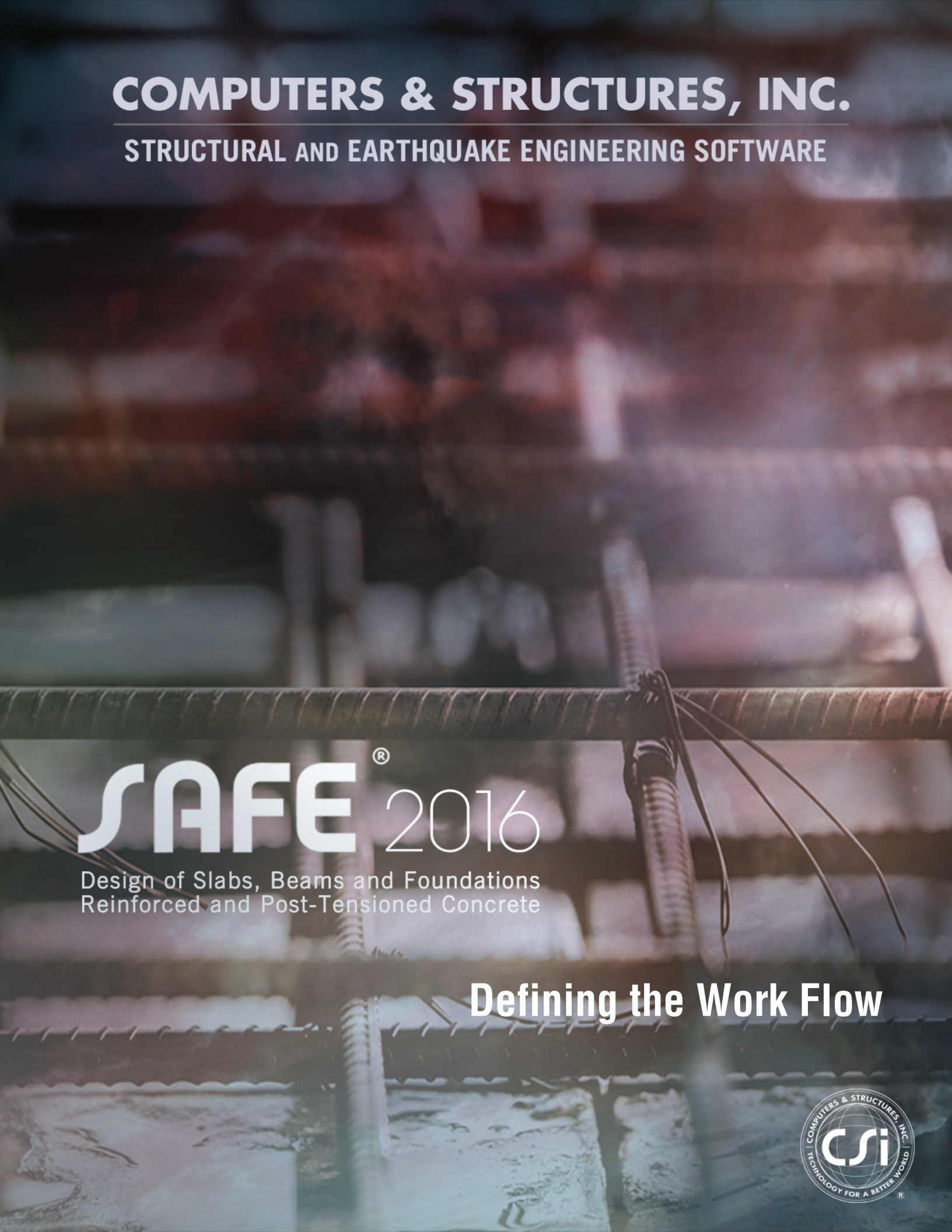


COMPUTERS & STRUCTURES, INC.

STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

SAFE® 2016

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

Defining the Work Flow

COMPUTERS & STRUCTURES INC.

SAFE®

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

Defining the Work Flow

ISO SAF112816M2 Rev. 0
Proudly developed in the United States of America
November 2016

Copyright

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

Computers & Structures, Inc.
www.csiamerica.com

info@csiamerica.com (for general information)
support@csiamerica.com (for technical support)

DISCLAIMER

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

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

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

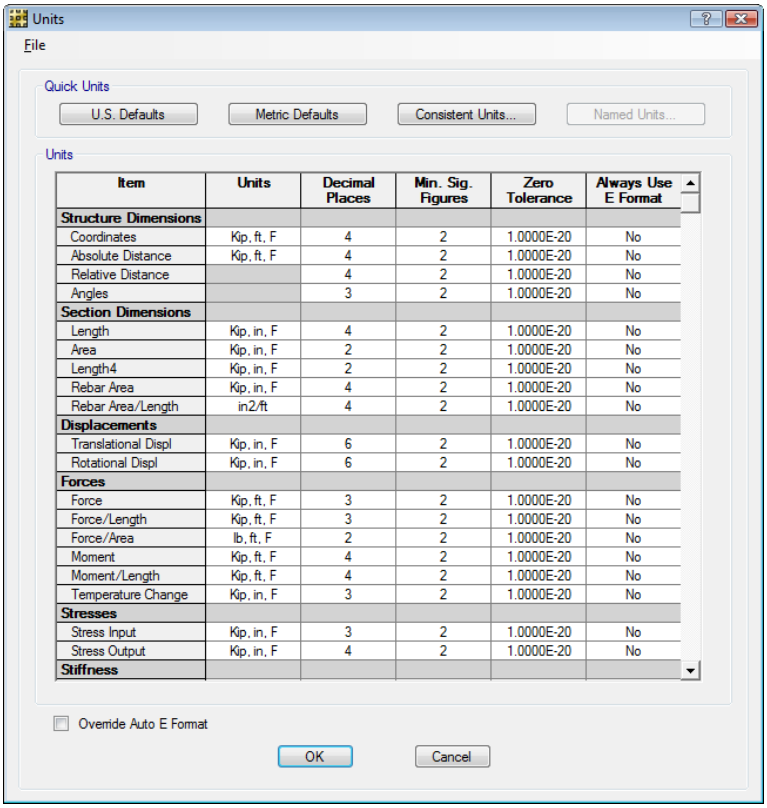
Contents

Chapter 1
Set the Units

This chapter describes how to set the database units.

The units for a new model are set by clicking on the units **Modify/Show** button on the *New Model Initialization* form (see Chapter 2), or for an existing model, by clicking the **Options menu > Units** command or the **Units** button located in the lower right-hand corner of the SAFE window. Any of those actions will display the *Units* form shown in Figure 1-1. Use that form to set the units for all input, output, and display values. Quick unit selection buttons instantly set all unit values to US defaults, Metric defaults, or to Consistent values. The units for individual items may be changed by clicking in the associated units edit box and selecting units from the drop-down list.

The units selected for a particular item in the *Units* form will be displayed adjacent to the item's edit box when doing input. To use units for inputting an item other than those selected in the *Units* form, enter the unit designation in the edit box. For example, assume that the length unit for section dimensions is set to inch in the *Units* form, but when entering the thickness for a mat foundation slab object, you wish to use feet. To do this, simply type "3ft" in the edit box; the program will automatically convert this value to 36 inches.



Item	Units	Decimal Places	Min. Sig. Figures	Zero Tolerance	Always Use E Format
Structure Dimensions					
Coordinates	Kip, ft, F	4	2	1.0000E-20	No
Absolute Distance	Kip, ft, F	4	2	1.0000E-20	No
Relative Distance		4	2	1.0000E-20	No
Angles		3	2	1.0000E-20	No
Section Dimensions					
Length	Kip, in, F	4	2	1.0000E-20	No
Area	Kip, in, F	2	2	1.0000E-20	No
Length4	Kip, in, F	2	2	1.0000E-20	No
Rebar Area	Kip, in, F	4	2	1.0000E-20	No
Rebar Area/Length	in2/ft	4	2	1.0000E-20	No
Displacements					
Translational Displ	Kip, in, F	6	2	1.0000E-20	No
Rotational Displ	Kip, in, F	6	2	1.0000E-20	No
Forces					
Force	Kip, ft, F	3	2	1.0000E-20	No
Force/Length	Kip, ft, F	3	2	1.0000E-20	No
Force/Area	lb, ft, F	2	2	1.0000E-20	No
Moment	Kip, ft, F	4	2	1.0000E-20	No
Moment/Length	Kip, ft, F	4	2	1.0000E-20	No
Temperature Change	Kip, in, F	3	2	1.0000E-20	No
Stresses					
Stress Input	Kip, in, F	3	2	1.0000E-20	No
Stress Output	Kip, in, F	4	2	1.0000E-20	No
Stiffness					

Figure 1-1 Units drop-down list

The units selected for display items in the *Units* form will be shown in the display window title bar and adjacent to cursor values. To change the display units, use the **Options menu > Units** command or the **Units** button to access the *Units* form and select units for the display item; the display window will be updated after the *Units* form is closed.

Although any units may be used at any time while working with the model, e.g., inch units for beam sections and feet units for grid layout, it is important to select an appropriate unit set before starting the model to reduce the possibility of round-off error. In other words, if work is being completed primarily in US units, US default units should be selected as the initial units. In such a case, Metric defaults would not be a good choice for the units because a conversion would always be required when saving the model, which could result in numerical variations.

Chapter 2
Start a Model

This chapter describes how to create a SAFE model using:

- A basic grid system
- One of the built-in templates
- A model exported from ETABS

These three options give a great deal of flexibility in choosing the method that is best suited to each project. As Building Information Modeling (BIM) software becomes more tightly integrated into the development of structural models, the advantages of importing a slab from the BIM-compatible-program ETABS become even more significant. Loading and properties can be assigned and applied to any slab model regardless of the method used to create it.

2.1 Begin a Model Using 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 2-1 will display.

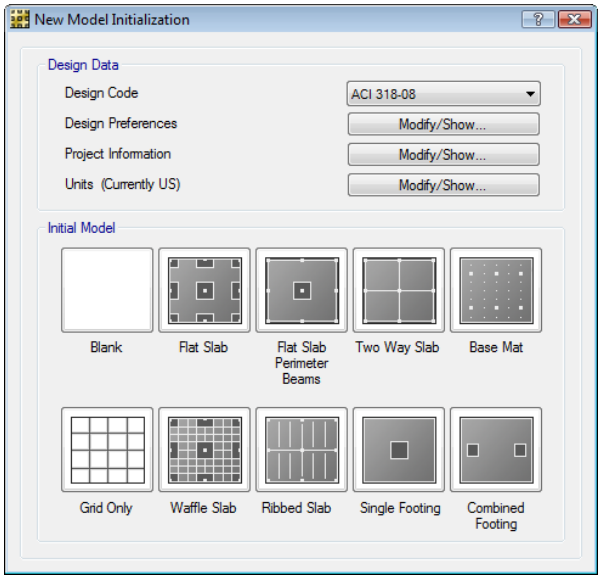


Figure 2-1 New Mode Initialization form

The *New Model Initialization* form contains a grid only button, five slab templates, one base mat template, and two templates for footings, as well as the option to start a model with a "blank" screen. To start a model using a grid system only, set the desired default units by clicking the Units **Modify/Show** button on the *New Model Initialization* form, and then click the **Grid Only** button. The *Coordinate System Definition* form shown in Figure 2-2 will display.

The *Coordinate System Definition* form is used to specify the number of horizontal grid lines and their 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. Changes to the labels and spacing of the grid lines may be made while in the *Coordinate System Definition* form by clicking the **Grid Labels** or **Edit Grid** buttons, respectively. Click the **OK** button to accept the input made on the form, or click **Cancel** to cancel the input.

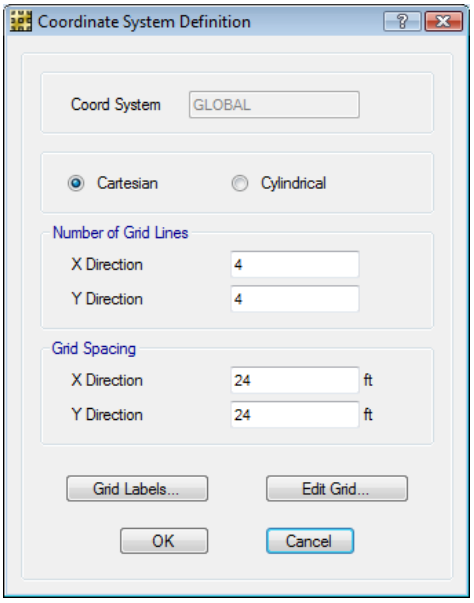


Figure 2-2 Coordinate System Definition form

To alter the spacing and labels of the grid lines after the grids have been generated, click the **Define menu** > **Coordinate Systems** command. This will display the *Coordinate Systems* form. Select the grid to be edited, and click the **Modify/Show System** button to display the *Coordinate System* form shown in Figure 2-3, which has options for changing grid labels and spacing for each grid line in the X or Y directions, as well as specifying general grid lines. The form also has options related to how the grid lines display, including an option to make the grid lines invisible. A reference "datum" elevation also may be specified, which is useful when analyzing multiple floors from the same building.

