SAFE™

Integrated Analysis and Design of Slab Systems

Introductory User's Guide



Computers and Structures, Inc.
Berkeley, California, USA

Version 8.0.0
August 2004

Copyright

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

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

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

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2004.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
SAFE and CSiDETAILER are trademarks of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.
Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.
AutoCAD is a registered trademark of Autodesk, Inc.

DISCLAIMER

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

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



Contents

Introductory User's Guide

1 Program Description

This is SAFE 1-1
Time Saving Options 1-2
Templates and Defaults 1-3
Basic Process 1-3

2 The SAFE "Screen"

Objective 2-1
The Graphical User Interface 2-1
Units 2-3
File Operations 2-3
Viewing Options 2-4
Aerial View 2-5
Refreshing the Display Window 2-5
Draw 2-6

Select 2-6
Edit 2-7
Assign 2-8
Define 2-9
Analyze 2-9
Display 2-10
Design 2-11
Detailing 2-11
Preferences 2-12
More Information 2-12

3 Basic Modes, Drawing Tools, Mouse Pointers, Coordinate Systems and Grids

Objective 3-1
Select or Draw 3-1
Coordinate Systems and Grids 3-3

4 Begin a Model

Objective 4-1
Create a Basic Grid System 4-2
Begin a Model Using Templates 4-4
Save the Model 4-6

5 Add Structural Objects and Define Properties

Objective 5-1
Define Properties and Supports 5-1
Draw Point Objects 5-3
Draw Line Objects 5-3
Draw Area Objects 5-4

6 Select Structural Objects

Objective 6-1

Selection Methods 6-1
Selection Commands 6-3
Deselect Command 6-4
Get Previous Selection Command 6-4
Clear Selection 6-4

7 Load the Structural Model

Objective 7-1
Define Static Load Case Names 7-1
Self-Weight Multiplier 7-2
Long-Term Deflection Multiplier 7-3
Assign Structural Loads 7-3

8 Assign Properties to the Model

Objective 8-1
Assign/Change Properties 8-1
Overlapping Area Objects 8-3

9 Edit the Model Geometry

Objective 9-1
Editing Tools 9-1

10 Analyze the Structural Model

Objective 10-1
Set the Analysis Options 10-1
Run the Analysis 10-2
Locking and Unlocking the Model 10-3

11 Design the Structure

Objective 11-1
Design Process 11-1

12 CSiDETAILER

Objective 12-1

SAFE to CSiDETAILER 12-1

13 Obtain Basic Graphical Displays

Objective 13-1

Display Analysis Results 13-1

14 Generate Analysis and Design Results

Objective 14-1

Output Options 14-1



Chapter 1

Program Description

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

This is SAFE

SAFE is a powerful tool for the analysis and design of concrete slabs and basemats. For the first time, modeling, analysis, and design are all integrated in an easy-to-use object-based program that can handle simple or complex slabs and foundations. Seamless integration with CSiDETAILER provides for the automated generation of detail drawings. SAFE may be used as a stand-alone application, or may be used in conjunction with ETABS to complete detailed analysis and design of floor plates created in ETABS.

The basic approach for using SAFE is very straightforward. The user establishes grid lines, places structural objects relative to the grid lines using points, lines and areas, and assigns loads and structural properties to those structural objects. Analysis and design 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 to a file for use in other programs.

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

SAFE Menu Commands:

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

The subsequent chapters of this manual describe each of the menu commands in greater detail. Familiarity with the submenus will enable the user to create models for complex Flat Slabs with Openings, Slabs with Beams, Footings and Mats.

Additional Technical Notes are available to explain how the program performs concrete beam and slab design in accordance with applicable building codes. Information regarding the creation of detail drawings may be found in the CSiDETAILER manual.

Time Saving Options

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

- **Single Click Drawing** - Allows the user to draw line and area objects with a single mouse click, including drop panels.
- **Snap To** - Allows the user to place structural objects with accuracy.
- **Properties of Object Form** - Allows the user to assign structural properties to the object concurrent with drawing the object.

Templates and Defaults

1

SAFE provides a number of templates that allow for the rapid generation of models for a wide range of slab and foundation systems. Those templates serve as a good starting point because they can be modified easily or appended to as needed.

The program includes defaults parameters, some of which are building code specific. Those defaults are accessed using "Preferences." The possible options available for the default values for preferences are identified in the design Technical Notes.

By using the built-in templates and defaults, the user can create a model in a matter of minutes. Consult the SAFE Help topics for more information.

Basic Process

The following provides a broad overview of the basic modeling process:

1. Set the units.
2. Start a new model.
3. Set up grid lines.
4. Define slab and beam properties.
5. Draw structural objects.
6. Define supports.
7. Assign supports.
8. Define load cases.
9. Assign loads.
10. View and edit the model.
11. Analyze the model.
12. Display results for checking.
13. Design the model.
14. Generate output.
15. Save the model.



Chapter 2

The SAFE "Screen"

Objective

This chapter briefly describes the graphical user interface.

The Graphical User Interface

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

- **Main Window** - This window may be moved, resized, maximized, minimized, or closed using standard Windows operations. Refer to Windows help, available on the Start menu, for additional information on those items.
- **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.

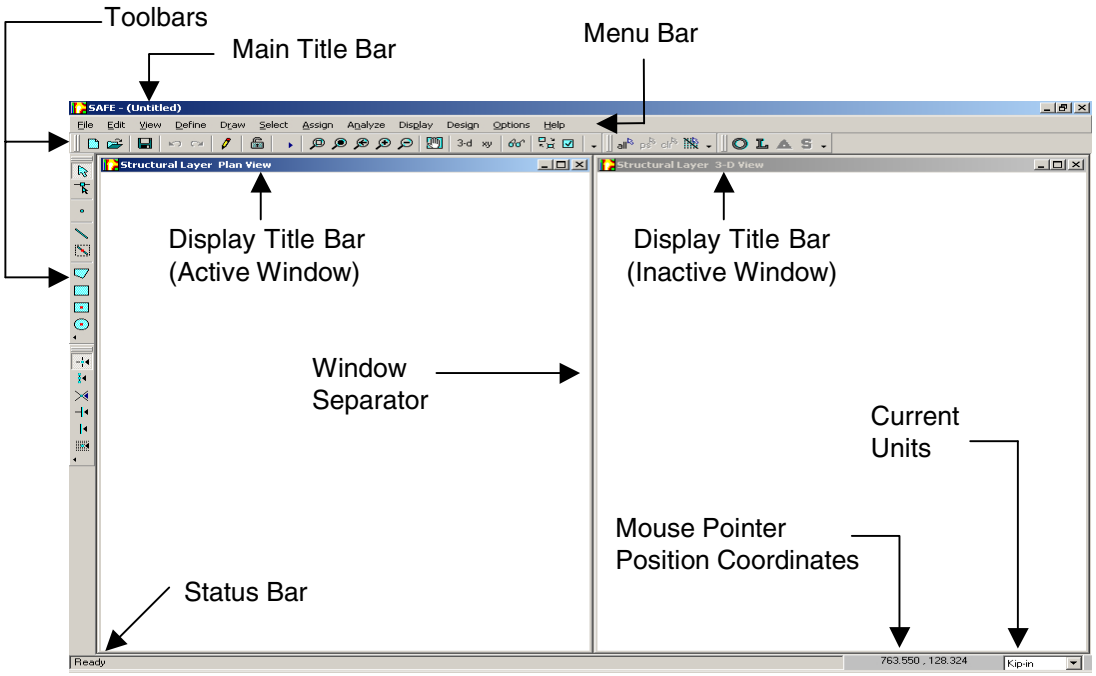


Figure 2-1 The SAFE graphical user interface

Move the main window by left clicking in the main title bar and holding down the mouse button to drag the window.

