAFE can not be exported back to ETABS, so any changes to the slab that may affect the ETABS global building model also should be applied to the ETABS model.

4.8   More Information

This chapter provides an overview of the modeling features that SAFE offers for simulating slab systems. Additional information may be found in the program Help and in the "*Watch & Learn™ Movies*" series.

Chapter 5
SAFE Analysis Features

This chapter provides an overview of the key analysis features available within SAFE. The analysis features described include linear static analysis, nonlinear analysis for uplift, and nonlinear analysis for cracking.

In a given run, any number of different analyses may be performed, including both linear and nonlinear analyses.

5.1 Overview of the Analysis Process

At the time of analysis, SAFE converts the object-based model into a finite element model, referred to as the analysis model. The finite element mesh used in the analysis is based on a user-defined maximum element size. However, additional meshing lines are automatically introduced at all locations of objects, object boundaries, and gridlines. Additional user-defined meshing lines can be introduced at specified locations by adding additional gridlines or adding line objects with NONE properties and then assigning them to be included in the analysis mesh.

SAFE automatically creates slab elements by subdividing all of the slab objects. SAFE also creates beam elements by subdividing all of the beam objects.

Column and wall objects provide vertical, as well as lateral support to the slab structure. Alternatively, point or line springs can be used to support the slab structure.

Soil subgrade modulus properties assigned to area objects are lumped into discrete springs and applied to all of the mesh points that exist on the area, based on the tributary area associated with each mesh point.

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

The internal meshing performed by SAFE does not alter the number or size of the objects, which allows for simple revisions and modifications to be executed at the object level, as well as simple extraction of analysis results.

5.2   The Analysis Model

The analysis model consists of slab, beam, column, wall, tendon, and support elements.

Each slab element is an isotropic or orthotropic, shell element. The key features of the shell elements are as follows:

- Each element node has six degrees of freedom.

- The material property within each element is constant.

- The element includes shear deformation.

- The thickness of each element is constant, unless orthotropic effects are considered, for which it is possible to specify three different effective thicknesses: local 1-axis bending, local 2-axis bending, and twisting.

- The element must lie in the horizontal XY plane.

- The calculation of the self weight of the element is based on the thickness, the planar area, and the unit weight of the material associated with the element.

- The element moments and shears are calculated at the nodes of the element.

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

- Beam elements are prismatic.

- The calculation of the self weight of the element is based on the actual area for rectangular or general sections, and the stem area for flanged sections, the element length, and the unit weight of the material associated with the element.

- The beam moments, shears, and torques are calculated at the two ends of each element, corresponding to each mesh point.

- The beam must lie in the horizontal XY plane.

The column/brace element is similar in formulation to the beam element. The main difference is that columns/braces must be vertical/diagonal, design results are not presented for them, and therefore they are not designed in SAFE.

Wall/ramp elements are modeled as shell elements, similar to the slabs. Similar to columns/braces, walls/ramps must be vertical/inclined, are not designed, and do not have design results presented.

For soil supports, SAFE generates equivalent linear elastic springs at mesh points. Compression only (zero tension) conditions in the soil supports are modeled using nonlinear gap elements. The key features of the support elements are as follows:

- The support elements are weightless.

- Soil supports have a single vertical degree of freedom.

- The support reaction values are calculated at every supported mesh point.

5.3 Linear Static Analysis

A linear static load case is automatically created for each load pattern that is defined. Any number of load patterns and load cases may be defined. Load cases can include multiple load patterns, each with an associated scale factor. The results of different static load cases can be combined using load combinations. A linear static analysis is the default and most common load case type in SAFE.

5.4 Nonlinear Analysis for Uplift

SAFE has an option to request that a nonlinear load case be used to consider soil supports as providing compression only. This capability can be extremely useful for analyzing basemats and footings.

If the nonlinear uplift analysis type is specified for a load case, nonlinear gap elements are automatically activated in the soil support springs. The gap elements can transfer only compressive loads, hence capturing any uplift behavior. This analysis type requires that any loads acting in combination be added together in the load case, and not in combinations.

5.5   Nonlinear Analysis for Cracking

A nonlinear cracking analysis for slabs may be executed based on user-specified rebar using Slab Rebar objects, on reinforcing calculated by the program using the Finite Element Based design, or on reinforcement specified using the Quick Rebar Specification option.

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 resulting from cracking.

The estimation of true deflections is a complex task. Guidelines are available in some design codes, such as ACI 318 Section 9.5. For non-prestressed, one-way construction, ACI 318 recommends the calculation of an effective stiffness to obtain the cracked deflections and further application of modification factors to account for long-term deflections.

SAFE has the capability to use the recommendation described above for calculating the effective stiffness of slabs and beams of floor systems when doing analysis. This technique requires that the reinforcement be known. Slab reinforcement may be calculated by SAFE or be user-defined. The automated procedure for calculating the effective stiffness is as follows:

▪ Perform an elastic (uncracked) analysis.

▪ Determine the slab reinforcement from the Finite Element Based design or from user input.

▪ Design the beam reinforcement.

▪ Calculate the cracking moment and cracked moment of inertia for each element, for each direction of moment, and for top and bottom reinforcement.

▪ Calculate the ratio of effective stiffness to gross-section stiffness for both local 1 and 2 directions for the slab and beam elements.

- Reanalyze the structure using the effective stiffness calculated for each element.