The definition of a grid system is important for the following reasons:

- Objects snap to grid lines when drawn in the model; thus, grid lines should 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.

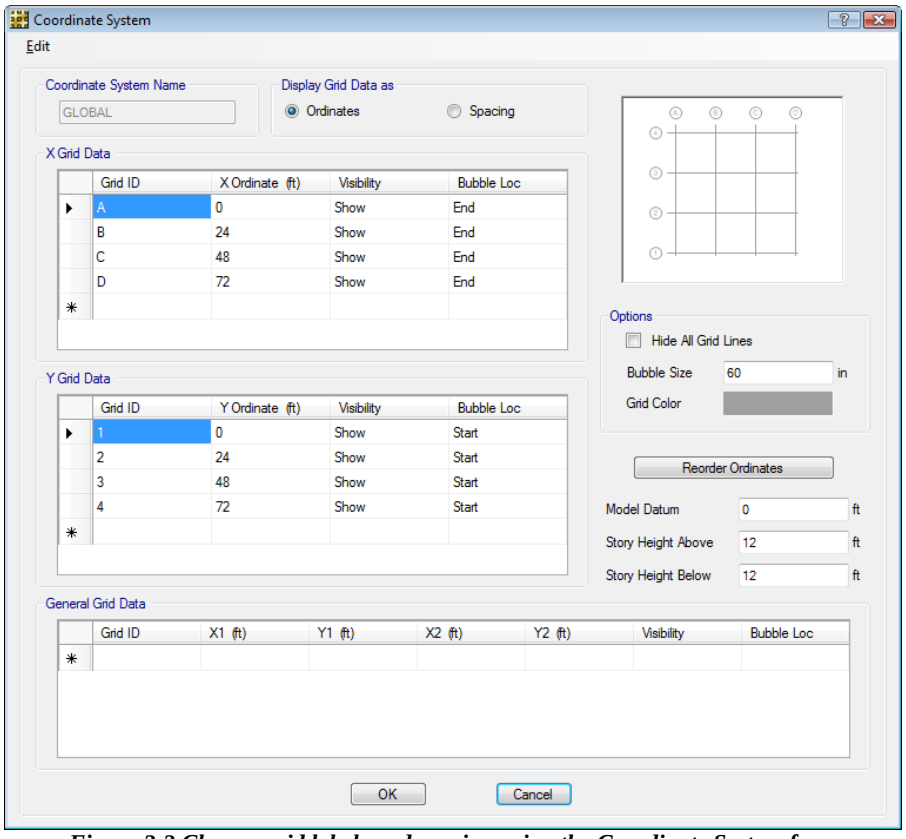


Figure 2-3 Change grid labels and spacing using the Coordinate System form

2.2 Begin a Model Using Templates

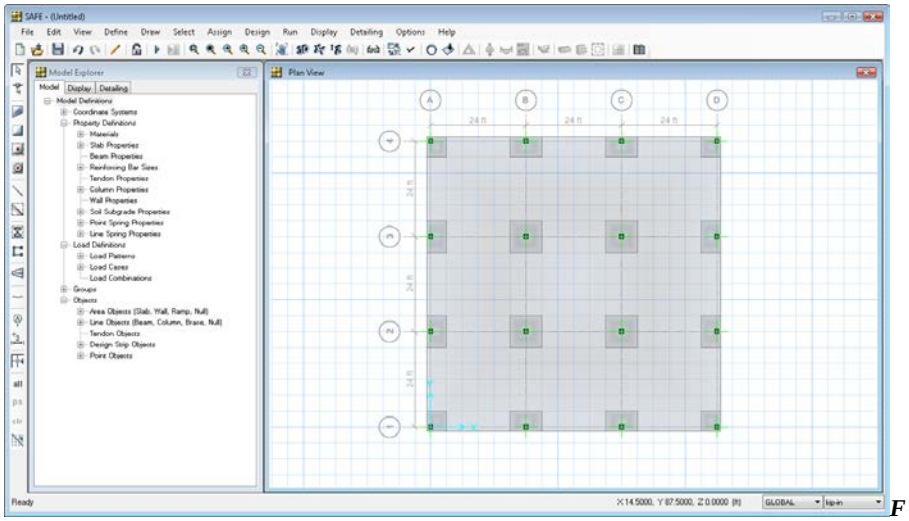
SAFE also has an option to begin a model using templates. Begin a model using templates by following the same procedure used to start a model with only a grid, namely clicking the **File menu > New Model** command. The *New Model Initialization* form shown in Figure 2-1 will again display.

Template models provide a quick, easy way of starting a model. They automatically add structural objects with appropriate properties to a model. It is highly recommended that you start your models using templates whenever possible.

Choose any of the templates by clicking the associated button. When a template button is clicked, a form for that template will display. Use the form to specify various data for the template model, such as plan dimensions and spacing, slab or mat thicknesses, loads, and column dimensions.

Important Note: When using the templates, beams and ribs are modeled using depths equal to the dimension from the top of the slab to the bottom of the beam or rib. Beams are modeled as line objects, while ribs are modeled using equivalent slab properties.

After specifying data for a template, the model will appear in the display window. The default display will show a "Plan View," as shown in Figure 2-4.



F

Figure 2-4 An active Plan View (as indicated by the highlighted title bar) of a template model

The *Model Explorer*, containing all of the model definition data, also will be displayed to the left of the plan view. Use the **Options menu** > **Windows** command to change the number of windows displayed.

Note that the *Plan View* is active in Figure 2-4. When a window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window.

2.3 Begin a Post-Tensioned Model Using Templates

The steps necessary to begin a post-tensioned model from the templates are the same as outlined in the previous section, with the addition that the *Add P/T* check box on the Flat Slab template data form must be checked. Click the **P/T Data** button to display the form shown in Figure 2-5 and define the post-tensioning data to be used in the initial model. Note that post-tensioning currently is available for the flat slab template only, although tendons can be added manually to any model.

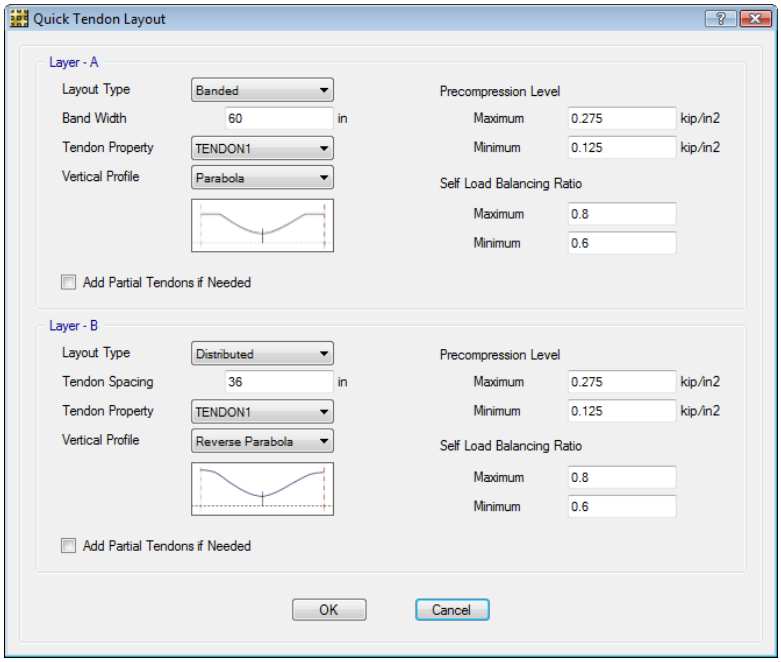


Figure 2-5 Define initial post-tensioning data using the Quick Tendon Layout form

The *Quick Tendon Layout* form allows the definition of the tendon layout type, band width or spacing, property, and vertical profile, along with the desired precompression level and amount of self-weight load balancing.

The methodology used for the final determination of the tendon vertical profiles and other post-tensioning design capabilities is described in detail in the *Post-Tensioning Concrete Design* manual.

2.4 Import a Model from ETABS

A very useful and powerful way to start a model in SAFE is to import the model from ETABS. Floor slabs or basemats that have been modeled in ETABS can be exported from ETABS using the **File menu > Export > Save Story as SAFE V12 .f2k Text File** command. That command will bring up the form shown in Figure 2-6.

From that form, the appropriate floor load option can be selected, along with the desired load cases. After the model has been exported as an .f2k text file, the same file can then be imported into SAFE using the **File menu > Import** command. Using the export and import steps will complete the transfer of the slab geometry, section properties, and loading for the selected load cases. The design strips need to be added to the imported model since design strips are not defined as part of the ETABS model.

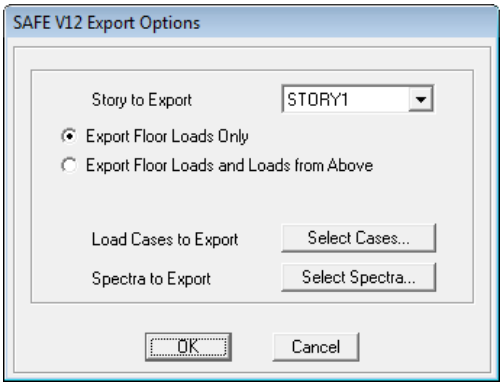


Figure 2-6 ETABS to SAFE Export form

2.5 Saving a Model

It is good practice to save your model often. To save the model, click the **File menu > Save** command or the **Save**  button. The first time the model is saved, SAFE will prompt for a file name using the *Save Model File As* form. An auto-save increment can be specified using the **Options menu > Auto Save Model** command. Note that auto-save is turned off by default because after the model is saved, the undo history is reset and SAFE remembers only steps taken since the last save.

Chapter 3
Define Materials

Materials are named entities that are referenced by slab, beam, column, wall, and tendon properties and slab rebar objects. Materials define properties such as the modulus of elasticity, Poisson's ratio, and material strengths.

3.1 Modify Existing Materials

Default and previously defined materials may be reviewed using the **Define menu > Materials** command to display the *Materials* form shown in Figure 3-1.

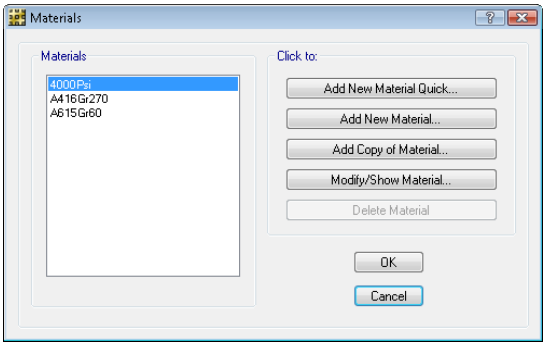


Figure 3-1 Define Materials form

Select a material from the list and click the **Modify/Show Material** button to display the *Material Property Data* form shown in Figure 3-2.

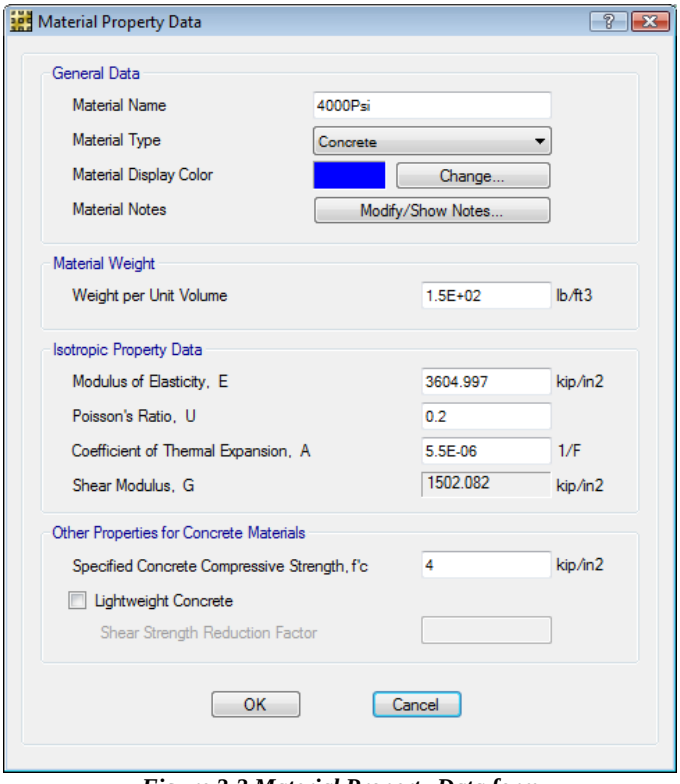


Figure 3-2 Material Property Data form

Alternatively, this form also could be displayed by expanding the *Materials* item located in the *Model Explorer* window, and then double clicking on the appropriate material property. Modify any of the properties as needed, and then click the **OK** button to accept the changes made on the form, or click **Cancel** to cancel the changes.

3.2 Input New Materials

To input a new material property, click the **Define menu > Materials** command to again display the *Materials* form shown in Figure 3-1. Click the **Add New Material Quick** button to display the *Quick Material Definition* form shown in Figure 3-3. Use the form to select concrete, rebar, tendon, or steel properties from common specifications.