- **Menu Bar** - The menu bar contains all of the program's menus.
- **Toolbars and Buttons** - Toolbars are made up of buttons. Buttons provide "one-click" access to commonly used commands. Holding the mouse pointer over a toolbar button for a few seconds without clicking or holding down any mouse buttons will display a short description of the button's function in a small text box.
- **Display Windows.** A display window shows the geometry of the model and may also include displays of properties, loading and analysis, or design results. Up to four windows may display at any one time.
- **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 view in the associated display window.

- **Status Bar** - The status bar is located at the bottom of the main window. Text describing the current status of the program is displayed on the left side of the status bar.
- **Mouse Pointer Position Coordinates** - The mouse pointer position coordinates are displayed on the right-hand side of the status bar. A window does not need to be active for the mouse pointer position coordinates to be displayed. It is only necessary that the mouse pointer be over the window. The mouse pointer position coordinates are only displayed for two-dimensional plan views.

Units

SAFE works with two basic units: force and length. The program offers many different compatible sets of force and length units to choose from, such as “Kip-in” or “N-mm.” Angular geometry is measured in degrees.

Upon starting a new model, a set of units should be specified. Those become the “base units” for the model. Although data may be input or reported using any set of units, those values are always converted to and from the base units of the model.

The current units are displayed in a drop-down box located on the far right-hand side of the status bar. The units can be changed at any time during the model creation process.

File Operations

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

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

In addition to opening files in the standard SAFE database format (.FDB extension), the program also allows files to be imported in the following formats: V6/V7 text file (.F2K extension), V5 text file, or .DXF/.DWG files (geometry only) created by AutoCAD.

Models created by the program may be saved in the .FDB format, or exported in the following formats: SAFE text (.F2K), SAP2000 text (.S2K), AutoCAD (.DXF/.DWG – geometry only) and Access Database file.

CSiDETAILER, which can be started from within SAFE, is an application for generating detail drawings for slabs and beams analyzed and designed using SAFE.

Viewing Options

View options, which affect how the structure displays, may be set for each display window and the options may differ from window to window. Those options are available on the View menu and from the toolbar.

- **Set Layers of the Slab** – The definition of the slab for analysis and design purposes is accomplished using layers. The three layers in the SAFE model of a slab are Structural Layer, X-Design Strip Layer, and Y-Design Strip Layer. The Structural Layer is used to define slab geometry, load, and boundary conditions, which are related to analysis only. The Design Strip Layers are used to define the design strips. The Design Strip Layers hide all information related to analysis.
- **Set 2-D View** - A 2-D view consists of a single plane parallel to the X-Y coordinate plane.
- **Set 3-D View** - A 3-D view shows the whole model from a vantage point of your choice. The view direction is defined by an angle in the horizontal plane and an angle above the horizontal plane.
- **Set Object Options** – Various options are available to control how objects appear in a display window. The options primarily affect views of the undeformed shape. They include controls for visibility and labeling for the various types of objects, as well as the "shrunken-object

view." This option shrinks objects away from the joints allowing better review of the model's connectivity.

- **Zoom Features** – Zoom in to a view to see more detail, or zoom out to see more of the structure. Zooming in and out may be completed in predefined increments. You may also zoom in to a part of the structure by dragging a selection window around the portion of interest.
- **Pan** – Panning allows dynamic movement of the structure around the display window by clicking and moving the mouse.
- **Other Options** – Grid lines and the global axes can be turned on and off. Automatically generated mesh can be viewed. After these and other settings have been set, the view parameters can be saved as a Named View and subsequently recalled and applied to any display window.

Aerial View

The Aerial View displays a full view of the active window's drawing in a separate window. Use the Aerial View to help move around the active window of a larger model, and with the zoom features, to view smaller areas more easily. Also use the aerial view to track which part of the model is displayed in the active window. Each time the model is edited, the aerial view is updated. The Aerial View window can be made visible using the **Options menu > Show Aerial View Window** command.

Refreshing the Display Window

After performing certain options, the Display Window may need to be redrawn. The display window is not automatically refreshed after every change to save time. To refresh a display, click the **Refresh Window** button .

Draw

Drawing is used to add new objects to the model or to modify one object at a time. Objects include beams, slabs, slab design strips, and other point, line and area objects. To draw, the program must be in **Draw Mode**, which is activated by clicking one of the draw buttons on the toolbar or using a Draw menu command.

In **Draw Mode**, the left mouse button is used to draw and edit objects, and the right mouse button is used to query the properties of objects. Drawing may be performed in the XY plane only.

Depending on the type of object to be drawn, a "Properties of Object" form appears that can be used to specify various structural and support properties for the object. As point objects are drawn, column support properties can be assigned simultaneously. Line objects may be assigned beam structural properties or wall support properties when being drawn, and area objects can be assigned slab properties or defined as openings. After an object has been drawn, the object may be selected and loads may be assigned to it, or existing assignments can be modified.

Draw operations are also affected by the view layer of the slab. For structural objects and loading, usually the view is set to the Structural Layer. To define design slab strips, set the view to Strip Layers.

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

Select

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

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

To select, the program must be in **Select Mode**, which is activated by clicking one of the select buttons on the toolbar. Alternatively, selecting any action from the Select menu puts the program into Select Mode.

Many different types of selection are available, including:

- Selecting individual objects.
- Drawing a window around objects.
- Drawing a line that intersects objects.
- Selecting beam or slab objects having the same property type.
- Selecting beams or slabs by groups.
- Selecting all objects.
- Selecting previously selected objects again.

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

Draw Mode and Select Mode are mutually exclusive.

Edit

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

- Cutting and Copying the geometry of selected objects to the Windows clipboard. Geometry information put on the clipboard can be accessed by other programs, such as spreadsheets.
- Pasting object geometry from the Windows clipboard into the model. The geometry could be edited in a spreadsheet program before being copied and Pasted into SAFE.
- Deleting objects.

- Moving points, which also modifies connected objects.
- Replicating objects in a linear or radial array or mirroring them.
- Aligning points and their connected objects to be exactly vertical or horizontal.

Assign

Certain assignments may be made when drawing an object, such as assigning the beam structural property when drawing a line object. However, additional assignments, or changes to assignments, may be made to one or more objects that were selected immediately before using the Assign menu command. Assignment operations are selected from the Assign menu, including:

- Assigning slab properties to area objects.
- Assigning openings to area objects.
- Assigning rib/joist locations for ribbed/waffle type slabs.
- Assigning beam properties to line objects.
- Assigning elevation offsets to area and line objects.
- Assigning column support properties to point objects.
- Assigning wall support properties to line objects.
- Assigning soil support properties to area objects.
- Assigning slab discontinuities to line objects.
- Assigning point, line, and surface loads.
- Assigning objects to named groups.

Define

Define is used to create named entities that are not part of the geometry of the model. Those entities, accessed from the Define menu, include the following:

- Slab material and section properties and design parameters.
- Beam material and section properties and design parameters.
- Column support properties.
- Wall support properties.
- Soil support properties.
- Static load cases.
- Load combinations.
- Group names.

Definition of those entities does not require prior selection of an object.

Analyze

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

Before running an analysis, Set Options for the analysis using the Analyze menu, including:

- Type of analysis and any control parameters.
- Maximum mesh dimension.

After setting the options, especially the “Maximum Mesh Dimension,” it may be advantageous to view the generated finite element mesh. To view the mesh and verify that it is satisfactory, click the **Set Object Options** button on the toolbar and check the Show Mesh box on the resulting

form; click the button again and uncheck the box to turn off the display of the mesh, if desired.

To run the analysis, select Run Analysis from the Analyze menu, or click the **Run Analysis** button on the toolbar. The program saves the model in a SAFE database file, then checks and analyzes the model. During the checking and analysis phases, messages from the analysis engine appear in a monitor window. When the analysis is complete, the scroll bar on the monitor window can be used to scroll through and review the analysis messages.