The results for nonlinear cracked analyses include both the displacements and stresses, although again, most design codes allow for the use of stresses from elastic, or uncracked analyses.

5.6 Nonlinear Analysis for Long-Term Cracking

A nonlinear, long-term cracking analysis can be carried out in a manner similar to the normal cracking analysis. The long-term analysis requires creep coefficient and shrinkage strain values in order to account for creep and shrinkage.

5.7 Modal Analysis

A modal analysis load case can be defined to compute the modal frequencies or periods of the structure. This analysis type is best used with vertical mass. The modes can be based on Eigen or Ritz vectors, and a minimum and maximum number of requested modes can specified.

5.8 Hyperstatic Analysis

A hyperstatic analysis can be defined as a load case. This type of analysis is based on an already defined static load case, in which the reactions from the static load case are applied as loads to an unsupported structure.

5.9 More Information

This chapter provides a general introduction to the primary analytical features that SAFE offers for linear and nonlinear analysis of slabs. Much more information may be found in the program Help and in the "*Watch & Learn™ Movies*" series.

Chapter 6
SAFE Design Features

This chapter provides an overview of the key design features available within SAFE. The design features described include slab flexural design with or without post-tensioning, slab punching shear checks and reinforcement design, and beam flexural, shear, and torsion design with or without post-tensioning.

6.1 Overview of the Design Process

After the analysis is complete, the design can be performed. Strength design loads are generated from built-in default combinations for the currently selected design code (user-defined load combinations also can be defined, if needed). Default serviceability combinations also are available if the model includes post-tensioning. Using these combinations, SAFE designs both slabs and beams, and reports the required reinforcement.

For slabs, SAFE uses either design strips or the finite element based design to calculate the slab flexural reinforcement in accordance with the selected design code. SAFE also designs the flexural, shear, and torsion reinforcement of any beam object.

6.2   Slab Flexural Design

Similar to conventional design, the flexural design of slabs in SAFE involves defining sets of strips in two directions (A and B), integrating the strip moments and shears (on the basis of the Wood-Armer technique), and then designing the required reinforcement for each strip in accordance with the selected design code. As an alternative to this strip based method, SAFE also offers a FEM based design, which is useful for irregular slabs where strip based techniques may not be appropriate.

6.2.1   Design Strips

SAFE provides two principal strip layers used for analysis and design, namely Layer A design strips and Layer B design strips. The strips are defined as polylines with associated widths that can vary along the length of the strip.

The strips need not be mutually perpendicular. The extent of the design strips is defined by the edges of the area objects defining the slab, and strips can overlap if needed. The location of the design strips is usually governed by the support or grid locations. Design strips may be defined as either Column Strips or Middle Strips – models with post-tensioning typically have column strips only.

6.2.2   Integration of Moments – Wood-Armer

Use of the Wood-Armer method for the integration of design moments results in the following steps:

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

$$m_{bB} = m_B + |m_{AB}|$$

$$m_{bA} = m_A + |m_{AB}|$$

$$m_{tB} = m_B + |m_{AB}|$$

$$m_{tB} = m_A + |m_{AB}|$$

where m_A, m_B, and m_{AB} are internal moments per unit width, and m_{bA} and m_{bB} are design moments per unit width for the bottom of the slab in the A and B directions, respectively, and m_{tA} and m_{tB} are the design moments for the top of the slab in the A and B directions, respectively. Design moments may be factored combinations for strength design, or unfactored moments for service load checks when post-tensioning is present. 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 provides a good design. This is the case because the Wood-Armer integration scheme effectively calculates design moments when cracking occurs diagonally to the design strip directions.

6.2.3 Required Reinforcement – Strip Based

If post-tensioning is present, stresses are checked at the slab top and bottom for unfactored service level loads. Stress checks are carried out at three stages, namely at transfer of the prestress load (transfer), after all losses have occurred and service loads are applied (normal), and after all losses have occurred and long-term service loads are applied (long term). These stress checks are carried out at each transverse mesh line along the length of the strips. The moments are integrated across all elements of like property in the strip to determine the service moments, which are converted to stresses assuming a linear strain variation through the section depth. If the stress limits are exceeded, mild reinforcement is added to carry the tensile stresses, as long as they are below the code specified limits, which can be adjusted in the Design Preferences. Any added mild reinforcement is then used with the user-defined post-tensioning tendons to calculate a nominal moment capacity, which is checked against the factored moments from the strength combinations.

The design of the mild reinforcement for slabs without post-tensioning also is carried out for a particular strip at each transverse mesh line loca-

tion along the length of the strip. The moments are integrated across all elements of like property in the strip to determine the factored design moments. The required reinforcement for slabs is computed for each set of elements with the same property type, and then summed to give the total required reinforcement for the strip. The maximum required top and bottom strip reinforcement for a particular mesh line is obtained, along with the corresponding controlling combinations.

As an option, minimum reinforcement requirements may be enforced in accordance with the selected design code.

6.2.4 Required Reinforcement – FEM Based

The FEM Based design determines the required reinforcement on an element-by-element basis and is therefore independent of design strips. Several options are available for refining the display of the required reinforcement, including averaging, which will flatten peaks and provide an averaged required reinforcement over a particular area of the slab.

6.2.5 Slab Rebar Objects

SAFE allows users to place the size, quantity, and location of slab reinforcing using Slab Rebar objects. Although this user-defined reinforcing does not directly affect design results, it can be used when performing a nonlinear cracking analysis, as well as compared to the required reinforcing calculated by the program using the Strip Based or FEM Based methods.