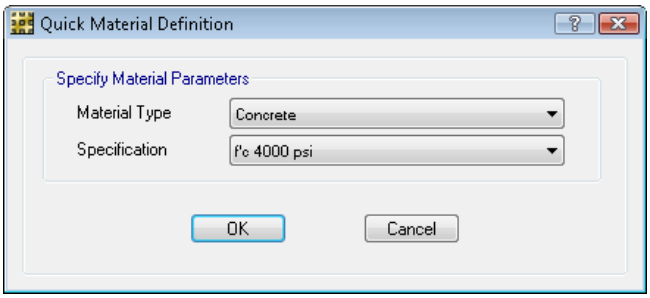


Figure 3-3 Quick Material Definition form

If the material required does not exist on the *Quick Material Definition* form, click the **Cancel** button and return to the *Materials* form. Next, click the **Add New Material** button, which will display the *Material Property Data* form similar to that shown in Figure 3-2. On the *Material Property Data* form, any material property values may be specified. Click the **OK** button to accept the changes and save the new material, or click **Cancel** to cancel the material property definition.

Chapter 4
Define Properties

Properties are defined to reflect the structural behavior of point, line, and area objects in the model. Properties may be structural objects, such as slabs and columns, or supports, such as soil springs.

4.1 Input Structural and Support Properties

Structural properties, that is, slab, beam, tendon, column, and wall properties that contain section definitions, are named entities that must be specified using the commands on the **Define menu** before assigning them to the model, as described later in this chapter and in Chapter 7. If you use a property in a model, for example a beam or column property, any changes to the definition of the property will automatically apply to the model. A named property has no effect unless it is used in the model.

Spring support properties (i.e., point, line, and soil) may be assigned to point, beam, and slab objects, and similar to structural properties, they are named entities that must be specified before they are assigned.

Table 4-1 identifies the structural and spring support property 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** keyboard function key when the forms are displayed.

TABLE 4-1 Property Definitions

Property	Object Type	Input Form
Slab Properties (Drop, Stiff, Mat)	Slab/Area	Slab Property Data
Beam Properties	Beam/Line	Beam Property Data
Tendon Properties	Tendon	Tendon Property Data
Column Properties	Column/Brace	Column Property Data
Wall Properties	Wall/Ramp	Wall Property Data
Soil Subgrade Properties	Area	Soil Subgrade Property Data
Point Spring Properties	Point	Point Spring Property Data
Line Spring Properties	Line	Line Spring Property Data

The input forms identified in Table 4-1 are preceded by standard forms that have options to add new definitions, add a copy of an existing definition, modify or review existing definitions, and delete existing definitions. Each property definition should have a unique name that is user-defined or should use the program-suggested name. It is important that the names make it easy to assign them to the various objects correctly.

Depending on the type of property, the input forms may include user-defined analysis and design parameters, such as thicknesses, top and bottom cover, spring constants, and other parameters. In addition to those items, the *Slab Property Data* form also has a parameter for specifying the type of slab (e.g., slab, waffle, drop), which controls not only the analysis formulation, but also the design and detailing.

Previously defined properties, either structural or spring support, also may be accessed through the *Model Explorer* window. Expand the desired item and double click on the appropriate property to display the associated input form. As an example, to review a previously defined beam property, expand the *Beam Properties* item, and then double click on the appropriate beam property to display the *Beam Property Data* form shown in Figure 4-1.

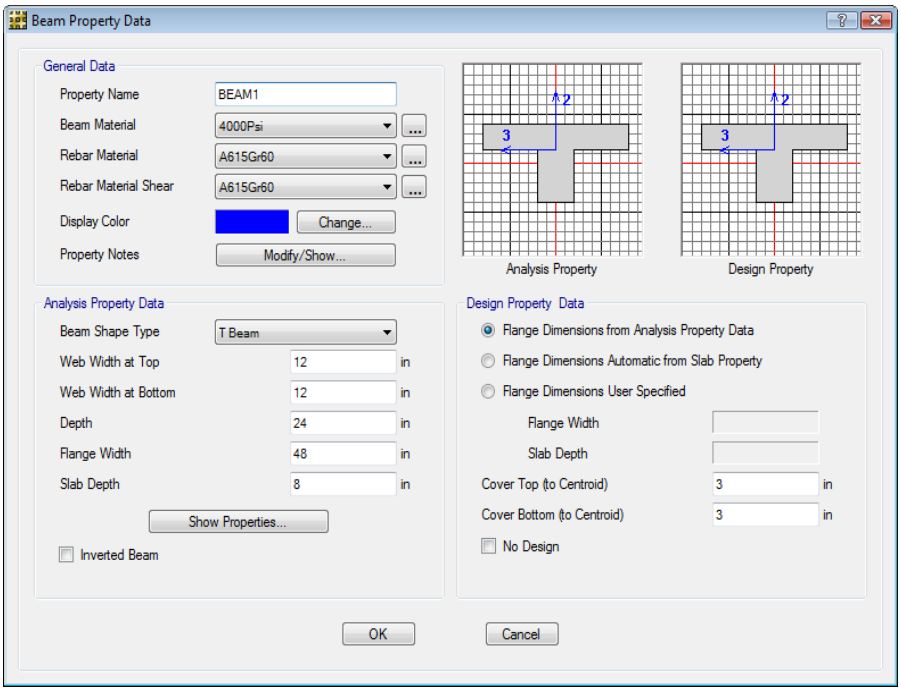


Figure 4-1 Beam Property Data form

Additional material properties may be defined by clicking the **Add Material** button next to the *Beam Material*, *Rebar Material*, or *Rebar Material Shear* drop-down lists. The **Add Material** button opens the form used to define the item in the adjacent drop-down list, which in this case is the *Materials* form.

Modify any of the values as needed, and click the **OK** button to accept the changes made in the form, or click **Cancel** to cancel the changes.

Chapter 5
Draw Objects

Slab, beam, tendon, column, wall, and point objects are used in SAFE to represent the slab or mat structure. Properties are defined and assigned to the model to reflect the structural characteristics. These objects can be added to any model regardless of how the model was initially created. The Draw commands allow customization and editing of any model.

5.1 Draw Slabs/Areas

Slabs or areas are used most commonly to create the main slabs (including footings and mats) or openings in slabs. To draw a slab object, activate the Plan View. Click the **Draw Slabs/Areas** button or select the **Draw menu > Draw Slabs/Areas** command to bring up the form shown in Figure 5-1.

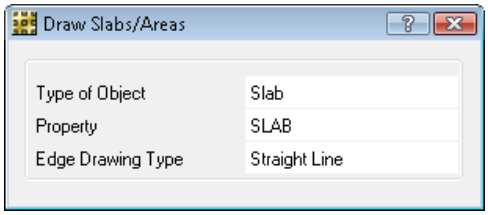


Figure 5-1 Draw Slabs/Areas floating form

Note: This chapter makes frequent reference to forms that "float." That is, a form remains visible when the cursor is moved over the model and the mouse button is clicked in the active window. Because the form remains visible, changes can be made to it without reusing the command required to access it, which differs from most forms in the program. Floating forms are used in the draw mode so that the parameters for the object being drawn can be changed easily during drawing operations. Also note that only one floating form can be visible at a time (i.e., *Draw Slabs/Areas, Draw Rectangular Slabs/Areas, Draw Beams/Lines* and so on.)

The floating form provides a *Property* drop-down list of the previously defined properties (see Chapter 4) that reflect the slab/area object to be drawn next. To change an entry on the form, click on it and make a new selection from the drop-down list or type new information into the edit box, as appropriate.

TIP: If the slab/area object is to be drawn using grid lines, ensure that the appropriate snap options are active by clicking the **Draw menu** > **Snap Options** command to display the *Snap Options* form shown in Figure 5-2. On that form, verify that the *Grid Intersections* and *Points* boxes are checked; this is the default setting. Click the **OK** button to accept the settings, or the **Cancel** button to exit without saving any changes made.

Snap options help make creating a model simple. Along with specific types of snap options, such as endpoints, intersections, and the like, there also are intelligent snaps that locate nearby objects and provide snap dimensions. The snaps can be turned on or off by using the **Snap** button in the left toolbar.

With the parameters in the *Draw Slabs/Areas* floating form set, click the left mouse button once at a grid intersection (or any other position in the Plan View) to begin the area object at that location. Next, move around the perimeter of the area object, clicking once at additional locations, to draw the outline of the object. Press the Enter key on the keyboard to complete the drawing of the object.

If a mistake is made while drawing the object, click the **Select Object** button to change from Draw mode to Select mode. Next, click the **Edit menu > Undo Area Add** command to remove the area object just drawn. Note that the floating form disappears (closes) when the **Select Object** button is clicked. It is also possible, while still in the draw mode, to undo the last point defined for the area/slab object by pressing the Backspace key on the keyboard.

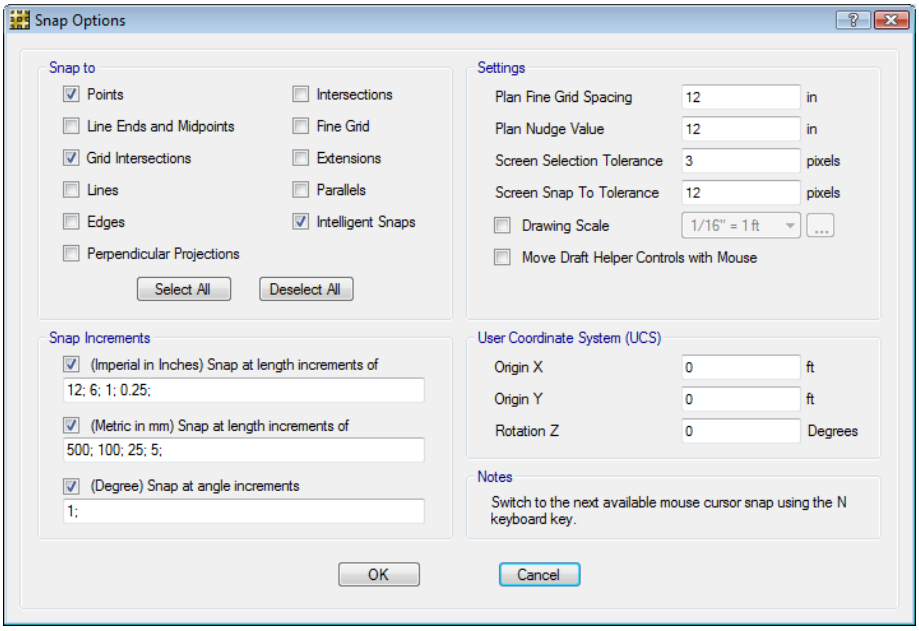


Figure 5-2 Snap Options form

5.2 Draw Rectangular Slabs/Areas

The **Draw Slabs/Areas** command previously described generates slab/area objects that can be of arbitrary shape with three or more sides. Often, slab systems are rectangular and can be quickly drawn by clicking the **Draw Rectangular Slabs/Areas** button, or by selecting the **Draw menu > Draw Rectangular Slabs/Areas** command. The *Draw Rectangular Slabs/Areas* floating form shown in Figure 5-3 will display.

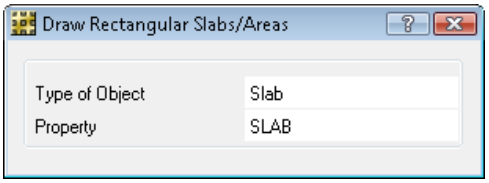


Figure 5-3 Draw Rectangular Slabs/Areas form

With the parameters in the form set, click the left mouse button once at a grid intersection (or any other position in the Plan View) to locate one corner of the rectangular slab/area object. While holding the left mouse button down, drag the cursor to the diagonally opposite corner, and release the mouse button to complete the drawing of the rectangular slab/area object.

To leave the Draw mode, click the **Select Object** button.

5.3 Quick Draw Slabs/Areas

The **Quick Draw Slabs/Areas** command allows for slab/area objects bounded by a bay or multiple bays to be drawn using a single click. To use this option, activate the Plan View, and click the **Quick Draw Slabs/Areas** button, or select the **Draw menu** > **Quick Draw Slabs/Areas** command. The *Draw Quick Draw Slabs/Areas* floating form shown in Figure 5-4 will display.

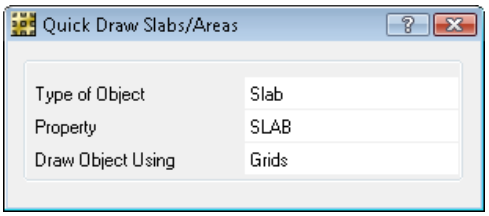


Figure 5-4 Quick Draw Slabs/Areas form

With the *Draw Object Using* option set to grids, click the left mouse button once in the center of the grid lines outlining a bay, to draw a slab/area object covering the entire bay. Alternatively, slab/area objects can be added over multiple bays by holding the left mouse button down

and dragging a window around the desired bays. Make sure the desired bays are fully enclosed by the window drawn.

If the *Draw Object Using* option is set to points, dragging a window around a group of points will result in an area object being drawn that uses the points as corners. If the *Draw Object Using* option is set to lines, dragging a window around a set of lines that create a closed polygon will result in an area object being drawn with the shape of the polygon. If the *Draw Object Using* option is set to arch. layer, dragging a window around a region of an architectural plan will result in area objects being drawn at every closed polygon entity that exists on the displayed architectural layer.