Display

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

- **Graphical Displays** – Different types of graphical display may be selected for each display window. Each window may also have its own view orientation and display options. Available displays of the model input include the undeformed geometry and loads. Analysis results that can be graphically displayed include deformed shapes; beam force and moment diagrams; slab moment and shear contour plots; integrated force and moment diagrams for design slab strips; and reactive forces, including bearing pressures. Additionally, deformed shape plots may be animated.

- **Tabular Displays** – Tabular information can be displayed for selected objects by choosing Show Input Tables or Show Output Tables from the Display menu. Tabular information can be printed or saved to a file for selected objects by choosing the **File menu > Print Input Tables** or **File menu > Print Output Tables** command. If no objects are selected, the tables produced are for the entire model.

Design

After an analysis has been completed, concrete slabs and beams can be designed with respect to design code requirements. Design may be performed for the given load combinations by choosing the **Design menu > Start Design** command. Before designing, verify the selected design code using the **Options menu > Preferences** command and the Design tab.

Graphical displays of reinforcing steel and other design parameters are available. Tabular design information can also be displayed using commands from the Design menu. Tabular design information can be printed for selected objects using the **File menu > Print Design Tables** command.

Detailing

After analysis and design are complete, use the optional CSiDETAILER program to create a detail model and generate plans, sections and tables showing the placement and size of the reinforcing based on the user-selected code. The detailing model created by CSiDETAILER is an internal model, with input consisting of data from the analysis and design of the user's model in SAFE and detailing options and preferences set by the user.

Use the commands on the Detailing menu to set preferences for the appearance of the drawings and to start the CSiDETAILER program. Use the commands within the CSiDETAILER program to specify additional preference settings that include, among other items, code, smallest and largest rebar, minimum and maximum number of rebar, shortest and longest rebar, rebar cutoff (curtailment) rules, rebar shapes, and rebar marks.

Output from the detailing model consists of detailed area objects that represent slab, footing and mat members and detailed line objects that represent beam members. Detailing information that can be viewed in plans, sections, elevations, tables and schedules includes the number, the size, the shape and the location of reinforcing bars as calculated by

CSiDETAILER based on the preferences. Output from CSiDETAILER can be exported to .dwg or .dxf format for further refinement using AutoCAD.

Preferences

SAFE allows custom setting of some parameters using the **Options menu > Preferences** command, including the design code, the cross-sectional areas of nominal bars, tolerances, and fonts.

More Information

More information about SAFE features and commands is available from the SAFE Help topics contained within the program. Pressing the F1 key on the keyboard while a form is displayed in the program will access context-sensitive help for that form. Information about code-specific design is available in the *SAFE Design Manual* and information about detailing using the optional CSiDETAILER program is available in the CSiDETAILER *User's Manual*.



Chapter 3

Basic Modes, Drawing Tools, Mouse Pointers, Coordinate Systems and Grids

Objective

This chapter briefly describes the two modes of user operation for SAFE, the drawing tools, the mouse pointers, the coordinate systems and grid lines, and the system-generated mesh.

Select or Draw

The two distinct modes in this program are the ***select*** mode and the ***draw*** mode. The select mode allows you to select objects and is used for editing operations, making assignments to objects, and viewing or printing results. The draw mode allows you to draw objects. By default, the program is in select mode. Chapter 6 describes the various methods for selecting points, lines, and areas in your model.

The draw mode automatically enables when you select one of the following commands from the Draw menu or click on the corresponding button(s) on the toolbar:

- Draw Point Objects 
- Draw Line Objects  or 
- Draw Area Objects , , , 

The draw mode remains enabled until you do one of the following to return to the select mode:

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

The mouse pointer indicates which mode is enabled. The mouse pointer is defined by the mouse pointer properties identified in the Windows Control Panel for the Normal Select Pointer and the Alternate Select pointer.

- In select mode, the pointer is the Normal Select Pointer, and if the default settings are being used, the mouse pointer will look like this .
- In draw mode, the mouse pointer is the Alternate Select pointer, and if the default settings are being used, the mouse pointer will look like this .

Note that while in draw mode if the mouse pointer "hovers" over the toolbar buttons or the menus, the pointer temporarily changes to the selection pointer. If you do not click on one of the menus or toolbar buttons, the mouse pointer reverts to the draw mode pointer when you move back into the display window.

Typically, set the properties for the mouse by clicking on the Windows Start menu, then Control panel and clicking on Mouse to bring up your mouse properties form.

Other mouse properties used at various special times in the program include Help Select, Busy, Text Select, Vertical Resize, Horizontal Resize, and Precision Select. The appearance of each of those mouse pointers also will change depending on the mouse pointer properties specified.

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. All area, line and point objects lie on the XY (horizontal) plane, and although vertical offsets may be assigned in SAFE to slabs and beams for later use in CSiDETAILER, those offsets do not affect the analysis or design.

SAFE always considers the +Z direction as upward. By default, a positive load acts in the –Z (downward) direction.

In addition to the coordinate system, a two-dimensional grid system consisting of "construction" lines used for locating objects in the model is also defined. The grid system is also of Cartesian (rectangular) definition, and is positioned relative to the global system.

Drawing operations tend to "snap" to grid line intersections (default) unless this feature is turned off. Numerous other snaps also are available on the Draw menu, including snap to line ends and midpoints and snap to intersections. Those powerful tools should be used whenever possible to ensure accurate model construction. Not using the snaps may result in "gaps" between objects, causing errors in the model's connectivity.

Any number of grid lines in the X or Y directions with arbitrary spacing can be defined. When starting a new model, grid spacing must be uniform. Thereafter, grid lines can be added, moved, and deleted.



Chapter 4

Begin a Model

Objective

This chapter describes how to create a basic grid system, begin a model using templates, and save the model. The grid system is the basis for creating the model. Structural objects are placed relative to the grid system, and typically the model grids should be defined to match the grids shown on the design documents.

The built-in SAFE templates are very convenient tools for the immediate generation of models of slabs and basemats that have regular geometries. The templates require simple control information to define the basic parameters for the structural geometry and loading from which the complete slab model, including the slab design strips, is created. In many cases, templates can be used as starting points for the definition of more complex slab systems.

Create a Basic Grid System

Begin creating a grid system by clicking the **File menu > New Model** command or the **New Model** button . The form shown in Figure 4-1 will display.

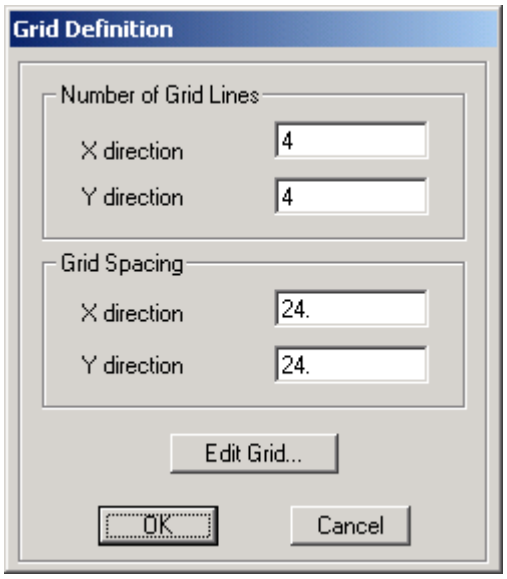


Figure 4-1 The Grid Definition form

The Grid Definition form is used to specify the number of horizontal grid lines and spacing. To obtain a uniform grid, simply specify the number of grid lines in the X and Y directions and the grid spacing for those lines. Note that the uniform spacing in the X and Y directions may be different. This option defines a grid system for the global coordinate system only. Click the **OK** button to accept the changes made to the form, or click **Cancel** to cancel the changes.