6.3 Slab Punching Shear Check

The distribution of stresses close to concentrated loads or reaction points in reinforced and prestressed 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 common geometries. If the check results in a punching shear ratio greater than unity (i.e., punching failure), SAFE will design punching shear reinforcement. The SAFE procedure for the punching shear check carried out for each column, for each design combination is as follows:

▪ Locate the critical section around the column or point load. SAFE reports whether it assumed the column to be an interior, edge, or corner column. This determination is based on whether the slab is present within 10 times the slab thickness along the column edges. A column is classified as a corner column when two slab edges are found within 10 times the slab thickness. A column with only one slab edge within 10 times the slab thickness is classified as an edge column, and a column is classified as interior when no slab edges are found within 10 times the slab thickness. For single footings, this determination is based on the minimum area of the critical section.

▪ Check that each slab element in the area enclosed between the face of the column and the critical section for punching shear has the same slab property label. If this is not the case, the slab thickness from the smallest area within the punching shear perimeter is used.

▪ Use the net shear to check punching shear if a point load or column (call it load/column A) is within the critical section for punching shear for another point load or column (call it load/column B).

▪ Calculate punching shear based on net forces in the slab when line objects (beams, walls, or releases) frame into a column.

▪ Calculate the reactive force and moments at the column for the combination. The shear and moment values used in the punching shear check are reduced by the load (or reaction) that is included within the boundaries of the punching shear critical section.

▪ 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 combination. A value above 1.0 indicates failure.

SAFE designs rebar ties or stud rails when such options are activated in the punching shear design overwrites. The details of rebar ties or stud rails are documented in the *Reinforced Concrete Design Manual* and the *Post-Tensioned Concrete Design Manual*.

Design overwrites are available to modify the location type, punching shear perimeter, openings in the slab, and reinforcement pattern, when the punching shear parameters computed need to be changed.

For computing the punching parameters, the following assumptions are used:

- Punching shear is calculated for columns punching through a slab or a drop panel. SAFE also checks the drop panel punching through a slab. The effect of column capitals is included in the punching shear calculation.

- SAFE uses the effective depth for computing the punching shear. The concrete cover to rebar is taken from the design preferences unless a design strip is present. In that case, the rebar cover is taken from the design strip.

- Openings within 10 times the slab thickness are automatically included in the punching shear calculations. The slab edge within the punching zone radius is subtracted from the punching shear perimeter.
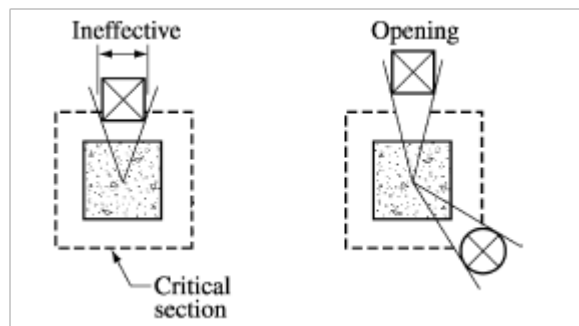
The vertical component of prestressing force is currently ignored in computing the punching shear capacity.

6.4   Beam Flexural, Shear, and Torsion Design

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

- The beams are designed for major axis moment, shear, and torsion only. Design for axial force or minor axis moment and shear that may exist in the beams must be investigated independently by the user.

- The required reinforcement reported by SAFE does not consider serviceability requirements.

- If post-tensioning is present, stresses are checked at the beam top and bottom for unfactored service level loads. Stresses are checked at the transfer of the prestress load without losses, with all service loads after losses, and with sustained service loads after losses. These checks are done assuming a linear strain variation through the section depth. If necessary, reinforcement is added to carry tensile stresses. Any added reinforcement is used with the user-defined post-tensioning tendons to calculate a nominal moment capacity, which is checked against the factored moments from the strength combinations.

- The beam section is designed for the maximum positive and maximum negative factored moment envelopes obtained from all strength combinations.

- Negative beam moments produce tension in the top fibers. For all but the inverted flanged beam, the beam is designed as a rectangular section.

- Positive beam moments produce tension in the bottom fibers. In such cases, the beam can be designed as a rectangular or a flanged beam.

- For the design of flexural reinforcement in beams without post-tensioning, 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 include the determination of the factored shear force, calculation of the shear force that can be resisted by the concrete, and calculation of the shear reinforcement required to carry the balance. The effects of post-tensioning are included.

- In designing the torsion reinforcement for a particular element, the steps typically include the determination of the factored torsion, calculation of the critical torsion below which torsion reinforcement is not required, calculation of special torsion section properties, calculation of the torsion reinforcement required as closed stirrups, and a check of the shear and torsion interaction.

6.5   Results Output

The design results can be viewed and saved in many different ways, including:

- Display and printing of two- and three-dimensional views of the model showing design results

- Design results output in tables, compatible with Microsoft Excel or Access

- Design results tables printed to RTF, HTML, and text files

- Custom reports with design results tables, graphics, and text

- Export of model to DXF, including design results

▪ Summary design information on an object-by-object basis by right clicking an object while design results are displayed. This information can be printed or saved to a RTF file.

6.6　More Information

This chapter provides a general introduction to the primary design features that SAFE offers for concrete slabs and beams. Additional information may be found in the program Help and in the "*Watch & Learn™ Movies*" series. Please also refer to the *Reinforced Concrete Design Manual* or the *Post-Tensioning Concrete Design* manual for information about code-specific design.