5.4 Quick Draw Areas Around Points

Often it is necessary to add multiple area objects at support points to model drop panels. These objects can be drawn quickly by clicking the **Quick Draw Areas Around Points** button, or by selecting the **Draw menu > Quick Draw Areas Around Points** command. The *Quick Draw Area Around Points* floating form shown in Figure 5-5 will display.

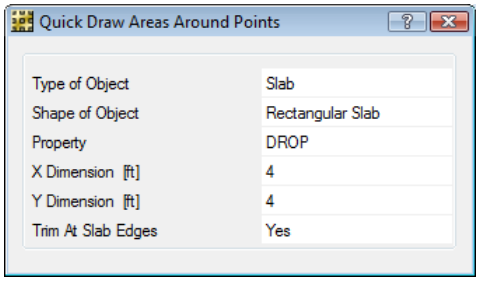


Figure 5-5 Quick Draw Area Around Points form

With the parameters in the form set, click the left mouse button once at a support point to draw a rectangular area object of size X by Y.

5.5 Draw Design Strips

Design strips define the location and layout of program calculated slab reinforcement and can be used for tendon placement. Design strips are defined using polylines that may be multilinear and non-orthogonal. Associated with these design strip polylines are widths on each side of the drawn polyline. To draw design strips, click the **Draw Design Strips** button, or select the **Draw menu > Draw Design Strips** command. The *Draw Design Strips* floating form shown in Figure 5-6 will display.

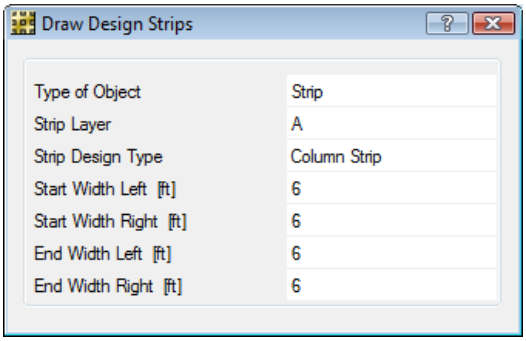


Figure 5-6 Draw Design Strips form

Change any entry in the floating form by clicking on it and making a new selection from the drop-down list or typing new information into the edit box, as appropriate. When modeling a slab with post-tensioning, *Strip Design Type* will typically be set to column strip only, while a slab without post-tensioning will use both column and middle strips depending upon the location of the strip. With the parameters set, click the left mouse button once in the Plan View to locate the starting point of the design strip. Continue to click at points along the design strip to define each segment of the strip. After the last point has been defined, press the **Enter** key on the keyboard to complete the drawing of the object.

To leave the Draw mode, click the **Select Object** button.

5.6 Draw Beams/Lines

To draw a beam/line object, activate the Plan View. Click the **Draw Beams/Lines** button, or select the **Draw menu > Draw Beams/ Lines** command. The *Draw Beams/Lines* floating form shown in Figure 5-7 will display.

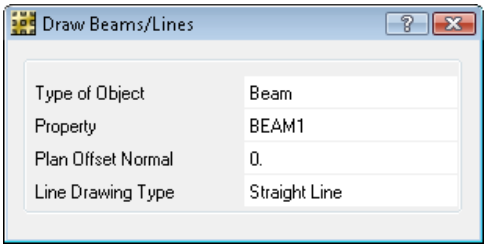


Figure 5-7 Draw Beams/Lines form

Similar to slabs/areas, the floating form provides a *Property* drop-down list of the previously defined properties (see Chapter 4) that can be assigned to the beam. To change an entry on the form, click on it and make a new selection from the drop-down list or type new information into the edit box, as appropriate.

With the parameters in the floating form set, click the left mouse button once anywhere in the Plan View to start drawing the beam/line. Move the cursor to the end point of the beam/line, and left click to complete the drawing of the beam/line. Moving the cursor to the next end point and left clicking will draw another segment. To start a new beam/line that is independent of the previous one, right click to discontinue the previous beam/line, then left click at the starting point of the new beam/line. To leave the Draw mode, click the **Draw menu > Select Object** command, or the **Select Object** button.

5.7 Quick Draw Beams/Lines

Beams/lines can be drawn quickly on the grid or between points by clicking the **Quick Draw Beams/Lines** button or the **Draw menu >**

Quick Draw Beams/Lines command. The *Quick Draw Beams/Lines* floating form similar to that shown in Figure 5-7 will display.

With the *Draw Object Using* option set to grids, change any other entry in the floating form by clicking on it and making a new selection from the drop-down list or typing new information into the edit box, as appropriate. With the parameters in the floating form set, left click once in the Plan View on a grid line to place a beam/line object.

Alternatively, beams/lines on multiple grid lines can be generated by holding the left mouse button down and dragging a window around the desired grid lines or points. Make sure the desired grid lines and points are fully enclosed by the window drawn.

If the *Draw Object Using* option is set to points, dragging a window around a group of points will result in beam/line objects being drawn that create a closed polygon using the points as corners.

If the *Draw Object Using* option is set to arch. layer, dragging a window around a region of an architectural plan will result in beam/line objects being drawn at every line entity that exists on the displayed architectural layer.

To leave the Draw mode, click the **Select Object** button.

5.8 Draw Tendons

Tendons are used to input post-tensioning forces into the model. To draw a tendon, activate the Plan View. Click the **Draw menu > Draw Tendons** command, or the **Draw Tendons** button. The *Draw Tendons* floating form shown in Figure 5-8 will display.

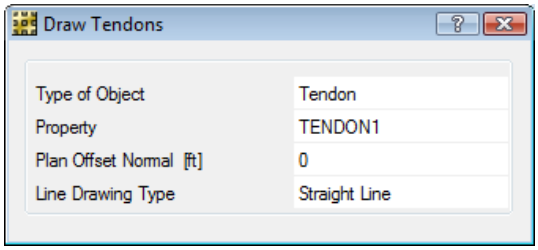


Figure 5-8 Draw Tendons form

Similar to slabs/areas and beams/lines, this floating form provides a *Property* drop-down list of the previously defined properties (see Chapter 4) that can be assigned to tendons. Change any entry in the floating form by clicking on it and making a new selection from the drop-down list or typing new information into the edit box, as appropriate.

The vertical profile of the tendon can be defined by selecting the desired tendon and using the **Edit menu > Add/Edit Tendons > Edit Vertical Profile** command to display a form similar to that shown in Figure 5-9.

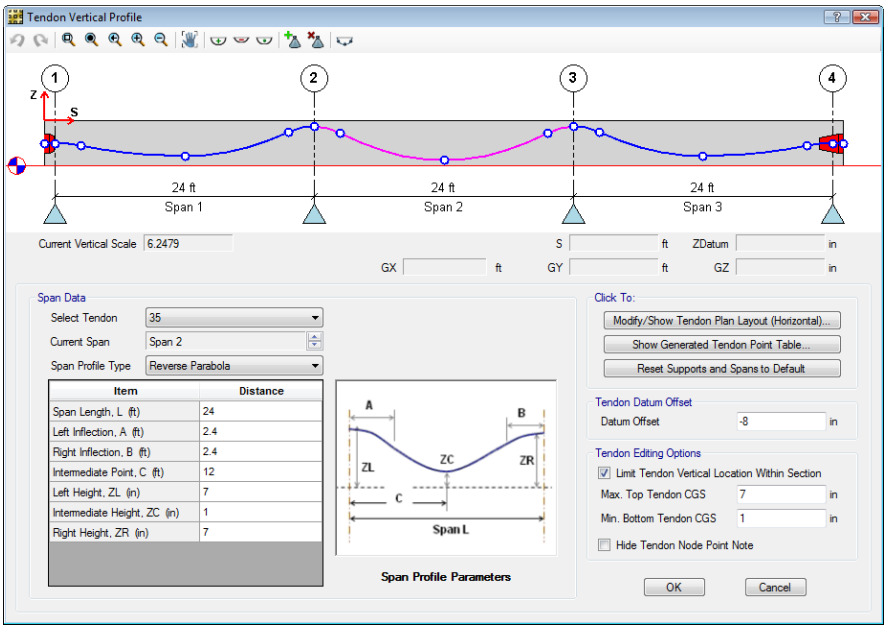


Figure 5-9 Tendon Vertical Profile form

The *Tendon Vertical Profile* form can be used to define the profile type and the vertical profile. A detailed description of the post-tensioning design methodology is described in the *Post-Tensioning Concrete Design* manual.

5.9 Draw Columns

To draw a column object, click the **Draw Columns** button or use the **Draw menu > Draw Columns** command. The *Draw Columns* form shown in Figure 5-10 will display.

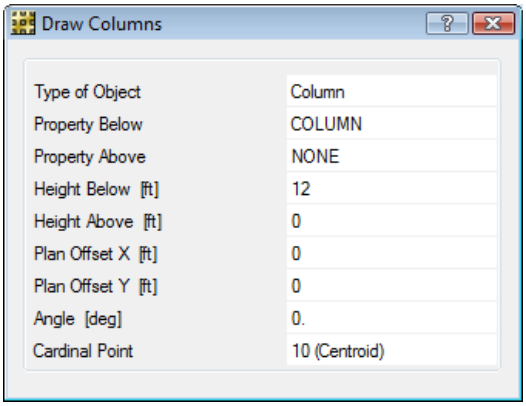


Figure 5-10 Draw Columns form

The form provides *Property Below* and *Property Above* drop-down lists of the previously defined properties (see Chapter 4) that can be assigned to the column. The corresponding heights of the column also are specified on this form. Review the definitions and drawing controls (*Plan Offset X, Plan Offset Y, Angle, Cardinal Point*) shown in the form before drawing the column. Change any entry in the form by clicking on it and making a new selection from the drop-down list or typing new information into the edit box, as appropriate.

With the parameters in the *Draw Columns* form set, left click anywhere to locate the column object. An outline of the column shape will appear in plan views and lines will be shown in 3-D views. Continue to place other columns by left clicking.

If a mistake is made while drawing, click the **Select Object** button to change from Draw mode to Select mode. Next click the **Edit menu > Undo Column Object Add** command. After all column objects have been drawn, click the **Draw menu > Select Object** command to leave the Draw mode.

5.10 Draw Walls

To draw a wall object, click the **Draw Walls** button or use the **Draw menu > Draw Walls** command. The *Draw Walls* form shown in Figure 5-11 will display.

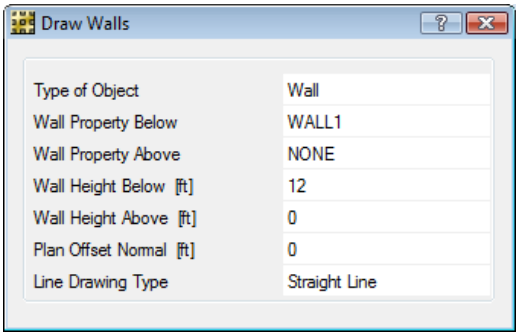


Figure 5-11 Draw Walls form

The form provides *Wall Property Below* and *Wall Property Above* drop-down lists of the previously defined properties (see Chapter 4) that can be assigned to the wall. The corresponding heights of the wall also are specified on this form. Review the definitions and drawing controls (*Plan Offset Normal*, *Line Drawing Type*) shown in the form before drawing the wall. Change any entry in the form by clicking on it and making a new selection from the drop-down list or typing new information into the edit box, as appropriate.

With the parameters in the *Draw Walls* form set, click the left mouse button once anywhere in the Plan View to start drawing the wall. Move the cursor to the end point of the wall, and left click to complete the drawing of the wall. Moving the cursor to the next end point and left clicking will draw another segment. To start a new wall that is independ-

ent of the previous one, right click to discontinue the previous wall, then left click at the starting point of the new wall.

If a mistake is made while drawing, click the **Select Object** button to change from Draw mode to Select mode. Next click the **Edit menu > Undo Wall Add** command. After all wall objects have been drawn, click the **Draw menu > Select Object** command to leave the Draw mode.

5.11 Draw Dimension Lines

Dimension lines help with the construction and verification of model geometry. Dimension lines may be added to the Plan View by selecting the **Draw menu > Draw Dimension Lines** command or clicking the **Draw Dimension Lines** button. Left click once to locate the first extension line and click again to mark the location of the second extension line. Move the mouse to show the orientation and position of the extension and dimension lines; click at the desired location. If the extension lines were located initially by snapping to points or other objects, the dimension lines will remain attached to those objects if the objects move. Dimension lines are saved with the model.

5.12 Draw Slab Rebar

Slab rebar objects define the location and layout of user-defined slab reinforcement and are defined using polylines that may be multilinear. Associated with these slab rebar object polylines are widths on each side of the drawn polyline. The slab rebar object is typically used to compare user specified slab reinforcing against program calculated reinforcing and to specify reinforcing for nonlinear cracking analyses.

To draw a slab rebar object, click the **Draw Slab Rebar** button or use the **Draw menu > Draw Slab Rebar** command. The *Draw Slab Rebar Object* form shown in Figure 5-12 will display.