To specify non-uniform spacing and to label the grid lines, click the **Edit menu > Edit Grid** command. This will display a Define Grid form (see Figure 4-2), which has options for changing grid labels and spacing for each grid line in the X or Y directions. The form also has options related to how the grid lines display, including an option to make the grid lines invisible. Note that the global coordinate/grid system is a Cartesian (rectangular) coordinate system. A reference "datum" elevation may also be

specified, which is useful when analyzing multiple floors from the same building.

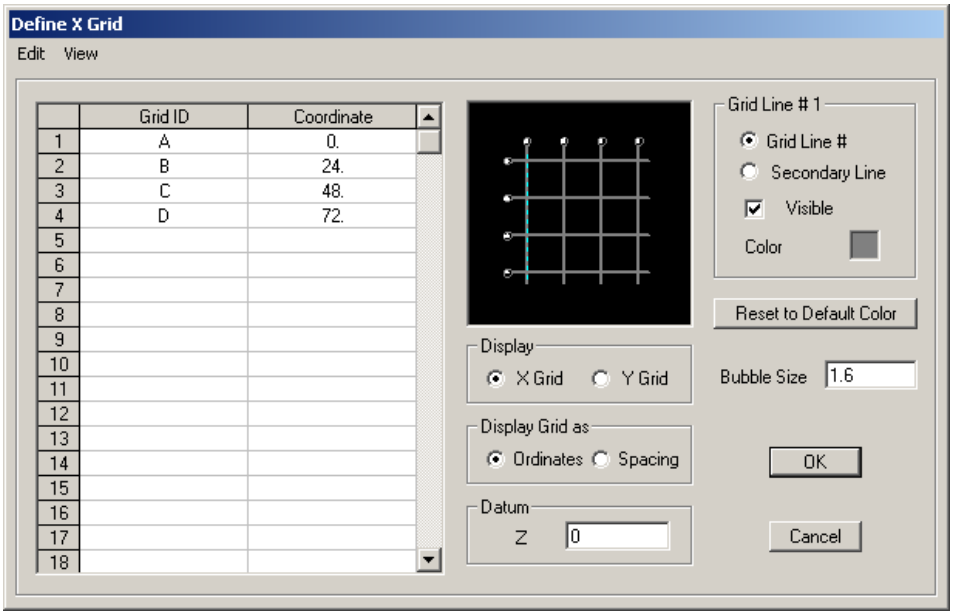


Figure 4-2
The Define Grid form

Definition of the grid system is important because of the following:

- Objects snap to grid lines when drawn in the model; thus, grid lines must be defined accurately so that the placement of objects can be completed efficiently.
- Objects mesh at their intersections with grid lines.
- The grid lines in the model can be defined with the same names as are used on the building plans, allowing for easier identification of specific locations in the model.

Begin a Model Using Templates

SAFE also has an option to begin a model using templates. Begin a model using templates by clicking the **File menu > New Model from Template** command. The Slab Templates form shown in Figure 4-3 will display.

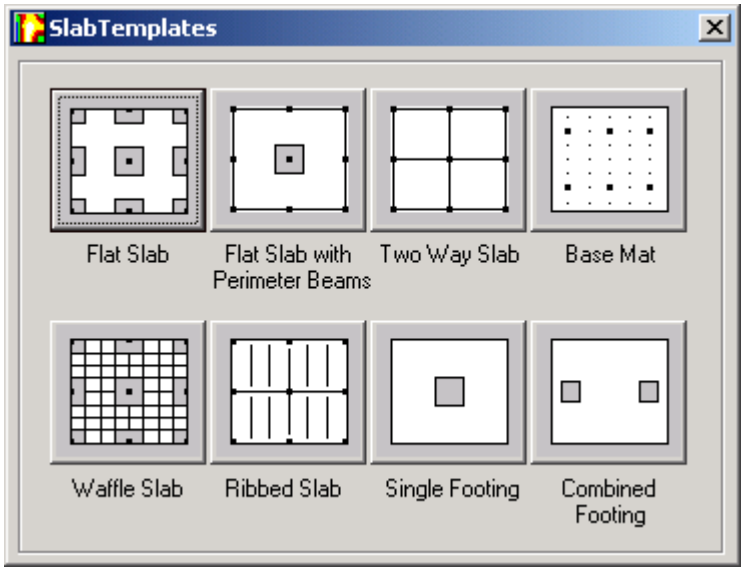


Figure 4-3 The Slab Templates form

Note that there are five slab templates, one base mat template, and two templates for footings. Template models provide a quick, easy way of starting a model. They automatically add structural objects with appropriate properties to a model. We highly recommend that you start your models using templates whenever possible.

Choose any of the templates by left clicking its associated button. When a template button is clicked, a form for that template will appear. Use the form to specify various data for the template, such as grid dimensions and spacing, slab or mat thicknesses, loads, drop panels or footings.

Important Note: When using the templates, beams, joists and ribs are modeled with depths equal to the dimension from the top of the slab to

the bottom of the beam, joist or rib. Beams are modeled as line objects, and joists and ribs are modeled with equivalent slab properties.

After specifying data for a template, the model will appear in the display window. The default display will show two views tiled vertically, a Structural Layer Plan View on the left and a Structural Layer 3-D View on the right, as shown in Figure 4-4. Use the **Options menu > Windows** command to change the number of windows displayed.

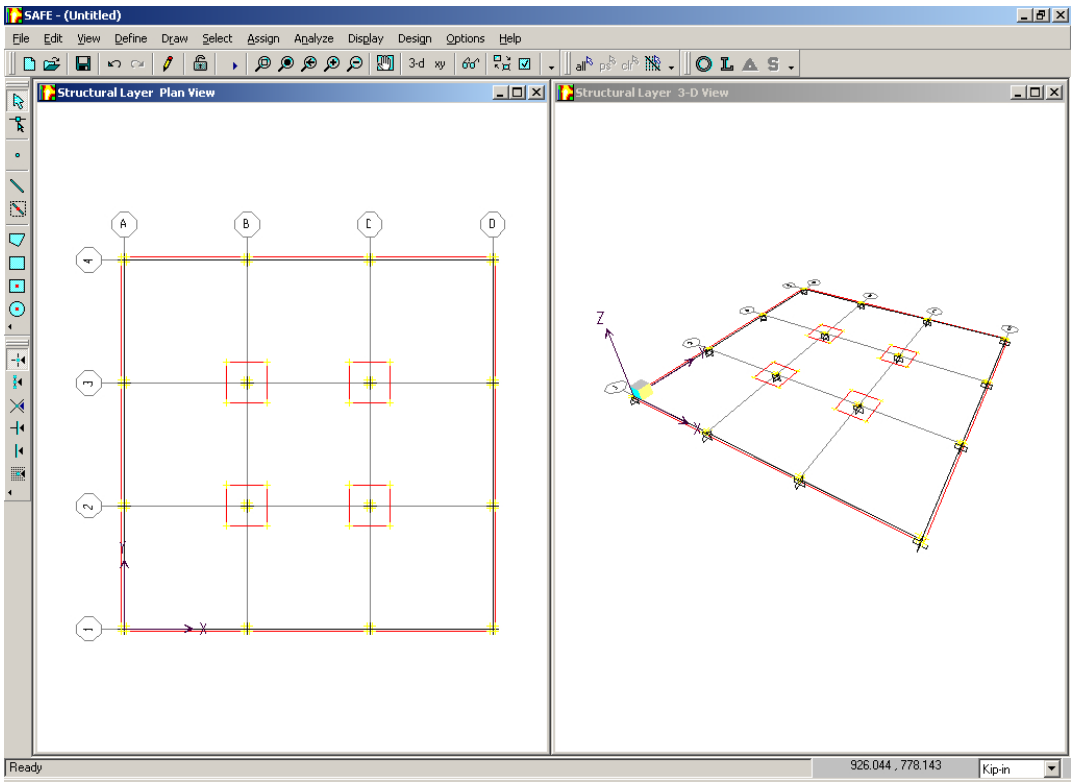


Figure 4-4
The SAFE main window

Note that the Structural Layer Plan View is active in Figure 4-4. When the window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window.