Chapter 7
SAFE Detailing Features

This chapter provides an overview of the key detailing features available within SAFE. The detailing features described include detailing preferences, drawing component views, and drawing sheets, as well as how these can be edited within the SAFE interface, printed, or exported to CAD software.

7.1 Overview of the Detailing Process

Detailing can be run after the analysis and design have been completed. The detailing is run using the **Run menu > Run Detailing** command.

The first time the detailer is run for a particular model, SAFE will generate a default set of drawings. Those drawings can be viewed, customized, printed, and exported from the Detailing tab of the *Model Explorer* window or the **Detailing menu > Show Detailing** command. Additional drawing sheets can be added and component views can be added or deleted from drawing sheets to create a customized set of drawings. If modifications are made to the model, including geometry, loading, or properties, the detailing can be rerun with the option of generating a new

set of drawings from scratch or synchronizing the new results with the previously customized drawings.

7.2   Detailing Preferences

To complete detailing in SAFE, analysis and design data and detailing preferences are required. Default detailing preferences are built into SAFE. These default preferences are typical values that should satisfy the majority of projects, but should be reviewed to ensure they are relevant for the project under consideration.

Some of the key detailing preferences that can be set in SAFE include the following:

- Dimension and material quantity units

- Bar callout style and content

- Drawing size, layout, scale, and so forth

- Rebar selection rules, including min/max bar sizes, spacing, and the like

- Preferred rebar sizes

- Curtailment rules

- Drawing display styles, including line type, weight, color and text size, font, alignment, color, and so on, for each type of drawing entity

The **Detailing menu > Detailing Preferences** command allows specification of the units and bar mark styles. The **Detailing menu > Slab/Mat Reinforcing Preferences** and **Beam Reinforcing Preferences** each provide selection rules, curtailment rules, and rebar call notations for slabs and beams, respectively. The **Detailing menu > Drawing Sheet Setup** command allows the properties of the drawing sheet to be set, including size, margins, and scales. Finally, the **Detailing menu > Drawing Format Properties** provides settings for all of the display styles, such as line types, weight, color, and text properties.

7.3 Drawing Component Views

Drawing component views are the building blocks of drawing sheets. Each view exists only once, but can be included on multiple drawings. A selection of the component views available include the following:

- Slab plans – framing, top bars, bottom bars, all bars

- Slab sections

- Beam framing plan

- Beam elevations

- Beam sections

- Slab rebar table

- Slab rebar schedule

- Beam rebar table

- Beam rebar schedule

- Tendon layout plan

- Tendon schedule

Additional component views can be found on the Detailing tab of the *Model Explorer* window. The component views generated depend on the objects in the model.

7.3.1 View Properties

Each drawing component view has a set of properties defining the scale, text and line sizes, and the caption. These properties can be set in the detailing tree of the *Model Explorer* by right clicking on a drawing component view and choosing the **Edit View Properties** command. The properties also can be set for each component view on the drawing sheets in the same way.

7.3.2 View Text

The text on each component view can be edited and rearranged (moved) by right clicking on the component view in the tree of the *Model Explorer* and choosing the **Edit View Text** command. Additional text, dimension lines, and notes also can be added. This feature should be used to rearrange overlapping text and add any additional notes or dimensions as necessary.

7.4 Drawing Sheets

A drawing sheet in SAFE is defined with a certain size, containing one or more drawing component views and is used in a manner consistent with other CAD programs. Additional drawing sheets can be added or existing sheets can be deleted.

Drawing component views can be added, deleted, or moved around on the drawing sheets. Multiple component views can be added to a drawing by right clicking on the drawing and using the various **Add Views** commands. Alternatively, while a drawing sheet is in the display, drawing component views can quickly and easily be added by dragging and dropping them from the tree in the *Model Explorer* window.

Drawing sheets can be printed directly from SAFE, saved, or exported to DXF or DWG files for use in CAD software.

7.5 Reinforcement Editing

SAFE provides capability for editing the detailed reinforcement that is shown in the drawing component views and drawing sheets. For slabs this is accomplished on a strip-by-strip basis while beams are handled one beam at a time. Using the edit reinforcement commands, the reinforcement location and length can be modified with simple click and drag mouse operations. The bar size and spacing also can be edited by clicking on any bar call (mark) and making the necessary changes. Note, changes should be made with care to ensure that the reinforcement quantity satisfies the required reinforcement for the design loads.

7.6   Synchronization

After the detailing has been generated the first time for a model, SAFE has the ability to synchronize and update the existing component views and drawings for subsequent runs of the detailing. The synchronization process maintains previous modifications and edits, to the extent practicable. The following is a summary of the situations when synchronization is carried out:

- Changes to the detailing results caused by changes to the detailing preferences, drawing preferences, or component view properties.

- Editing of section cuts, where possible.

- Editing of text location, where possible. This is determined based on a comparison of the unedited text position on the previous and new detailed items.

- Editing of reinforcement, as long as the object geometry has not been changed.

In general, it is recommended that if using the synchronization capability, the final drawings be reviewed for correctness.

7.7   More Information

This chapter provides a general introduction to the primary detailing features that SAFE offers. Additional information may be found in the program Help and in the "*Watch & Learn™ Movies*" series.