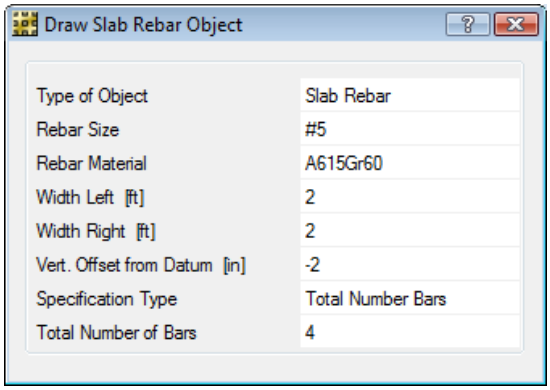


Figure 5-12 Draw Slab Rebar Object form

The form provides *Rebar Size* and *Rebar Material* drop-down lists of the previously defined properties (see Chapter 4) that can be assigned to the rebar object. The corresponding widths of the slab rebar object, as well as the vertical offset (top & bottom rebar) also are specified on this form. Depending upon the specification type, review either the total number of bars or the maximum bar spacing before drawing the slab rebar object. Change any entry in the form by clicking on it and making a new selection from the drop-down list or typing new information into the edit box, as appropriate.

With the parameters in the *Draw Slab Rebar Object* form set, click the left mouse button once anywhere in the Plan View to start drawing the slab rebar object. Move the cursor to the end point of the slab rebar, and left click to complete the drawing of the slab rebar object. Moving the cursor to the next end point and left clicking will draw another segment. To start a new slab rebar object that is independent of the previous one, right click to discontinue the previous slab rebar object, then left click at the starting point of the new rebar object.

If a mistake is made while drawing, click the **Select Object** button to change from Draw mode to Select mode. Next click the **Edit menu** > **Undo Slab Rebar Add** command. After all slab rebar objects have been drawn, click the **Draw menu** > **Select Object** command to leave the Draw mode.

Chapter 6
Select Objects

This chapter describes the options available for making selections in SAFE. Selections are used to identify existing objects to which the next operation will apply. It is necessary to first select objects in order to assign properties to objects. Selections can be made graphically or by referencing various properties of the objects. Selection operations are cumulative, i.e., each selection operation adds objects to the current set of selected objects and each deselection operation removes objects from the set.

Selected objects are shown graphically in the display windows with dashed lines. The number of selected objects of each type is shown in the status bar. It is always a good idea to check the status bar when performing selection operations.

6.1 Selection by Graphical Methods

To enable the basic select mode, click the **Select Object** button on the toolbar, or click the **Select menu > Select > Pointer/Window** command. SAFE has three basic methods of selecting objects in this mode:

- **Left click:** Click the left mouse button when the pointer is on an object to select it. If there are multiple objects, one on top of the other, hold down the **Shift** key on the keyboard and left click on the objects. The *Selection List* form similar to that shown in Figure 6-1 will display 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.

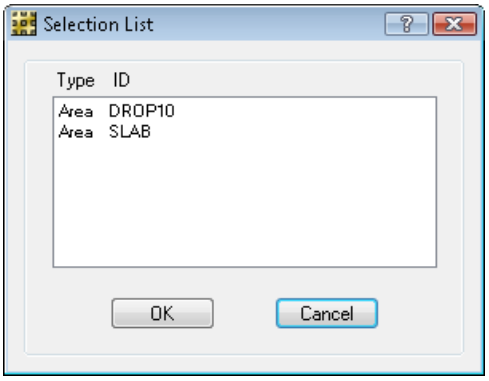


Figure 6-1 Selection List form

- **Enclosing Window (left to right):** Draw a window around objects to select them by positioning the mouse pointer to the left and beyond the limits of the objects. Press and hold down the left mouse button and drag the mouse to a position to the right of the objects. Release the left mouse button to complete the selection.

  As the mouse is dragged, a "rubber band window" will display. 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.

- **Intersecting Window (right to left):** Position the mouse pointer to the right and beyond the limits of the objects to be selected. Press and hold down the left mouse button and drag the mouse to a position to the left of the objects. Release the left mouse button to complete the selection. Any visible object that is fully or partially enclosed in the window will be selected.

In addition to the default pointer/window selection mode, three other graphical selection options are available:

- **Poly:** To select one or more objects by enclosing them within a polygon, click the **Select menu > Select > Poly** command. Position the mouse pointer beyond the limits of the objects to be selected. Click the left button on the mouse to indicate the first point of the polygon. Move the mouse pointer to the next point of the polygon and click again. As the mouse is moved, a "rubber band polygon" will display. After all points of the polygon have been input, complete the selection command by pressing the **Enter** key on the keyboard. Any visible objects that are completely inside the rubber band polygon are selected.

- **Intersecting Poly:** To select one or more objects crossed by a polygon, click the **Select menu > Select > Intersecting Poly** command. Draw the polygon in the same manner as the previous command. Any visible objects fully enclosed within or in contact with the rubber band polygon are selected.

- **Intersecting Line:** To select one or more objects by drawing a line through them, click the **Select menu > Select > Intersecting Line** command or the **Select using Intersecting Line**  button. Position the mouse pointer to one side of the objects to be selected. Click the left button on the mouse to indicate the beginning of the line. Move the mouse pointer to the end of the line segment and click again. As the mouse is moved, a "rubber band line" will display. The intersecting line may consist of multiple segments, and after all of the line segments have been input, complete the selection by pressing the **Enter** key on the keyboard. Any visible object that is intersected (crossed) by the rubber band line segments is selected.

After using any of the preceding graphical selection methods, SAFE defaults to the pointer/window selection mode. Thus, the menu commands or buttons must be used each time to access the select using Poly, Intersecting Poly, or Intersecting Line methods.

6.2 Selection by Features

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

TABLE 6-1 Select Submenu Commands

Command / Button	Action
Select > Properties > Material Properties	Select one or more material properties from the *Select Materials* form, and all objects that have been assigned those materials will be selected.
Select > Properties > Slab Properties	Select one or more slab property names from the *Select Slab Properties* form, and all slab objects that have been assigned those slab properties will be selected.
Select > Properties > Slab Rebar Properties	Select one or more slab rebar property names from the *Select Slab Rebar Property* form, and all slab rebar objects that were drawn with those rebar properties will be selected.
Select > Properties > Beam Properties	Select one or more beam property names from the *Select Beam Properties* form, and all beam objects that have been assigned those beam properties will be selected.
Select > Properties > Tendon Properties	Select one or more tendon property names from the *Select Tendon Properties* form, and all tendon objects that have been assigned those tendon properties will be selected.
Select > Properties > Column Properties	Select one or more column property names from the *Select Column Properties* form, and all the column objects that have been assigned those column properties will be selected.
Select > Properties > Wall Properties	Select one or more wall property names from the *Select Wall Properties* form, and all the wall objects that have been assigned those wall properties will be selected.
Select > Properties > Soil Subgrade Properties	Select one or more soil subgrade property names from the *Select Soil Subgrade Properties* form, and all the slab objects that have those soil subgrade properties assigned to them will be selected.
Select > Properties > Point Spring Properties	Select one or more point spring property names from the *Select Point Spring Properties* form, and all the point objects that have those point spring properties assigned to them will be selected.
Select > Properties > Line Spring Properties	Select one or more line spring property names from the *Select Line Spring Properties* form, and all the line objects that have those line spring properties assigned to them will be selected.

TABLE 6-1 Select Submenu Commands

Command / Button	Action
Select > Properties > Design Strip Layers	Select one or more strip layers from the *Select Design Strip Layers* form, and all the strips on those layers will be selected.
Select > Groups	Select from the *Select Groups* form, the names of any collections of objects that have been defined as groups, and those groups will be selected.
Select > Labels	Select a label(s) from the *Select by Labels* form and the objects that have been assigned those labels will be selected.
Select > All	Selects all objects in the model, both visible and invisible objects. Be careful using this command. The Select All button also can be used to execute this command.

6.3 Deselect

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 two sections, except that executing the **Select menu > Deselect** command and an associated submenu item deselects rather than selects objects. 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. This can be accomplished quickly and easily by first using the **Select menu > Select > All** command and then using the **Select menu > Deselect > Properties > Beam Properties** command.

6.4 Invert Selection

Any objects not currently selected may be selected by using the **Select menu > Invert Selection** command, which simultaneously results in all currently selected objects being deselected. This command is helpful with a large model where only a few items should not be selected; select those objects first, and then use the **Invert Selection** command.

6.5 Get Previous Selection

The **Select menu > Get Previous Selection** command reselects the previously selected objects. 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** ps button to reselect the area objects and assign something else to them, such as surface loads.

6.6 Clear Selection

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

Chapter 7
Assign Properties to the Model

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.

In creating a model, slabs/areas, beams/lines, tendons, columns, walls, and point objects are drawn. To enable analysis and design, these objects must have properties assigned. The definition of properties is explained in Chapter 4. The assignments that can be made to an object depend on the type of object. Table 7-1 identifies the assignments that can be made using the **Assign menu** commands, depending on the type of object.

TABLE 7-1 Available Assignments to Objects

Object	Assignment Option	Name of Form(s)
Points	Support Data > Point Restraints	Point Restraints
	Support Data > Point Springs	Point Spring Properties / Point Spring Property Data
Beams	Beam Data > Properties	Beam Properties/Beam Property Data
	Beam Data > Property Modifiers	Property/Stiffness Modification Factors
	Beam Data > End Releases	Assign Beam End Releases
	Beam Data > Insertion Point	Insertion Point

TABLE 7-1 Available Assignments to Objects

Object	Assignment Option	Name of Form(s)
	Support Data > Line Springs	Line Spring Properties/Line Spring Property Data
Tendons	Tendon Properties	Tendon Property Assign
Slabs	Slab Data > Properties	Slab Properties/Slab Property Data
	Slab Data > Property Modifiers	Property/Stiffness Modification Factors
	Slab Data > Vertical Offset	Slab Offset
	Slab Data > Local Axis	Slab Local Axis
	Slab Data > Edge Releases	Slab Edge Releases
	Slab Data > Line Releases	Slab Line Releases
	Slab Data > Rib Locations	Slab Rib Location
	Slab Data > Opening	Slab Opening
	Support Data > Soil Properties	Soil Subgrade Properties/Soil Subgrade Property Data
Columns/ Braces	Column/Brace Data > Properties	Column Properties/Column Property Data
	Column/Brace Data > Property Modifiers	Property/Stiffness Modification Factors
	Column/Brace Data > Local Axis	Column Local Axis
	Column/Brace Data > End Releases	Assign Column End Releases
	Column/Brace Data > Insertion Point	Insertion Point
Walls/ Ramps	Wall/Ramp Data > Properties	Wall Properties/Wall Property Data
	Wall/Ramp Data > Property Modifiers	Property/Stiffness Modification Factors
	Wall/Ramp Data > Opening (Walls Only)	Wall Opening
	Wall/Ramp Data > Normal Offset	Wall/Ramp Offset
Supports	Support Data > Soil Properties	Soil Subgrade Properties/Soil Subgrade Property Data
	Support Data > Line Springs	Line Spring Properties/Line Spring Property Data
	Support Data > Point Restraints	Point Restraints

TABLE 7-1 Available Assignments to Objects

Object	Assignment Option	Name of Form(s)
	Support Data > Point Springs	Point Spring Properties/Point Spring Property Data

Releases, point restraints, insertion points, rib locations, property modifiers, local axes, and openings are assigned directly to objects. These properties can be changed only by making another assignment of that same property to the object. They are not named entities and do not exist independently from the objects.

The assignments made to point, beam, column, tendon, slab, and wall objects can be viewed by clicking the right mouse button while the mouse pointer is on an object. The appropriate *Point Object Information* form, *Beam-Type Line Object Information* form, *Tendon Object Information* form, *Slab-Type Area Object Information* form, and so on, will display.

An object must be selected before executing the desired assignment command (for example, a beam object must be selected before using the **Assign menu > Beam Data** commands). As explained in Chapter 6, using the **Shift** key and left clicking on a location in the model can simplify the process of selecting objects when multiple objects are present at the same location.

As shown in Table 7-1, the availability of commands depends on the type of object selected. The input forms include object assignment-specific input fields that enable refinement of the assignment. Modifications to the assignments can be made by accessing the input forms using the appropriate **Assign menu** commands. Context sensitive help explaining the various forms is available by pressing the **F1** function key on the keyboard while a form is displayed.

Loads also can be assigned to slabs, beams, tendons, and points. These commands are described in Chapter 8.

Chapter 8
Load the Model

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

Note that SAFE automatically creates a load case for each load pattern, and these load cases are assembled into design load combinations in accordance with the specified building code. The design code can be selected using the **Design menu > Design Preferences** command.

8.1 Define Load Patterns

Click the **Define menu > Load Patterns** command or double click on a load pattern name found under the *Load Patterns* item in the *Model Explorer* window to display the *Load Patterns* form shown in Figure 8-1.

Use the form to specify the following information:

- The name of the load pattern. SAFE does not allow use of duplicate names.