Save the Model

It is a good idea to save your model often. To save the model, click on the **File menu > Save** command or the **Save** button. The first time the model is saved, SAFE will prompt you to specify a file name in the Save Model File As form.



Chapter 5

Add Structural Objects and Define Properties

Objective

Point, line, and area objects are used in SAFE to represent columns supports, beams and slabs. Properties are defined to reflect their structural behavior in the model. This chapter touches briefly on defining properties and supports and describes how to add objects to the structural model. Point, line, and area objects can be added to a model that was started using only a grid system or to a model started using a template (see the previous chapter).

Define Properties and Supports

Some properties, such as slab and beam properties that contain both material and section definitions, are named entities that must be specified using the commands on the Define menu before assigning them to objects as described later in this chapter and in Chapter 8. If you assign a property to an object, for example a beam property to a line object, any

changes to the definition of the property will automatically apply to the object. A named property has no effect on the model unless it is assigned to an object.

Supports may be assigned to point, line and area objects, and similar to properties, they are named entities that must be specified before they are assigned. Column and wall supports may be defined as "Below," "Above" or "Above & Below" the slab. Based on the object type and the associated support properties, the program generates spring elements at each mesh location.

Table 5-1 identifies the subcommands on the Define menu, the type of object to which the definition can be applied, and the form used to complete the definition. Context sensitive help is available by pressing the F1 key when the form is displayed on-screen.

TABLE 5-1 Property Definitions

Property	Object Type	Name of Input Form
Slab Properties (Drop, Column, Openings)	Area	Slab Property Data
Beam Properties	Line	Property Data for Beam
Column Supports	Point	Column Support Property Data
Wall Supports	Line	Wall Support Property Data
Soil Supports	Area	Soil Support Property Data

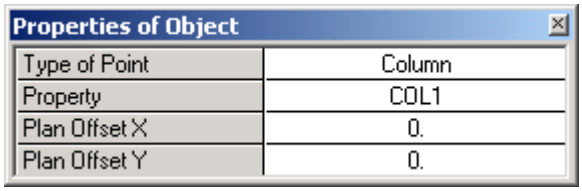
The input forms have options to add new definitions, modify or review existing definitions, and delete existing definitions. Each property definition should have a unique name that the user can specify, or use the program-suggested name. It is important that the definition names make it easy to assign them to the various objects correctly!

Depending on the type of property, the definitions can include user-specified analysis parameters such as thicknesses, heights, weights, Modulus of elasticity, Poisson's ratio, and design parameters such as top and bottom cover, material strengths and many other parameters. In addition to those items, the Slab Property Data form also has a parameter for specifying the type of slab (e.g., flat slab, waffle, drop), which controls not only the analysis formulation, but also the design and detailing.

Draw Point Objects

Point objects are most commonly used to create column supports. To draw a point object, make sure that the Structural Layer Plan View is active. Click the **Draw Point Object** button or use the **Draw menu > Draw Point Objects** command. The Properties of Object pop-up form for points shown in Figure 5-1 will appear.

Figure 5-1
Properties of Object form for Points



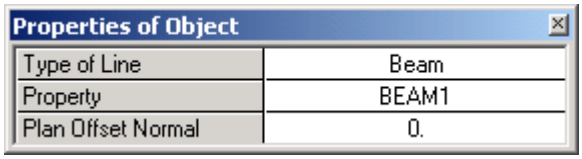
The Properties of Object form provides a Property drop-down list of the previously defined properties that can be assigned to the Type of Point. Review the definitions and drawing controls (Plan Offset X, Plan Offset Y) shown in the form before drawing the point. Change any entry in the form by clicking on it and making a new selection from the drop-down list or typing in new information into the edit box, as appropriate.

With the parameters in the Properties of Object form set, left click anywhere in the Structural Layer Plan View to locate the point object. A "plus" mark should appear at that point in the views. Continue to place other points/columns by left clicking.

Draw Line Objects

Line objects are most commonly used to create beams or wall supports. To draw a line object, activate the Structural Layer Plan View. Click the **Quick Draw Line Objects** button, or the **Draw menu > Quick Draw Line Objects** command. The Properties of Object pop-up form for line objects shown in Figure 5-2 will appear.

Figure 5-2
Properties of Object form for Lines



Similar to points, the Properties of Object form provides a Property drop-down list of the previously defined properties that can be assigned to the Type of Line. Change any entry in the form by clicking on it and making a new selection from the drop-down list or typing in new information into the edit box, as appropriate.

With the parameters in the Properties of Object form set, left click once in the Structural Layer Plan View on a *grid line* to place a line object. With the parameters set as shown in Figure 5-2, a beam will appear along the selected grid line in the display window. Continue to place other beams, or change the *Type of Line* to Wall to place wall supports.

Alternatively, click once in the center of the grid lines outlining a bay to draw beams or wall supports around the entire bay simultaneously.

Draw Area Objects

Area objects are most commonly used to create slabs (including footings and mats) or openings in slabs. To draw an area object, activate the Structural Layer Plan View. Click the **Draw Area Objects** button or select the **Draw menu > Draw Area Objects** command. The Properties of Object pop-up form for area objects shown in Figure 5-3 will appear.

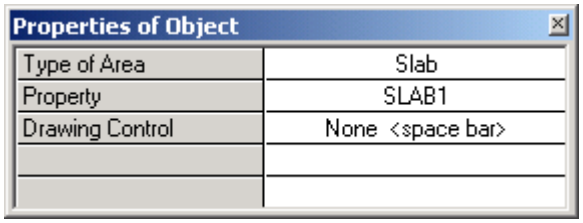


Figure 5-3 Properties of Object form for Areas

Similar to points and lines, this Properties of Object form provides a Property drop-down list of the previously defined properties that can be assigned to the Type of Area. Change any entry in the form by clicking on it and making a new selection from the drop-down list or typing in new information into the edit box, as appropriate.

With the parameters in the Properties of Object form set, ensure that the **Snap to Grid Intersections and Points** command is active to assist in accurately drawing the area object. This command is active when its as-

sociated button appears depressed. Alternatively, use the **Draw menu > Snap To > Points** command to ensure that this command is active. The **Snap to Grid Intersections and Points** command is active by default.

Left click once at a grid intersection (or any other position in the Structural Layer Plan View) to begin the area object at that location. Then, moving around the perimeter of the area object, click once at other grid intersections to draw the outline of the object. Press the Enter key on the keyboard to complete the object.

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

To better view the area addition, click the **Set Object Options** button . When the Set Objects form appears, check the Fill Elements check box as shown in Figure 5-4. Click the **OK** button to update the view.

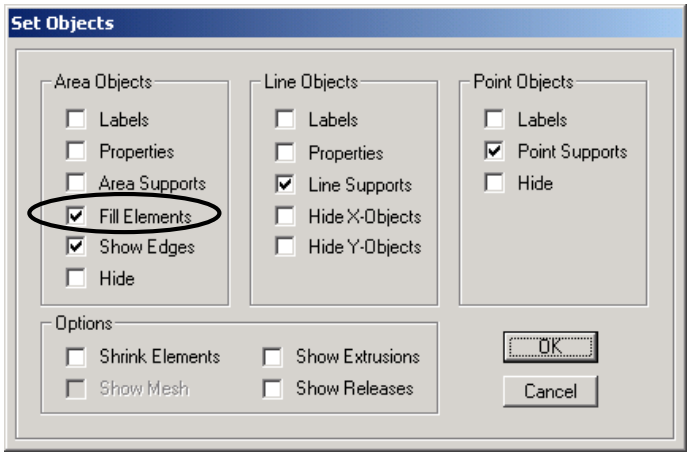


Figure 5-4
Set Objects form



Chapter 6

Select Structural Objects

Objective

This chapter describes the options available for selecting objects in SAFE.