Chapter 8
The SAFE "Screen"

This chapter provides an overview of the key features of the graphical user interface (GUI). As the model is developed using the graphical user interface, a database file is created concurrently by SAFE. If desired, the database can be directly edited using the **Edit menu > Interactive Database Editing** command. While this feature is well suited to some situations, the vast majority of model creation and modification is completed using the graphical user interface.

8.1   The Graphical User Interface

The graphical user interface, shown in Figure 8-1, includes the main window, main title bar, menu bar, toolbars, model explorer, display window, display title bar, status bar, mouse pointer coordinates, and a units button. Each of these items is described as follows.

Main Window – This window may be moved, resized, maximized, minimized, or closed using standard Windows operations. Refer to Windows help, available on the Start menu for additional information on performing these actions.

Main Title Bar – The main title bar includes the program and model names. The main title bar is highlighted when the program is in use. Move the main window by left clicking in the main title bar and holding down the mouse button to drag the window.
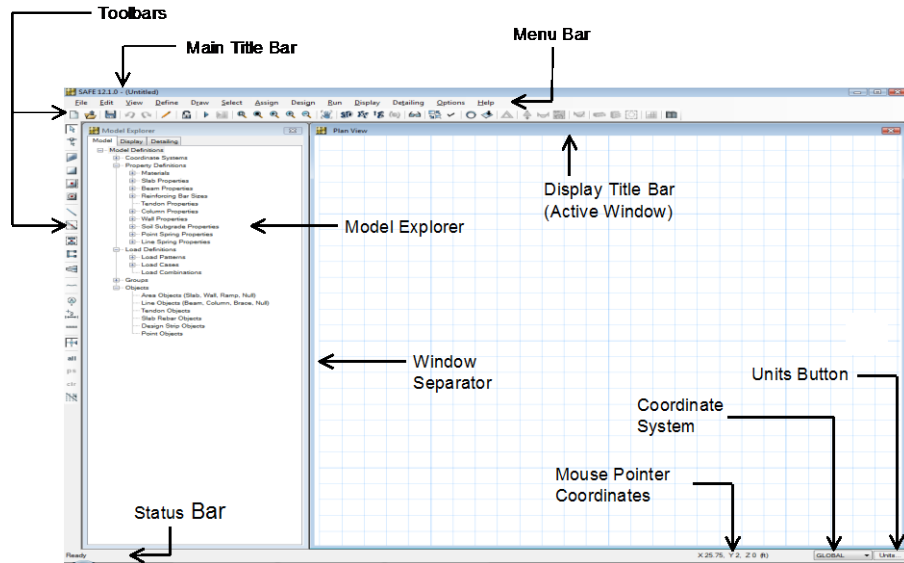

Figure 8-1 SAFE graphical user interface

Menu Bar – The menu bar contains the program menus from which various commands can be selected to perform specific actions.

Toolbars – Toolbars are made up of buttons. Buttons provide "single-click" access to commonly used commands. Hovering the mouse pointer over a toolbar button for a few seconds without clicking or holding down any mouse buttons will display a short description of the button's function.

Model Explorer – The model explorer allows easy access to model definition data, including property forms, load definitions, and object forms, as well as analysis and design results. These items are grouped in three tabs in the *Model Explorer*, namely Model, Display, and Detailing. Double clicking on an item in the tree with the left mouse button will bring

up the associated form or display, while a right click generates a context menu with additional options.

Display Window – A display window shows the geometry of the model and also may include displays of properties, loading, and analysis or design results. A maximum of four windows may be displayed simultaneously.

Display Title Bar – The display title bar is located at the top of the display window. The display title bar is highlighted when the associated display window is active. The text in the display title bar typically includes the type of the current view along with any other relevant data, such as load pattern, load combination, and so forth.

Status Bar – The status bar is located at the bottom of the main window. Text describing the current status of the program is displayed on the left side of the status bar.

Mouse Pointer Coordinates – The mouse pointer coordinates are displayed on the right-hand side of the status bar. A window does not need to be active for the mouse pointer coordinates to be displayed. It is only necessary that the mouse pointer be over the window.

Units Button – Clicking the **Units** button displays the Units form which lists the units associated with all input and display items; units are not typically consistent, i.e., stress may be displayed in ksi units, while applied loads may be done in psf units. Unit selections may be changed at any time.

8.2   File Menu

File operations are used to start a new model, open or import an existing model for display or modification, to import an architectural plan for tracing, to save or export the current model for use in SAFE or other applications, and to produce output. File operations are selected from the **File menu**.

New models can be started from scratch or from predefined templates built into SAFE.

In addition to opening files in the standard SAFE database format (*.fdb extension), SAFE also allows import of the following file formats: SAFE text files (*.$sf and *.f2k extensions), MS Access Database files (*.mdb extension), MS Excel Spreadsheet files (*.xls extension), or DXF/DWG files (geometry import only) and DXF/DWG Architectural Plan.

Models created in SAFE are saved in the database *.fdb and the text *.$sf formats by default – both of these files contain the complete model. They may also be exported to the following formats: SAFE text (*.f2k), MS Access Database (*.mdb), MS Excel Spreadsheet (*.xls), and Auto-CAD (*.DXF/*.DWG – geometry only).

8.3   Edit Menu

Editing is used to make changes to the model. Most editing operations work with one or more objects that are selected immediately before using the edit commands. Editing operations on the **Edit menu** include the following:

- Cut and Copy geometry of selected objects to the Windows clipboard. Geometry information on the clipboard can be pasted into other programs, such as spreadsheets.