- A load type, which can be selected from the *Type* drop-down list.
- A self weight multiplier, explained later in this chapter.
- A description of the load pattern, if so desired, in the *Notes* column.

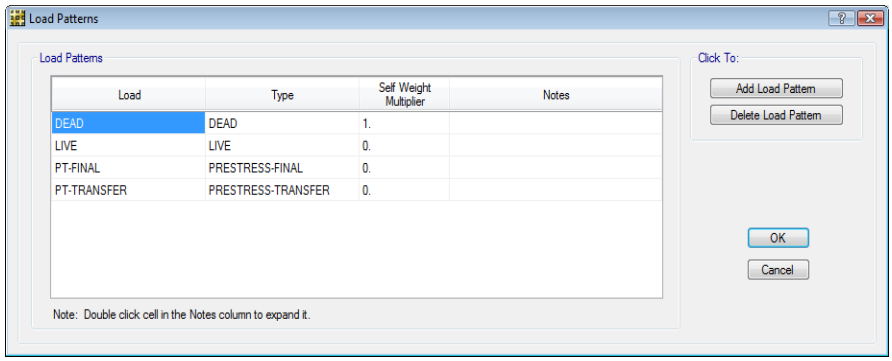


Figure 8-1 Load Patterns form

PT-Final and PT-Transfer will be available on the *Load Patterns* form when post-tensioning is included in the model.

To delete a load pattern, place the cursor anywhere on the line of data and hit the **Delete** key on the keyboard. An unlimited number of load patterns can be defined in SAFE. Consult the SAFE Help for more specific information about defining loads.

8.1.1 Self Weight Multiplier

The self weight of the structure is determined by multiplying the material weight per unit volume of each object that has structural properties, by the volume of the object. The unit weight is specified in the material properties, which are accessed using the **Define menu** > **Materials** command.

It is possible to specify that a portion of the self weight be applied to any load pattern. The self weight multiplier controls what portion of the self weight is included in a load pattern. A self weight multiplier of 1 indicates that the full self weight of the structure is included in the load pat-

tern. A self weight multiplier of 0.5 indicates that half of the self weight of the structure is included in the load pattern.

Normally a self weight multiplier of 1 should be specified in one load pattern only, usually the dead load pattern. All other static load patterns then have self weight multipliers of zero.

Important Note: If a self weight multiplier of 1 is defined for two different load patterns and they are combined in a load case or combination, the results for the load case or combination are based on an analysis where double the self weight of the structure is applied.

8.1.2 Auto Live Load Patterning

Auto live load patterning allows a live load pattern to be automatically applied to the slab in critical patterns to increase the negative and positive moments in slab panels and columns. With auto live load patterning, the associated loads applied to the load pattern are automatically divided up into smaller "single panel" load patterns based on the panels created by the layout of the grids. SAFE then uses the Range Add load combination feature to combine the results from each of these "single panel" loads automatically.

8.2 Assign Loads

Static loads can be assigned to point, beam, tendon, and slab objects. The objects must first be selected before a load can be assigned. Chapter 6 describes the options for selecting structural objects.

After the objects have 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 a slab object is selected, the **Assign menu > Load Data > Surface Loads** or **Assign menu > Load Data > Slab Temperature Loads** commands will be available, while a point selection will allow use of either the **Assign menu > Load Data > Point Loads** or **Load Data > Point Displacements** commands. If a tendon object is selected, the

Assign menu > Load Data > Tendon Loads or **Assign menu > Load Data > Tendon Losses** commands will be active. If only point objects (e.g., column supports) and beam objects are selected, the **Assign menu > Load Data > Surface Loads** command will not be active.

After clicking the **Assign menu** and the submenu command applicable to the type of object, a form will display. Table 8-1 identifies the forms generated when the various **Assign menu > Load Data** commands are used. Consult the SAFE Help for more specific information about assigning loads.

TABLE 8-1 Load Commands on the Assign Menu

Command	Name of Input Form
Load Data > Surface Loads	Surface Loads
Load Data > Slab Temperature Loads	Slab Temperature Loads
Load Data > Point Loads on Lines	Point Loads on Lines
Load Data > Distributed Loads on Lines	Distributed Loads on Lines
Load Data > Point Loads	Point Loads
Load Data > Point Displacements	Point Displacement Loads
Load Data > Tendon Loads	Tendon Load
Load Data > Tendon Losses	Tendon Loss Options

Although the forms vary depending on the command used, each form has a drop-down list that allows selection of the load pattern to which the loads are to be assigned. The forms also include other object assignment-specific input fields that enable refinement of the load assignment. Context-sensitive help is available for each form by pressing the **F1** function key on the keyboard when the form is displayed.

8.3 Define Mass Source

Before performing a modal (dynamic) analysis, the mass source for the model must be defined. Click the **Define menu > Mass Source** command to display the *Mass Source* form shown in Figure 8-2.

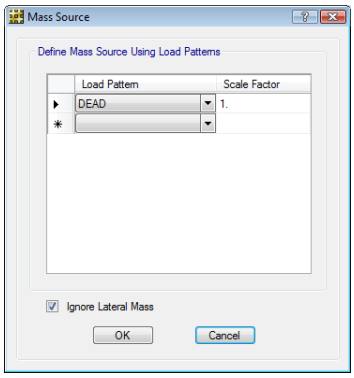


Figure 8-2 Mass Source form

Use the form to specify the load patterns that should be used for calculating the mass. The program will take the loads associated with each selected load pattern and will combine them using the input scale factors. These combined loads will then be divided by gravity to obtain the masses. More often than not, the only load pattern used will be the dead load pattern with self weight. Leave the *Ignore Lateral Mass* option checked to extract only vertical modes and frequencies; unchecking this option may result in instabilities if the model has not been properly restrained.

Chapter 9
Define Load Cases

A load case defines how loads are to be applied to the structure, and how the structural response is to be calculated. Analyses are classified in the broad sense as either linear, nonlinear, modal, or hyperstatic, depending on how the model responds to the loading. The results of linear analyses may be superposed, i.e., added together after analysis. The results of nonlinear analyses normally should not be superposed. Instead, all loads acting together on the structure should be combined directly within the nonlinear load case. Any number of named load cases may be defined.

9.1 Review/Create Load Cases

After all geometry and loading input has been specified for a model, review, modify, or add load cases using the **Define menu > Load Cases** command. The *Load Cases* form shown in Figure 9-1 will display. Highlight a load case (SAFE automatically generates a load case for each load pattern defined) and click the **Modify/Show Case** button to review or alter the load case definition. Click the **Delete Case** button to delete a highlighted load case.

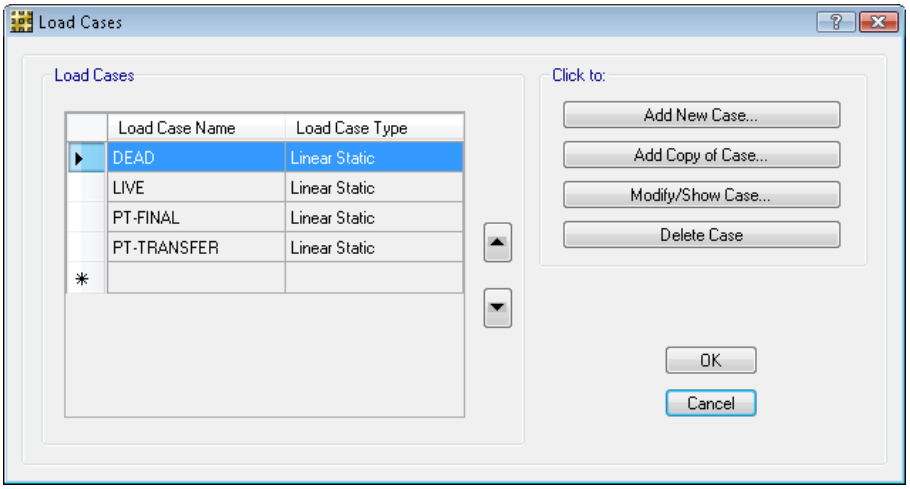


Figure 9-1 Load Cases form

To define a new load case, click the **Add New Case** button to display the *Load Case Data* form shown in Figure 9-2.

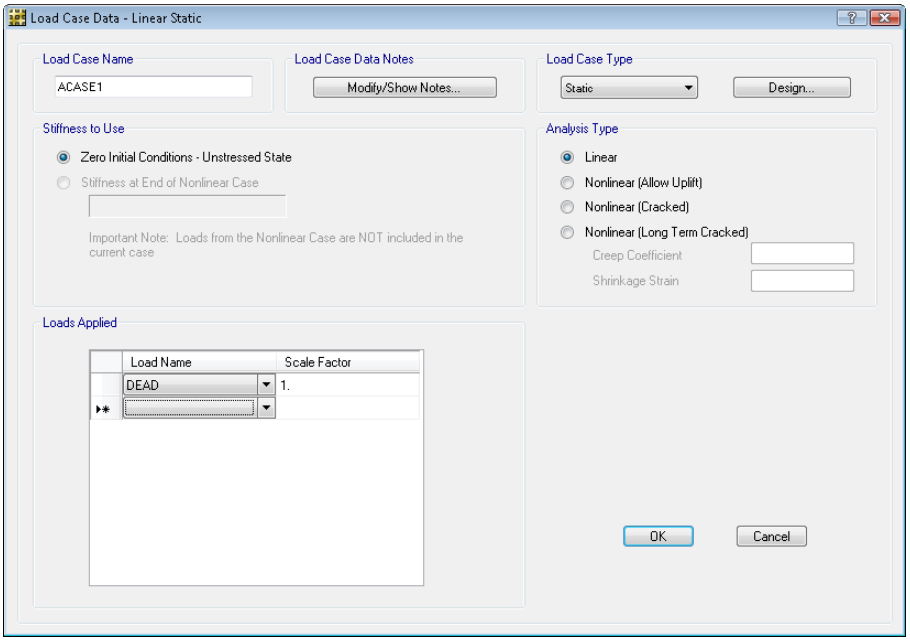


Figure 9-2 Load Case Data form

Use that form to specify the following information:

- The name of the load case. SAFE does not allow use of duplicate names.
- Notes about the specific load case.
- A load case type, which can be selected from the **Load Case Type** drop-down list. The default setting of static is the most common choice, but modal and hyperstatic also are available. A static case considers loads defined in a load pattern, a modal case carries out a frequency analysis, and a hyperstatic case applies loads to the structure in an unsupported state, from a chosen static case.
- The analysis type: linear or nonlinear (uplift, cracked, or long term cracked). If nonlinear (long term cracked) has been selected, creep coefficient and shrinkage strain values should be specified to effectively compute long term deflections.
- The initial stiffness to use. The load case may start with zero initial conditions or with a stiffness determined from a nonlinear load case.
- The loads to be applied, defined by the load name and a scale factor.

Click the **OK** button to accept the changes made on the form, or click **Cancel** to cancel the changes and return to the *Load Cases* form.

Chapter 10
View and Edit the Model Geometry

SAFE provides many ways to view a model, as well as a wide selection of tools to edit and revise the model data. This chapter provides an overview of some of the commands available to make editing and reviewing a model easy.

10.1 Changing Views

SAFE always starts with a Plan View as the default view. Clicking the **Options menu > Windows** command and selecting one of the subcommands allows the number of display windows to be changed. Table 10-1 identifies a few **View menu** commands that may be used to change what is displayed in the active window. For the commands listed, a window should be made active (click anywhere in the window) before using the commands.

TABLE 10-1 View Menu Commands in SAFE

Command/Button	Action	Immediate/ Form/ Toggle

TABLE 10-1 View Menu Commands in SAFE

Command/Button	Action	Immediate/ Form/ Toggle
Set Default 3D View 	The **Set Default 3D View** command or button sets the view to the default 3-D View.	Immediate
Set Plan View 	The **Set Plan View** command or button sets the view to the default Plan View.	Immediate
Set Elevation View 	The **Set Elevation View** command or button sets the view to a chosen elevation.	*Select Elevation* form
Set Display Options 	The **Set Display Options** command or button controls which objects, labels, and property identifiers are displayed, along with how they are displayed, e.g., show extrusions.	*Set Display Options* form
Rubber Band Zoom 	The **Rubber Band Zoom** command or button zooms in on the portion of the model that lies within a drawn rubber band window.	Immediate
Restore Full View 	The **Restore Full View** command or button restores the view such that the entire model fills the window.	Immediate
Previous Zoom 	The **Previous Zoom** command or button returns the view one step back to the previous zoom setting.	Immediate
Zoom In One Step 	The **Zoom In One Step** command or button zooms the view in one step. The default step is 10 percent, but this value can be changed using the **Options menu > Graphics Preferences** command.	Immediate
Zoom Out One Step 	The **Zoom Out One Step** command or button zooms the view out one step. The default step is 10 percent, but this value can be changed using the **Options menu > Graphics Preferences** command.	Immediate
Pan 	The **Pan** command or button allows the model to be moved around in the active window.	Immediate

10.2 Editing Tools

During the course of creating a model, it may be necessary to edit the model. Table 10-2 at the end of this chapter identifies the various edit

commands available in SAFE, some of which are familiar Windows commands.