Selection Methods

To enable the select mode, click the Pointer button on the toolbar . The program has three basic methods of selecting objects:

- **Left click:** Left click on an object to select it. If there are multiple objects, one on top of the other, hold down the **Ctrl** key on the keyboard and left click on the objects. A Selection List form similar to that shown in Figure 6-1will appear and identify the objects that exist at that location. Select the desired object by moving the mouse pointer over it in the form and left clicking on it.

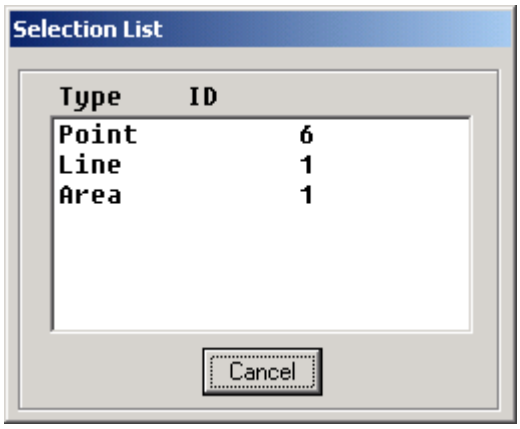


Figure 6-1
Selection List form

- **Window or "Windowing":** To draw a window around an object(s) to select it, position the mouse pointer beyond the limits of the object; for example, above and to the left of the object(s). Then depress and hold down the left mouse button. While keeping the left button depressed, drag the mouse to a position below and to the right of the object(s). Release the left mouse button to complete the selection. Note the following about window selection.

  As the mouse is dragged, a "rubber band window" will appear. The rubber band window is a dashed rectangle that changes shape as the mouse is moved. Any visible object that is completely inside the rubber band window when the left mouse button is released is selected.

  The window can be started at any point as long as the starting point is beyond the limits of the object(s) to be selected. For example, the starting point can be above and to the right, below and to the left or below and to the right of the object(s). In all cases, drag the mouse diagonally across the object(s) to be selected.

  An *entire* object must lie within the rubber band window for the object to be selected.

- **Intersecting Line:** To select one or more objects by drawing a line through it or them, click the **Select menu > Select > Intersecting Line** command or the **Select using Intersecting Line** button . Then position the mouse pointer to one side of the object(s) to be selected. De-

press and hold down the left button on the mouse. While keeping the left button depressed, drag the mouse across the object(s). Release the left mouse button to complete the selection. Note the following about the intersecting line selection method:

As the mouse is dragged, a "rubber band line" appears. The rubber band line is a dashed line that changes length and orientation as the mouse is moved. The rubber band line extends from the point where the left mouse button is first depressed to the current mouse pointer position. Any visible object that is intersected (crossed) by the rubber band line when the left mouse button is released is selected.

After using this method to make a selection, the program defaults to the window selection mode. Thus, the menu command or button  must be used each time to access the Select Using Intersecting Line method.

Selection Commands

Table 6-1 identifies the submenu commands and related actions accessed using the **Select menu** command.

Table 6-1 Submenus of the Select Menu Command

Command	Action
Select menu > Select by groups	Select the name of any collection of objects that has been defined as a group from the Select Group box and that group will be selected.
Select menu > Select by beam properties	Select a beam property name from the Select Beam Properties form and all line objects that have been assigned that beam property will be selected.
Select menu > Select by slab properties	Select a slab property name from the Select Slab Properties form and all area objects that have been assigned that slab property will be selected.

Table 6-1 Submenus of the Select Menu Command

Command	Action
Select menu > Select all	Selects all objects in the model, both visible and invisible objects. Be careful using this command. You can also use the **Select All** button  to execute this command.

Deselect Command

Objects can be deselected one at a time by left clicking on the selected objects. Alternatively, use the **Select menu > Deselect** command and its submenus for quicker and more specific deselection actions. This command accesses submenu items similar to those described in the previous section, except that executing the **Select menu > Deselect** command and an associated submenu item deselects rather than selects an object(s). As an example of the advantage of this option, assume that all objects in a model need to be selected except for those with a particular beam property. Do this quickly and easily by first using the **Select menu > Select > All** command and then using the **Select menu > Deselect > Beam Properties** command.

Get Previous Selection Command

The **Select menu > Get Previous Selection** command reselects the previously selected object(s). For example, assume that some area objects were selected by clicking on them and slab properties were assigned to them. Use the **Get Previous Selection** command or the **Get Previous Selection** button  to select the area objects again and assign something else to them, such as surface loads.

Clear Selection

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

Load the Structural Model

Objective



Note:

An unlimited number of load cases can be defined in SAFE.

This chapter describes defining and assigning structural loads for the model. Loads, including dead, live, earthquake, snow and the like, are defined as static load cases that are assigned to various structural objects in the model.

Note that the program automatically assembles the static load cases into design load combinations in accordance with the specified building code. Design codes can be selected using the **Options menu > Preferences** command.

Define Static Load Case Names

Click the **Define menu > Static Load Cases** command to access the Static Load Case Names form. Use the form to specify the following information:

- The name of the load case. The program does not allow use of duplicate names.
- A load type, which can be selected from the Type drop-down list.
- A self-weight multiplier (see the explanation about the s**elf-weight multiplier** that follows).
- A long-term deflection multiplier (see the explanation about the **long-term deflection multiplier** that follows).

Click the **Add New Load** button to add a new load case name. Select a previously defined load case and click the **Modify Load** button to modify that load case, or click the **Delete Load** button to delete it. Consult the SAFE Help for more specific information about defining loads.

Self-Weight Multiplier

The self-weight of the structure is determined by multiplying the unit weight (weight per unit volume) of each object that has structural properties times the volume of the object. The unit weight is specified in the slab or beam properties, which are accessed using the **Define menu > Slab Properties** or **Define menu > Beam Properties** commands, respectively.

Specify that a portion of the self-weight be applied to any static load case. The self-weight multiplier controls what portion of the self-weight is included in a load case. A self-weight multiplier of 1 means to include the full self-weight of the structure in the load case. A self-weight multiplier of 0.5 means to include half of the self-weight of the structure in the load case.

Normally you should specify a self-weight multiplier of 1 in one static load case only, usually the dead load case. All other static load cases then have self-weight multipliers of zero. Note that if you include a self-weight multiplier of 1 in two different load cases, and then combine those two load cases in a load combination, the results for the load combination are based on an analysis where double the self-weight of the structure has been applied as a load.

Long-Term Deflection Multiplier

SAFE allows for an elastic (linear) analysis or a cracked deflection (non-linear) analysis. In a cracked deflection analysis, SAFE takes into account the reduction of the gross section properties resulting from the cracking of the concrete, which results in an increase in the deflections.

When performing a cracked deflection analysis, the user has the option to assign a long-term deflection multiplier to each load case to account for behavior such as shrinkage and creep. **Normally, a long-term deflection multiplier with a value greater than 1 should be applied only to permanent load cases**, **such as the dead load case**. All other temporary load cases (for example, live, wind, earthquake or snow) should use a factor of 1. The long-term deflection multiplier works only with the cracked deflection analysis.

Assign Structural Loads

Static loads can be assigned to point, line, and area objects. The object(s) must first be selected before a load can be assigned. Chapter 6 of this guide describes the options for selecting structural objects.

After the object has been selected, click the **Assign menu** command to access the applicable submenu and assignment options. Note that the type of object selected determines which assignment can be made. For example, if an area object is selected, the **Surface Loads** command will be available, while a point selection will allow the user to specify either **Point Loads** or **Point Deflections**. If only point objects (e.g., column supports) and line objects (e.g., beams) are selected before clicking the **Assign menu** command, the **Surface Loads** command will not be available.