- Paste object geometry from the Windows clipboard into the model. The geometry can be edited in a spreadsheet application before being copied and pasted into SAFE.

- Interactive editing of the model in a database format.

- Delete objects.

- Replicate objects in a linear or radial array, or mirror them.

- Move points, which also modifies connected objects.

- Merge points, which also modifies connected objects.

- Align points and their connected objects to specified ordinates or selected lines; this includes trimming or extending selected beam, strip or tendon objects.

- Divide selected beam or slab objects into smaller objects.

- Join selected line objects at a common joint.

- Merge selected area objects.

- Expand or shrink the size of area objects.

- Add tendon objects to selected design strips and beams.

- Adjust the horizontal layout and vertical profile of selected tendons.

- Add user-defined slab reinforcing.

- Generate design strips based on grid lines or support locations.

8.4 View Menu

View options, which affect the active display, may be set for each display window individually. The options listed below are available on the **View menu** and from the toolbar.

- **Set Default 3D View** – A 3-D view shows the entire model from a vantage point. The default view provides a perspective view of the entire model and can be rotated using the **Rotate 3D View** button.

- **Set Plan View** – A plan view is generated in a single plane parallel to the X-Y coordinate plane.

- **Set Elevation View** – An elevation view is generated in a single vertical plane

- **Set Display Options** – Various options are available to control how objects appear in a display window. These options primarily affect

views of the undeformed shape, including changing visibility, labeling of objects, showing an extruded view, as well as others.

- **Zoom Features** – **Zoom In** to see more detail, or **Zoom Out** to see more of the structure. A rubber-band zoom also can be used to draw a window to zoom into. **Restore Full View** zooms in or out, as necessary, to fit the view of the entire model in the active window.

- **Pan** – Panning allows dynamic movement of the structure around the display window by clicking and moving the mouse.

8.5   Define Menu

Define operations are used primarily to create named entities that are not part of the geometry of the model. These entities, accessed from the **Define menu**, include the following:

- Material properties

- Slab and beam section properties and design parameters

- Reinforcing bar sizes

- Tendon properties

- Column and wall properties

- Soil subgrade properties

- Point and line spring properties

- Coordinate/grid systems

- Group names

- Mass source

- Load patterns

- Load cases

- Load combinations

Definition of these entities does not require prior selection of any objects, and some of these entities also can be defined during the assignment operations from the **Assign menu**.

8.6   Draw Menu

Drawing is used to add new objects to the model or to modify one object at a time. Objects include beams, columns, slabs, walls, tendons, design strips, gridlines, dimension lines, and slab rebar. To draw, SAFE must be in **Draw Mode**, which is activated by clicking one of the draw buttons on the toolbar or using a **Draw menu** command.

In **Draw Mode**, the left mouse button is used to draw and edit objects, and the right mouse button terminates the current drawing action.

Depending on the type of object to be drawn, a floating properties form appears that can be used to specify various structural and support properties for the object. For example, as column objects are drawn, column properties can be assigned simultaneously, beam objects may be assigned beam structural properties, and similar options exist when drawing other object types.

8.7   Select Menu

SAFE uses a "noun-verb" concept; that is, a selection is made and then an operation is performed. Certain editing, assigning, printing, and display operations require a prior selection of objects. Selection is used to identify those objects to which the next operation will apply.

To make a selection, SAFE must first be in **Select Mode**, which is activated by clicking one of the select buttons on the toolbar. Alternatively, selecting any command from the **Select menu** puts SAFE into **Select Mode**.

Several types of selection are available in SAFE, including:

- Selecting individual objects.

- Drawing a window or polygon around objects.

- Drawing line segments that intersect objects.

- Selecting objects by material property.

- Selecting beam, column, slab, wall, or tendon objects by property.

- Selecting objects by soil subgrade or point and line spring properties.

- Selecting objects by groups.

- Selecting objects by labels.

- Selecting all objects.

- Reselecting previously selected objects.

In **Select Mode**, the left mouse button is used to select/deselect objects, and the right mouse button is used to review the properties of objects.

8.8   Assign Menu

Certain assignments may be made when drawing an object, such as beam structural properties when drawing a beam object. However, additional assignments or changes to assignments may be made to selected objects using the **Assign menu** commands. Some of the assignment operations available from the **Assign menu** include:

- Assigning slab properties.

- Assigning openings to area objects.

- Assigning vertical offsets to slabs.

- Assigning rib locations for ribbed and waffle type slabs.

- Assigning beam properties.

- Assigning tendon properties.

- Assigning column/brace properties.

- Assigning wall/ramp properties.

- Assigning soil subgrade properties to area objects.

- Assigning releases to slab, beam, and column objects.

- Assigning point, beam, tendon, and surface loads.

- Assigning objects to named groups.

- Assigning points and lines to be included in the analysis mesh.

8.9   Design Menu

The **Design menu** is used both before and after running the analysis and design. Before running the design, the design preferences and design combinations are accessed from the **Design menu**. The design preferences specify the design code to be used, cover to reinforcement, stress check limits, and other relevant parameters. The combinations to be considered for design also are specified from the **Design menu**.

Design overwrites can be applied to the structural objects to adjust the design input values and parameters on an object-by-object basis. Overwrites can be applied to slabs, beams, and punching checks. These overwrites can be applied before or after design.