In most cases, first select the point, beam, column, tendon, design strip, wall, slab rebar, or slab object and then click the appropriate menu item or button. In some cases, the action will be immediate (for example, the **Undo** or **Redo** commands). In other cases, a form will display that allows specification of how the object is to be edited. For example, the **Edit menu > Replicate** command accesses the *Replicate* form, which allows replication of point, beam, column, tendon, design strip, wall, slab rebar, or slab 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.

10.2.1 Interactive Database Editing

Interactive database editing is a very powerful editing capability within SAFE that allows the model or a selection of the model to be edited directly in SAFE or passed out to Microsoft Excel, edited, and read back into SAFE. Interactive database editing is accessed using the **Edit menu > Interactive Database Editing** command, which brings up the form shown in Figure 10-1.

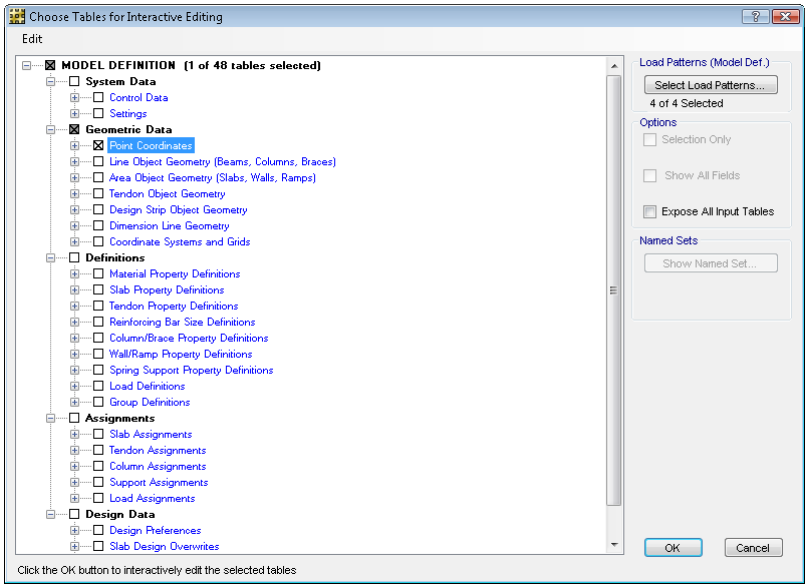
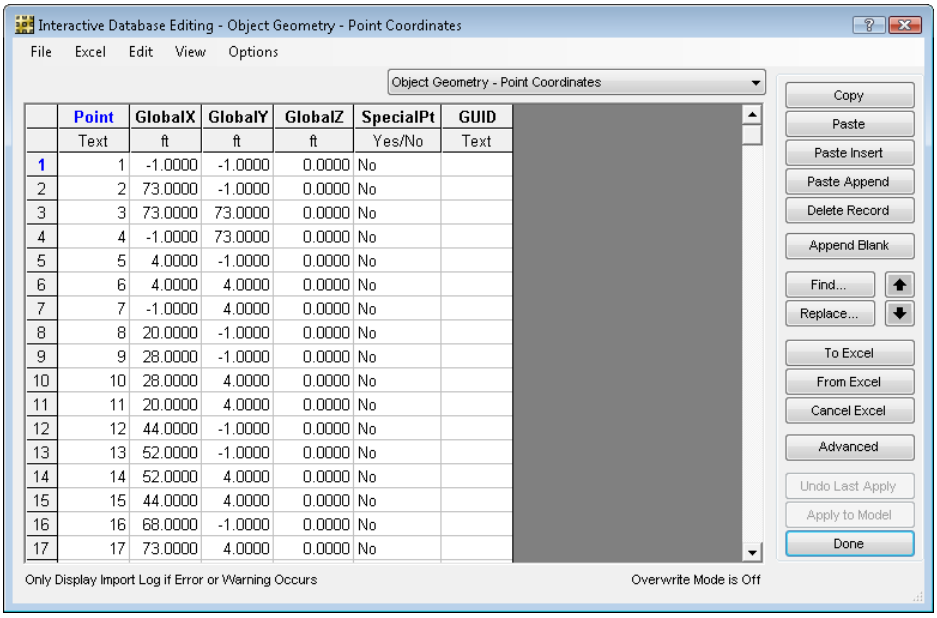


Figure 10-1 Choose Tables for Interactive Editing form

That form displays a tree of the available tables for editing. The listed tables will depend on the objects, definitions, and assignments present in the model. Click on a table to select it for editing. Multiple tables can be marked for editing.

Upon clicking the **OK** button, a form similar to that shown in Figure 10-2 will display with the chosen tables and their associated data.



	Point	GlobalX	GlobalY	GlobalZ	SpecialPt	GUID
	Text	ft	ft	ft	Yes/No	Text
1	1	-1.0000	-1.0000	0.0000	No	
2	2	73.0000	-1.0000	0.0000	No	
3	3	73.0000	73.0000	0.0000	No	
4	4	-1.0000	73.0000	0.0000	No	
5	5	4.0000	-1.0000	0.0000	No	
6	6	4.0000	4.0000	0.0000	No	
7	7	-1.0000	4.0000	0.0000	No	
8	8	20.0000	-1.0000	0.0000	No	
9	9	28.0000	-1.0000	0.0000	No	
10	10	28.0000	4.0000	0.0000	No	
11	11	20.0000	4.0000	0.0000	No	
12	12	44.0000	-1.0000	0.0000	No	
13	13	52.0000	-1.0000	0.0000	No	
14	14	52.0000	4.0000	0.0000	No	
15	15	44.0000	4.0000	0.0000	No	
16	16	68.0000	-1.0000	0.0000	No	
17	17	73.0000	4.0000	0.0000	No	

Figure 10-2 Interactive Database Editing form

The data can be edited directly in that form by clicking in a cell of the table and modifying the current value. It is possible to apply the changes to the model without closing the interactive editing form using the **Apply to Model** button. When finished, click the **Done** button to exit interactive editing.

Alternatively, the data can be taken out to Microsoft Excel using the **To Excel** button. This will open Microsoft Excel with a copy of the currently displayed table. Without closing the form in SAFE, changes should be made to the Excel file and when finished, click the **From Excel** button in the SAFE form. Note, it is not necessary to save the Excel file before reading it back into SAFE.

TABLE 10-2 Edit Menu Commands in SAFE

Command/Button	Action	Immediate/ Form/ Toggle
Undo and Redo 	The Undo command or button deletes the last performed action. The Redo command or button restores the last step.	Immediate

TABLE 10-2 Edit Menu Commands in SAFE

Command/Button	Action	Immediate/ Form/ Toggle
Cut, Copy, and Paste	Generally similar to the standard cut, copy, and paste Windows commands.	Immediate
Delete	The **Delete** command deletes the selected objects and all of the associated assignments (loads, properties, supports, and the like).	Immediate
Grid Data > Add/Modify Grids	Allows coordinate systems to be modified or deleted, or a new coordinate system to be defined.	*Coordinate Systems* form
Grid Data > Add Grid at Selected Points	Adds grid lines at the selected points in specified orientations.	*Add Grid Lines at Selected Points* form
Grid Data > Glue Points to Grids	Enables or disables gluing of points to grids such that the points lying on a grid line move if the grid line is moved.	Toggle
Grid Data > Lock On-Screen Grid System Edit	Enables or disables onscreen editing/movement of grid lines.	Toggle
Interactive Database Editing	Database editing can be done in the SAFE form or externally, in Microsoft Excel using this command. Specific items can be selected before using the **Interactive Database Editing** command, thereby controlling the list of items to be edited.	*Choose Tables for Interactive Editing* form (see Fig. 10-1 and Fig. 10-2)
Replicate	Duplicates the selected 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
Merge Points	Merges points within the tolerance distance of the selected point. A tolerance distance is specified on the *Merge Points* form.	*Merge Points* form
Align Points/Lines/Edges	Aligns selected points. Points can be aligned to an X-ordinate, Y-ordinate, or to the nearest selected line. When a point object is re-aligned, all objects attached to the point are reoriented or resized to account for the movement. The *Align Points/Lines/Edges* form can also trim line/tendon/strip objects that are too long, or extend objects that are too short. NOTE: This is a sophisticated and complex command, and it is strongly suggested that the behavior be verified before using it on a large model.	*Align Points/Lines/Edges* form

TABLE 10-2 Edit Menu Commands in SAFE

Command/Button	Action	Immediate/ Form/ Toggle
Move Points/Lines/Areas	Moves point objects in the model. When a point object is moved, all objects attached to the point are reoriented or resized to account for the movement.	*Move Points/Lines/Areas* form
Edit Lines > Divide Lines	Divides a line object into multiple line objects.	*Divide Lines* form
Edit Lines > Join Lines	Joins two or more colinear line objects with common end points and the same type of property into a single line object.	Immediate
Edit Lines > Convert Beams to Slab Areas	Converts a beam line object to a slab area object with plan dimensions equal to the width and length of the beam and a thickness equal to the depth as defined by the assigned beam property. Any loads assigned to the beam will remain as a line load.	Immediate
Edit Lines > Modify/Show Line Type	Allows modification of the line type to be either straight or curved.	*Line Object Type Options* form
Edit Areas > Divide Areas	Divides an area object into multiple area objects.	*Divide Selected Areas* form
Edit Areas > Merge Areas	Converts selected area objects that are touching or overlapping into a single area object. Only two objects can be merged at a time.	Immediate
Edit Areas > Expand/ Shrink Areas	Expands or shrinks an area object by a specified offset value.	*Expand/Shrink Areas* form
Edit Areas > Split Area Edges	Splits the selected area object edges into a specified number of segments.	*Split Area Edges* form
Edit Areas > Remove Points from Areas	Removes the selected point from the selected area object resulting in a change to the area object shape, i.e., an area object defined by five points becomes an area object defined with four points. Not recommended for use on area objects with less than four points.	Immediate
Edit Areas > Chamfer Slab Corners	Allows chamfering of the selected slab corner using either beveled or rounded edges.	*Chamfer Slab Corners* form
Edit Areas > Modify/Show Slab Edge Type	Allows modification of the slab edge to be either straight or curved.	*Slab Area Object Edge Type Options* form
Edit Areas > Modify/Show Wall Curve Type	Allows modification of the wall to be either straight or curved.	*Wall Area Object Curve Type Options* form

TABLE 10-2 Edit Menu Commands in SAFE

Command/Button	Action	Immediate/ Form/ Toggle
Add/Edit Tendons > Add Tendons in Strips	Where PT tendons are to be added to a particular strip, this command provides easy tendon assignment for the selected strips.	*Quick Tendon Layout* form
Add/Edit Tendons > Add Tendon to Beams	Where PT tendons are to be added to a particular beam, this command provides easy tendon assignment for the selected beams.	Immediate
Add/Edit Tendons > Edit Plan Layout (Horizontal)	Allows modification of the tendon to be either straight or curved in the horizontal plane.	*Tendon Object Type Options* form
Add/Edit Tendons > Edit Vertical Profile	Allows modification of the tendons vertical profile on a span by span basis.	*Tendon Vertical Profile* form
Add/Edit Tendons > Reset Supports and Spans to Default	Resets the tendon support points and span definitions to the program default if they have been modified using the vertical profile form.	Immediate
Add/Edit Tendons > Copy Vertical Profile	Copies the vertical profile of a tendon. This command is only available when a single tendon is selected.	Immediate
Add/Edit Tendons > Paste Vertical Profile	Pastes the vertical profile copied from a tendon, to any number of selected tendons.	Immediate
Add/Edit Slab Rebar > Add Slab Rebar	Allows for placement of user-defined reinforcement – similar in functionality to the Draw menu > Draw Slab Rebar command.	*Slab Rebar Object Data* form
Add/Edit Slab Rebar > Edit Slab Rebar	Allows modification of the selected slab rebar object.	*Slab Rebar Object Data* form
Add/Edit Design Strips > Add Design Strips	Provides a quick method to define design strips along grids or based on structural supports.	*Add Design Strips* form
Add/Edit Design Strips > Edit Strip Widths	Allows the strip widths to be modified, either with user defined widths or automatic widening of strips by SAFE.	*Edit Strip Widths* form

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

Chapter 11
Analysis and Design

After all geometry and loads have been input into the model, it is ready for analysis and design. This chapter provides an overview of the analysis and design process in SAFE.

11.1 Set the Mesh Options

Before running the analysis, set the meshing options using the **Run menu > Automatic Slab Mesh Options** command. The *Automatic Slab Mesh Options* form shown in Figure 11-1 will display.