After clicking the **Assign menu** command and the submenu applicable to the type of object, a form will appear. Table 7-1 identifies the forms generated when the various commands are used. Consult the SAFE Help for more specific information about assigning loads.

TABLE 7-1 Input Forms for Load Commands on the Assign Menu

Command	Name of Input Form
Assign menu > Point Loads	Point Loads
Assign menu > Point Displacements	Point Displacement Loads
Assign menu > Line Loads	Line Loads
Assign menu > Surface Loads	Surface Loads

Although the forms vary depending on the command used, each form has a drop-down list that allows the user to select the load case to be assigned. The forms also include other object/assignment-specific input fields that enable the user to refine the load assignment. Context-sensitive help is available for each form by pressing the F1 key on the keyboard when the form is displayed on screen.

It is important to remember that SAFE does not allow any deformation in the plane of the slab, so loads that would cause only displacements in the slab plane are not available within the loading forms.

Assign Properties to the Model

Objective

This chapter describes how to assign or change the properties of structural objects in the model. Note that properties and supports can be assigned when the object is being drawn as described in Chapter 5.

Assign/Change Properties

In creating the model, the user draws point, line, and area objects. To enable analysis and design, those objects must be assigned properties, such as slab properties, beam properties, column supports, wall supports, and loads, among others. Note that the definition of properties and assignment of loads were explained in Chapters 5 and 7, respectively.

The types of assignments that can be made to an object depend on the type of object. Table 8-1 identifies the types of assignments that can be made using the Assign menu commands, depending on the type of object.

TABLE 8-1 Possible Assignments to Objects by Object Type

Object	Assignment Option	Name of Input Form*
Points	Point Restraints	Point Restraints
	Column Supports	Support Properties
Lines	Beam Properties	Beam Properties
	Beam Offset	Beam Offset
	Wall Supports	Support Properties
	Releases	Slab Releases
Shell/Areas	Slab Properties	Slab Properties
	Slab Offset	Slab Offset
	Slab Rib Locations	Slab Rib Location
	Opening	**
	Soil Supports	Support Properties

* With a form displayed on-screen, press the F1key to access context-sensitive help.

**No form; command converts area to an opening

Releases, point restraints, offsets and rib/joist locations are assigned directly to objects. Those properties can only be changed by making another assignment of that same property to the object; they are not named entities and they do not exist independent of the objects.

View the assignment made to point, line, and area objects by *right* clicking on the object. The appropriate Point Information, Line Information, or Area Information form will display.

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

As shown in Table 8-1, the availability of commands depends on the type of object selected. The input forms include object/assignment-specific input fields that enable the user to refine the assignment. Modifications to the assignments can be made by accessing the input forms using the appropriate Assign menu command. Context-sensitive help ex-

plaining the various forms is available by pressing the F1 key on the keyboard while a form is displayed on-screen.

Overlapping Area Objects

Overlapping area objects often occur at support locations, where it is not uncommon to draw a slab, a drop, and a column member at the same plan position. When multiple objects overlap, the program determines which property to use based on the following hierarchy:

> An area object designated as a column type will replace an area object designated as a drop, which in turn will replace an area object designated as a slab.

Thus, for a model containing both slab and column type area objects, the program will use only the column type properties in the column locations when generating the internal finite element mesh, and will ignore the overlapping slab properties. Type identifiers for area objects are specified on the Slab Property Data form.

Occasionally an area object may be drawn within the boundary of a larger area object, and both objects have the same type assignment (e.g., they are both slab types). In that case, SAFE will use the properties associated with the smaller object that lies completely within the larger object. In the case where the objects overlap and one is not completely embedded within the other (and assuming they are of similar type), the program will use the properties from the object that was last drawn in the overlapping area.

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 would be assigned to the primary slab area objects, and surface loads would not be assigned to drops or column area objects.

It should also be noted that any area object assigned as an opening using the **Assign menu > Opening** command will take precedence over an

area object with a slab property assignment in the same location. In addition, area objects assigned as openings lose all surface loads that may have been applied to the objects (i.e., they are unloaded openings).

Finally, when it comes to design, area objects designated as column types should not be designed as it is generally not appropriate to use slab design rules for a column. An option is available on the forms to specify No Design.

Note: As a general rule, area objects that are designated as column types should have a thickness approximately equal to five times that of the surrounding slab, which results in an element that will be at least an order of magnitude two times stiffer in bending than the slab.

Edit the Model Geometry

Objective

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

Editing Tools

During the course of creating the model, the user may need to edit the model. Table 9-1 identifies the various edit command available in SAFE. Some are familiar Windows commands.

In most cases, the user should first select the point, line, or area object and then click the appropriate menu or button. In some cases, the action will be immediate (for example, the **Undo** or **Redo** commands). In other cases, a form will appear that allows the user to specify how the object is to be edited (for example, the **Edit menu > Replicate** command accesses the Replicate form, which allows the user to replicate point, line or area objects in one of three ways). In other cases, the command is a toggle

that, when enabled, will affect subsequent actions. Note that the type of commands and options available depends on the type of object being edited.

TABLE 9-1 Edit Commands in SAFE

Command	Action	Immediate/ Form Used/ Toggle
Edit menu > Undo and Redo	The **Undo** command or button deletes the last performed action. The **Redo** command or button restores the last step that was undone.	Immediate
Edit menu > Cut, Copy and Paste	Generally similar to the standard cut, copy and paste Windows commands.	Immediate
Edit menu > Delete	The **Delete** command deletes the selected object(s) and all of its assignments (loads, properties, supports and the like).	Immediate
Edit menu > Edit Grid	Edits the coordinate/grid system in either the X or Y direction.	Define (X or Y) Grid Form
Edit menu > Add Grid at Selected Points	Inserts new grid lines parallel to either the X or Y direction at the selected points.	Add Grid Lines at Selected Points Form
Edit menu > Move	Moves point objects in the model. When a point object is moved, all line and area objects attached to the point are reoriented or resized to account for the movement. Move works only with selected points; thus, to move a line or area object using this command, you must select the points that define the line or area object, not the line or area object itself.	Move Selected Points Form
Edit menu > Replicate	Copies and pastes duplicates of the selected line or area objects to specified locations using linear replication, radial replication, or replication about a mirror plane. Note that this command differs from the **Copy** command because it also copies any assignments made to the line or area object.	Replicate Form

TABLE 9-1 Edit Commands in SAFE

Command	Action	Immediate/ Form Used/ Toggle
Edit menu > Align Vertical/Horizontal	Aligns selected points and the line and area objects defined by selected points. A maximum distance that the points can move is specified and if the selected points are within that tolerance distance, they will align on a grid line or with each other if a grid line is not within the tolerance distance. When a point object is realigned, all line and area objects attached to the point are reoriented or resized to account for the movement. NOTE: This is a sophisticated and complex command, and it is strongly suggested that the user verify its behavior before using it on a large model.	Align Selected Lines/Edges/ Points Form
Edit menu > Convert Rect to Area	Converts selected rectangular area objects into polynomial area objects.	Immediate

With a form displayed on-screen, press the F1 key on the keyboard to access context-sensitive help about that form.

Analyze the Structural Model

Objective

This chapter describes how to analyze the model.

Set the Analysis Options

Before running the analysis, set the analysis options using the **Analyze menu > Set Options** command. The Analysis Options form shown in Figure 10-1 will appear.

Use the Analysis Options form to specify the type of analysis to be performed, as well as to define other parameters specific to the type of analysis selected. During analysis, SAFE automatically meshes the model, creating a set of lines parallel to the coordinate axes that define the finite element model. Generation of the mesh is influenced by the locations of point and slab objects, openings, beams and their orientations, loads, soil support, and gridlines. The user can influence the size of the

mesh by setting the maximum mesh dimension using the **Analyze menu > Set Options** command.