8.10   Run Menu

Before running the analysis and design, the desired maximum mesh size should be set. This can be accomplished using the **Run menu > Automatic Slab Mesh Options** command. If performing a nonlinear cracking analysis, the reinforcement used for calculating the cracked moment of inertia should be specified using the **Run menu > Cracking Analysis Options** command.

The analysis and design can be run together using the **Run menu > Run Analysis & Design** command or by clicking the **Run Analysis & Design** button on the toolbar.

The program saves the model in a SAFE database file, then checks and analyzes the model. When the analysis is complete, a deformed shape will be displayed. The cursor can be moved over the slab to view the displacements for any location on the slab.

8.11   Display Menu

The **Display menu** commands are used to view the model and the results of the analysis. Graphical and tabular displays are available in SAFE. Display types may be chosen from the **Display menu** and most of them also can be accessed using toolbar buttons.

- Graphical Displays – Different types of graphical displays may be selected for each display window. Each window also may have its own view orientation and display options. Available displays of the model input include the undeformed geometry and applied loads. Analysis results can be displayed graphically for any case that has been run, and include deformed shapes, beam moment and shear diagrams, slab force and stress contour plots, integrated force and moment diagrams for design strips, and reaction forces, including bearing pressures. Design results can be displayed for slabs and beams in the form of required reinforcement. Additional options are available to refine the required reinforcement display if desired. Additionally, deformed shape plots may include mode shapes if a dynamic analysis is performed, and all shapes can be animated.

- Tabular Displays – Tabular information can be displayed for selected objects using the **Display menu > Display Tables** command. Choose the tables to be viewed and for which load patterns and load cases the data should be presented. If objects are selected, only data for the selected objects will be displayed. If no objects are selected, the tables include data for the entire model. Advanced options exist to control

the format of the tables and to create filters that will display only data that satisfy specified criteria.

- Tabular data also can be directly exported or printed from the tabular display, or by using commands available on the File menu.

8.12   Detailing Menu

After a complete structural model has been analyzed and designed, the reinforcement can be detailed. Before running the detailing, the **Detailing menu** commands should be used to define the detailing preferences, including items such as drawing size and layout, curtailment rules, desired bar sizes, rebar call (mark) notations, rebar shapes, min and max bar size and length, and so on.

After the detailing has been completed, additional commands on the **Detailing menu** can be used to edit the reinforcement, add or modify section cuts, and add or modify drawing sheets. Several other detailing commands are available from the right click context menus, when right clicking on drawing component views or the items on the detailing tree in the *Model Explorer* window.

8.13   Options Menu

The **Options menu** provides various commands that affect the overall operation of SAFE. Input, output, and display units can be reviewed and modified here. A set of global preferences define the merge tolerance, screen font size, zoom increment, and similar parameters. Colors of individual items, as well as general color schemes may be controlled. The number of windows to display, along with which layers of an architectural plan should be shown, can be set using this menu. SAFE also offers the user the choice between two graphics modes, Standard and DirectX; DirectX provides a rendered look to the model but may not be supported by all computers.

8.14   Help Menu

The program Help is available from this menu. Documentation and verification manuals in PDF format are accessed through the **Help menu** as well. A link to the CSI website, as well as information about the currently installed version of SAFE and its associated license file, also are available from the **Help menu**.

8.15   More Information

This chapter provides a general introduction to the SAFE graphical user interface and main menu structure. Additional information may be found in the program Help by pressing the **F1** function key on the keyboard.

Chapter 9
Key Topics

This chapter provides information on some of the key topics and features available in SAFE. It briefly describes the two modes of user operation for SAFE, the templates and defaults, the coordinate systems and grid lines, the slab overlap hierarchy, some of the useful keyboard shortcuts, and some time saving options.

9.1   Select or Draw Mode

The two distinct modes in SAFE are the *Select* mode and the *Draw* mode. The select mode allows selection of objects and is used for editing operations, making assignments to objects, and viewing or printing results. The draw mode allows drawing objects. By default, the program is in select mode.

The draw mode is automatically enabled when one of the drawing commands from the **Draw menu** or the corresponding toolbar buttons is clicked. The available drawing commands include the following:

▪ **Draw Column/Point Objects**

- **Draw Beam/Line Objects** ╲ or ◩

- **Draw Wall Objects** ⌐

- **Draw Design Strips** ◁

- **Draw Tendon Objects** ∿

- **Draw Slab/Area Objects** ◢ , ▢ , ▣ , or ◉

- **Draw Grids** Ⓐ

- **Draw Dimension Lines** ⁺₂↔

- **Draw Slab Rebar Objects** ▭

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

- **Click the Select Object** button on the toolbar ▷

- Press the **Esc** key on the keyboard.

- Select a command from the **Select menu**.

9.2   Templates and Defaults

SAFE provides a number of templates that allow for the rapid generation of models for a range of slab and foundation systems. These templates serve as a good starting point because they can be modified easily or appended to as needed. SAFE includes default parameters, some of which are building code specific. These defaults are accessed using "Preferences" throughout the various aspects of the program.

By using the built-in templates and defaults, a model can be created in a matter of minutes. Even if the template does not exactly match the structure under consideration, it can act as a good starting point for building

the desired model. The SAFE Help topics contain additional information on using the templates and modifying the program defaults.

9.3   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, X, Y, and Z, are mutually perpendicular and satisfy the right-hand rule. SAFE always considers the +Z direction as upward. By default, a positive load acts in the –Z (downward) gravity direction.

All slab, beam, and point objects lie on the XY (horizontal) plane. Tendon objects also are laid out on the XY plane, but also have a profile in the Z-direction. Vertical offsets may be assigned in SAFE to slabs and beams (called cardinal points).

Additional coordinate systems may be defined to aid in developing and viewing the model. For each coordinate system, you can define a three-dimensional grid system consisting of "construction" lines 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 origin. Any number of gridlines in the X or Y directions with arbitrary spacing can be defined, as well as arbitrary general gridlines.

When starting a new model from a template, grid spacing must be uniform. Thereafter, grid lines can be added, moved, and deleted. When a grid line is moved, it is possible to specify that points located on the grid line should or should not be moved with it.

Setting up one or more grid systems for a model has many benefits. If the grid is set up to match the architectural grid, it will allow for easier transfer of data between architectural drawings and the model. Drawing operations also tend to "snap" to gridline intersections (default) unless this feature is turned off. When the analysis is run and the internal finite

element mesh is created, the defined gridlines are used to help guide the meshing.

9.4 Overlapping Area Objects

Area objects are used not only to model slabs, but also drops and the stiffened area objects directly above columns and walls. Modeling drops and these stiffened regions using area objects ensures that the geometry of the structure is accurately represented, i.e., that the span lengths are correct. Thus, overlapping area objects often occur at support locations, where it is not uncommon to have a slab, a drop, and a stiffened member at the same plan position. Drop panels and stiff area objects also are defined using the **Define menu > Slab Properties** command, and when drawing the area objects to represent these members, SAFE knows to use the following hierarchy when the objects overlap.

An area object that is completely embedded within another area object takes precedence, e.g., SAFE will use the property of the stiff area object when it lies completely within a drop, and the property of the drop when it lies within the slab. On the occasion where a drop or stiff area object is not completely embedded within a single area object, but rather spans between two or more area objects, SAFE will assign a hierarchy based on the Analysis Property Data Type. An area object with a stiff type will replace an area object with a drop type, which in turn will replace an area object with a slab type.

Thus, for a model containing slab and stiff area objects, SAFE will use only the stiff area properties in the stiff area locations when generating the internal finite element mesh and will ignore the overlapping slab properties. Area objects that have an analysis property data type of *Stiff* have a modification factor of 100 applied to the bending stiffnesses, and are flagged for no design.

Occasionally multiple slab area objects (as opposed to drops and stiff area objects) may be drawn such that they overlap. Again, a slab area object completely embedded within another slab area object takes prece-

dence. In the case where one slab area object does not lie completely within another object, the last object drawn takes priority.

Important Note: Only the properties of the overlapping area objects are ignored; any surface loads applied to those redundant objects are *not* omitted and will be applied in an additive manner. Therefore, although the areas of overlap typically are small in comparison to the total area of the slab, it is generally recommended that load assignments be made to only one object at any given location. Typically the load should be assigned to the primary slab area objects, and surface loads should not be assigned to drops or column area objects.

It also should be noted that any area object drawn as an opening or assigned as an opening using the **Assign menu > Slab Data > Opening** command will take precedence over an area object at the same location with a property assignment. Area objects used as openings are by default unloaded. However, an option exists to load openings with the same surface loading as the area objects they replace, which can be done when drawing the opening or by using the **Assign menu > Slab Data > Opening** command.

9.5   Keyboard Shortcuts

SAFE includes several keyboard and mouse shortcuts for quickly and easily carrying out various operations. The following list summarizes some of the available keyboard and mouse shortcuts:

▪ Various key combinations for accessing menu commands, many of which are typical Windows keyboard shortcuts, such as **CTRL +S** to save, **CTRL +C** to copy, and many others. The shortcut key combinations are listed next to the menu items.

▪ Use the mouse roller wheel to zoom in and zoom out.

▪ Press down and hold the mouse roller wheel while in a Display Window, then drag the mouse to Pan the model.

- While holding the **Shift** keyboard key, press down and hold the mouse roller wheel while in a Display Window, then drag the mouse to rotate the model.

- Visual copying can be carried out by first selecting a set of objects, then while holding the **CTRL** keyboard key, click and hold the left mouse button and drag the mouse pointer to a new location to copy the selected objects.

- While drawing, pressing the **'N'** keyboard key will toggle through the activated snap options.

9.6   Time Saving Options

SAFE includes several options that help minimize the time spent creating models. A selection of these options is described in the following:

- **Model Explorer** – Allows quick access to model definition data, analysis
results, design results, and detailed drawings through an expandable hierarchical tree menu system.

- **Quick Drawing** – Allows drawing of point, line, and area objects using a single mouse click, including slabs, beams, drop panels, and columns.

- **Snap To** – Allows placement of structural objects with accuracy, using an extensive set of customizable snap settings.

- **Draw {Object Type} Form** – Allows assignment of attributes to the object while drawing the object.

- **Import .DXF/.DWG Architectural Plans** – Allows import of architectural plans that can then be used as a reference for defining SAFE models. The plans themselves are not part of the structural model, but rather act as templates that can be traced, using snap features, to generate a model.

- **Create Report** – Generates a summary report for the model, with a single button click, including model geometry, assignments, properties, analysis results, and design results.

9.7 More Information

This chapter provides a selection of the key topics within SAFE. Additional information on these topics and other aspects of the program may be found in the program Help, the other manuals, and in the "*Watch & Learn™ Movies*" series.