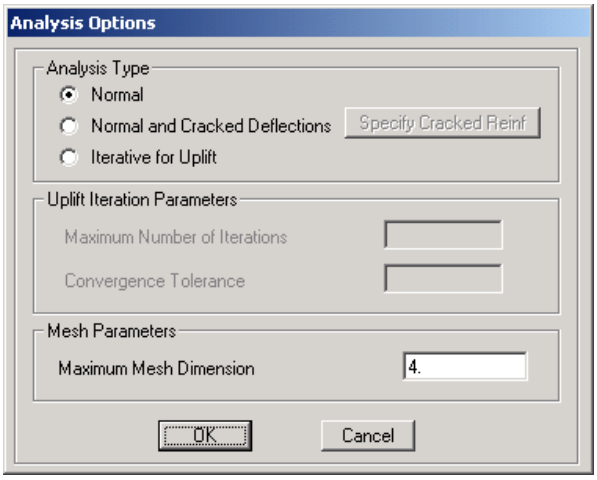


Figure 10-1
Analysis Options form

To view the element mesh generated by the program, click the **Set Object Options** button. When the Set Objects form appears, check the Show Mesh check box.

Run the Analysis

Run the analysis by clicking the **Analyze menu > Run Analysis** command or the **Run Analysis** button.

The program will display an "Analyzing, Please Wait" window. Data will scroll in this window as the program runs the analysis. After the analysis has been completed, it is possible to use the window to scroll through the data before closing the window.

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

Consult the SAFE Help topics for more information about analyzing your model.

Locking and Unlocking the Model

After an analysis has been performed, SAFE automatically locks the model to prevent any changes that would invalidate the analysis results and subsequent design results. The user also can lock the model at any time to prevent changes, or unlock it to permit changes using the **Lock/Unlock** button . SAFE will display a warning message that analysis results will be deleted if a model is unlocked following an analysis.

Design the Structure

Objective

This chapter describes design using the SAFE design option.

Design Process

SAFE designs both concrete slabs and beams. Slabs are those area objects that have been assigned slab properties, and beams are line objects with beam property assignments. Specific material properties and design data are input using the **Define menu > Slab Properties** or **Beam Properties** commands, and design code preferences are selected using the **Options menu > Preferences** command and the Design tab.

To perform the design, first run the analysis (described in the previous chapter), then click the **Design menu > Start Design** command. The type of design depends on the type of members used in the model (e.g., a beam design will be performed only if line objects with beam property assignments are included in the model).

After the design is complete, the plan view display will update to show the slab reinforcing required in the X-Strips (design strips in the X direction). Strips can be viewed using the **View menu > Set X-Strip Layer** or **Set Y-Strip Layer** commands.

The other commands available on the Design menu are somewhat sequential. That is, some commands must be used before other commands become available. This is intended to help the user step through the design process. Table 11-1 summarizes the commands used in the design process. With a form displayed on-screen, press the F1 key on the keyboard to access context-sensitive help about that form.

TABLE 11-1 Design Commands

Command	Action	Form
Design menu > Start Design	Initiates the design process. An analysis must precede use of this command.	Immediate, no form used
Design menu > Select Design Combo	Allows review of the default design load combinations determined by the program, or designation of user-specified design load combinations. Facilitates review or modification of load combinations during design.	Design Load Combinations Selection Form
Design menu > Display Slab Design Info	Allows the required slab reinforcing to be displayed directly on the model as an area or number of bars in the X or Y direction strips.	Slab Reinforcing Form
Design menu > Display Beam Design Info	Allows for the display on the model of the required beam reinforcing (flexural or shear).	Beam Reinforcing Form
Design menu > Display Punching Shear Ratios	Allows for the display on the model of calculated punching shear ratios.	No form; results are displayed onscreen.
Design menu > Show Design Tables	Allows the user to select the type of information to include in the design tables to be viewed on-screen, including design output and design forces.	Design Tables Form
Design menu > Show Total Quantities	Displays the slab area, concrete volume, and designed steel weight calculated by the program.	SAFE – Approximate Quantities Form

12

CSiDETAILER

Objective

This chapter describes how to use the CSiDETAILER program in conjunction with SAFE to generate reinforcing detail drawings.

SAFE to CSiDETAILER

The CSiDETAILER program will detail both slabs and beams after an analysis and design has been completed in SAFE. The program uses the required reinforcing obtained from the SAFE design along with code mandated minimums (as determined from the Detailing Options in CSiDETAILER) to produce drawings and schedules of reinforcing layouts. Within CSiDETAILER, the user has control over the drawing setup, including the sheet size and scale, the units and how each view should be displayed on the drawing.

To produce the reinforcing details (assuming that you have a licensed copy of CSiDETAILER in addition to a copy of SAFE), first run the

analysis followed by the design (described in Chapters 10 and 11, respectively), then set the preferences using the appropriate commands on the Detailing menu in SAFE. After the preferences have been set, click the **Detailing menu > Start Detailer** command to launch CSiDETAILER. The CSiDETAILER program will automatically generate all possible drawings. Consult the CSiDETAILER manuals and on-line Help for more information about using CSiDETAILER.

13

Obtain Basic Graphical Displays

Objective

This chapter describes how to display analysis results graphically.

Display Analysis Results

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

TABLE 13-1 Display Menu Options

Command	Action	Form
Display menu > Show Undeformed Shape	The **Show Undeformed Shape** command or button  plots the undeformed shape onscreen.	No form; results are displayed.

TABLE 13-1 Display Menu Options

Command	Action	Form
Display menu > Show Loads	The **Show Loads** command or button displays loads assigned to point, line and area objects.	Show Loads Form
Display menu > Show Input Tables	Provides the user with the opportunity to complete an on-screen review of the input parameters used in building the model.	Input Tables Form
Display menu > Show Deformed Shape	The **Show Deformed Shape** command or button plots a deformed shape on-screen based on user-specified loads. This plot can be animated for 3-D views.	Deformed Shape Form
Display menu > Show Slab Forces	The **Show Slab Forces** command or button displays slab forces on-screen based on user-specified loads.	Slab Forces Form
Display menu > Show Beam Forces	The **Show Beam Forces** command or button displays beam forces on-screen based on user-specified loads.	Beam Forces Form
Display menu > Show Strip Forces	Displays moments or shears in the X direction or Y direction strips based on user-specified loads.	Strip Forces Form
Display menu > Show Reaction Forces	Displays reaction forces (or soil pressures) on-screen based on user-specified loads.	Joint Forces Form
Display menu > Show Output Tables	Allows the user to select and show on-screen various tables of analysis results. Some of the output data is located via I and J grids, which may be displayed by checking the *Show Mesh* box on the **View menu > Set Object Options** command.	Output Tables Form

13

With a form displayed on-screen, press the F1 key on the keyboard to access context-sensitive help about that form.

14

Generate Analysis and Design Results

Objective

This chapter identifies the commands for generating analysis and design results.

Output Options

Analysis and design results can be printed to a printer or a file using the File menu commands. Table 14-1 identifies the print commands.

TABLE 14-1 File Menu Print Options

Command	Action	Form
File menu > Print Setup	Allows the user to specify page breaks and titles as well as the paper size and orientation of the page.	Print Setup Form

TABLE 14-1 File Menu Print Options

Command	Action	Form
File menu > Print Preview for Graphics	Provides a snapshot of how the file will print in graphical format.	No forms; results are displayed
File menu > Print Graphics	Prints whatever graphics are displayed in the active window to the printer that is currently specified as active.	Immediate, no form used
File menu > Print Input Tables	Prints tables of analysis input data to a printer or to a text file.	Input Tables Form
File menu > Print Output Tables	Prints tables of analysis output data to a printer or to a text file.	Output Tables Form
File menu > Print Design Tables	Prints tables of design output data to a printer or to a text file.	Design Tables Form

With a form displayed on-screen, press the F1 key on the keyboard to access context-sensitive help about that form.