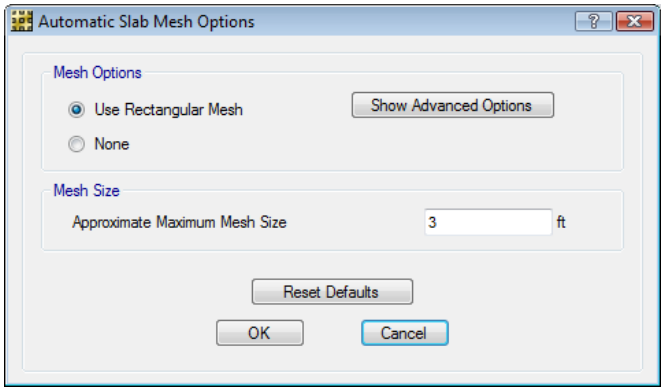


Figure 11-1 Automatic Slab Mesh Options form

Use the *Automatic Slab Mesh Options* form to specify the type and maximum size of mesh to use. The rectangular mesh provides three methods. It can either create a mesh that is parallel and perpendicular to the longest edge, the grid system, or the local axes of the area objects. During analysis, SAFE automatically meshes the model based on these parameters to create a finite element model. Generation of the mesh is influenced by the locations of slab objects, openings, beams and line springs, columns, walls, and point restraints and springs. To preview the element mesh before running the analysis and design, use the **View menu > Set Display Options** command. When the *Set Display Options* form displays, check the **Show Mesh** check box under the *Options* category.

11.2 Design Process

Design can be performed automatically as part of the analysis run if the **Run menu > Run Analysis & Design** command is used. SAFE designs both concrete slabs and beams, and checks punching shear. The type of design depends on the type of members used in the model (e.g., a beam design will be performed only if beam objects are included in the model). Specific material properties and design data are input using the **Define menu** commands, and design code preferences are selected using the **Design menu > Design Preferences** command.

The **Design menu > Design Preferences** command allows the desired design code to be selected, as well as defining the slab and beam reinforcement covers, the post-tensioned allowable stress limits, and preferred reinforcement bar sizes.

Table 11-1 summarizes the commands on the **Design menu** used in the design process. User-defined load combinations can be specified in addition to the automatic program generated load combinations that are specific to the selected design code. Overwrites to strips, slabs, beams, and punching shear design also can be specified to further customize the design process on an object-by-object basis.

TABLE 11-1 Design menu commands

Command	Action	Form
Design Preferences	Allows input and control of all design parameters for mild reinforcement and post-tensioning.	*Design Preferences*
Design Combos	Allows review of the load combinations. Allows specific combinations to be chosen for use in the design for strength and the various service conditions.	*Design Load Combinations Selection*
Slab Design Overwrites > Strip Based	Allows for the review/editing (overwriting) of default (as determined by the program) strip based slab design parameters.	*Strip Based Slab Design Overwrites*
Slab Design Overwrites > Finite Element Based	Allows for the review/editing (overwriting) of default (as determined by the program) finite element based slab design parameters.	*Finite Element Based Slab Design Overwrites*
Beam Design Overwrites	Allows for the review/editing (overwriting) of default (as determined by the program) beam design parameters.	*Beam Design Overwrites*
Punching Check Overwrites	Allows for the review/editing (overwriting) of default (as determined by the program) punching shear design parameters.	*Punching Shear Design Overwrites*

11.3 Run Analysis and Design

To start the analysis and design, use the **Run menu > Run Analysis & Design** command or click the **Run Analysis & Design** button. In the lower left- hand corner of the screen, the status bar will display mes-

sages indicating the status of the analysis. When the entire analysis process is complete, a deformed shape view of the model is automatically displayed, and the model is locked. The model is locked when the **Lock/Unlock Model** button shows the padlock in the closed, or locked position. Locking the model prevents changes to the model that would invalidate the analysis results.

Consult the SAFE Help topics and the *CSI Analysis Reference* manual for more information about analyzing models.

11.4 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 model also can be locked at any time to prevent changes, or unlock it to permit changes using the **Options menu > Lock/Unlock Model** command or the **Lock/Unlock Model** button. SAFE will display a warning message that analysis results will be deleted if a model is unlocked following an analysis.

Chapter 12
Reinforcement Detailing

After the analysis and design are complete, the reinforcement detailing can be run. This chapter provides an overview of the detailing process available for generating detailed drawings based on the analysis and design results.

Note that the analysis and design must be complete before running the detailing. Re-running the detailing does not execute a re-analysis. If changes have been made to the model that will affect the design, it is necessary to re-run the analysis and design before re-running the detailing.

12.1 Detailing Process

SAFE detailing generates two basic forms of output:

- Drawing component views of detailed objects, such as elevations, plans, sections, and tables
- Drawing sheets containing an assemblage of component views

The reinforcement detailing is based on user-defined preferences. Default preferences are defined as a starting point, but should be reviewed and modified as necessary before running the detailing.

Detailing generates a default set of drawing component views and drawing sheets that can be modified, which automatically updates all related views and drawing sheets. The generated views can be edited to improve text readability and to add additional annotations, allowing complete control over the information contained on the drawing sheets, including drawing size, scale, layout, title block, and component views. Drawings can be printed directly from SAFE or exported for further manipulation in CAD applications.

The editing and modifications to the objects, views, and drawing sheets are retained to the extent practicable during the synchronization process even if the model is changed and analysis and design is repeated. This automatic synchronization also helps to apply modified options and preferences to existing detailing output.

12.2 Preferences

The detailing preferences specify various parameters such as the curtailment rules, dimensioning units, preferred bar sizes, min and max bar lengths, drawing sheet size and scale, line styles, and many other customizable parameters.

The detailing units, as well as other preferences, are set using the **Detailing menu > Detailing Preferences** command. This brings up the form shown in Figure 12-1. This form also allows the desired rebar set to be chosen, the bar mark style to be set, as well as various other units to be defined.

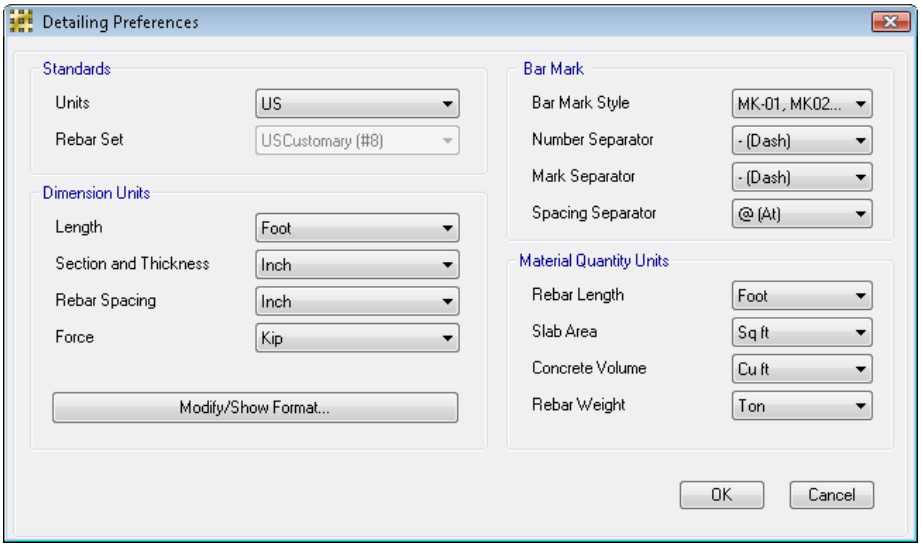


Figure 12-1 Codes, Dimension Units and Formatting form

Preferences for defining the drawing sheet size, scale, margins, and title block are defined using the **Detailing menu > Drawing Sheet Setup** command.

The slab, beam, mat, and footing object detailing preferences can be set using the **Detailing menu > Slab/Mat Reinforcing Preferences** and **Beam Reinforcing Preferences** commands. Each of those commands displays a form with various detailing preferences, such as min and max bar spacing, smallest and largest bar sizes, preferred bar sizes, and other preferences dependent on the object type. Figure 12-2 shows the rebar selection tab of the *Slab/Mat Detailing Preferences* form.

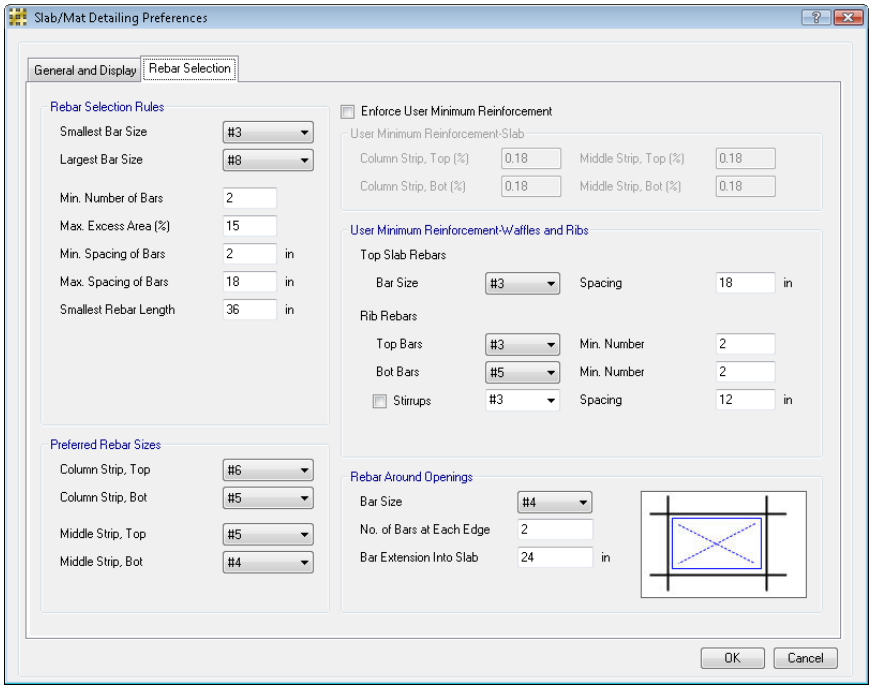


Figure 12-2 Slab/Mat Detailing Preferences form

Each detailed object is comprised of lines, text, arrows, and colored fill. The line types, line colors, arrow styles, font type, font size, font color, and so forth, can all be customized for each object type. The default settings are typically adequate, but it may be desirable to modify them. The **Detailing menu > Drawing Format Properties** command displays the *Drawing Format Properties* form. This form allows for setting the styles and colors of all the various types of formatting items used in the detailing.

The detailing preferences are saved with the model and can therefore be reused when the detailing is rerun or when a new model is initialized from an existing model that already has the detailing preferences set.

12.3 Run Detailing

To start the detailing, use the **Run menu** > **Run Detailing** command or click the **Run Detailing** button. The first time the detailing is run for a model, a default set of drawings is created. Subsequent runs of the detailing will provide the option, using the form shown in Figure 12-3, to generate a new set of drawings or synchronize the detailing and update the existing set of drawings. This allows for retention of modifications made to the drawing component views and drawing sheets.

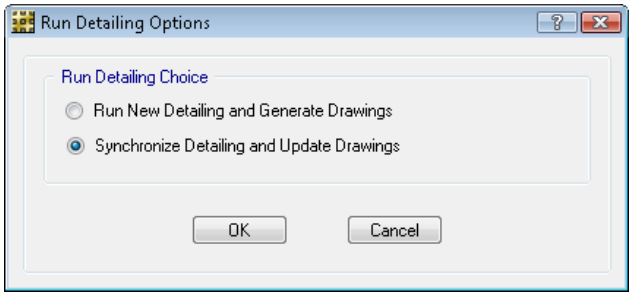


Figure 12-3 Run Detailing Options form

After a detailing run, a tree of the Drawing Component Views and Drawing Sheets is displayed in the Detailing tab of the *Model Explorer*. Expand the tree and double click one of the Drawing Component Views or Drawing Sheets to display it in the active window. SAFE places the Drawing Component Views on the Drawing Sheets.

12.4 Edit Component Views

Modifications can be made to the drawing component views to customize the detailed reinforcement, add or modify section cuts, add or modify the view text, and modify the view properties. These modifications affect the corresponding component views as well as the view copies placed on the drawing sheets after the modification is made. Component views already located on drawing sheets before making modifications are not updated.

The detailed reinforcement can be modified using the **Detailing menu > Edit Reinforcement** commands. Depending on the objects present in the model, separate commands will be available for slabs/mats and beams. The reinforcement editing allows the rebar sizes, spacing, lengths, and curtailment location to be modified. Note, it is strongly recommended that these changes be made with caution to ensure that the reinforcement specified adheres to code requirements, as specified by default.

Defined section cuts generate additional drawing component views for the main structural objects. Default section cuts are generated when the detailing is first run. These can be modified or deleted, or additional section cuts can be defined, using the **Detailing menu > Add/Modify Sections** commands. Depending on the objects present in the model, separate commands will be available for slabs/mats and beams.

The text on a drawing component view can be fully customized, including editing of the program generated text, as well as adding additional text, notes, and dimension lines. To modify the text on a specific drawing component view, right click on the component view in the display or on the detailing tree of the *Model Explorer* and choose the **Edit View Text** command. A form is opened showing the view and contains a wide selection of tools for zooming the view and making the necessary modifications.

Each drawing component view also has its own properties that specify its name, scale, as well as the text and line sizes. These preferences can be modified by right clicking the component view in either the display or on the detailing tree of the *Model Explorer* and choosing the **Edit View Properties** command.

12.5 Create and Manage Drawing Sheets

The drawing sheets are a collection of scaled drawing component views, ready for direct printing or export to other file formats. SAFE automatically creates a set of default drawings with appropriate views. Several tools are available to create new drawing sheets and to modify and manage exiting sheets.

The list of drawing sheets can be modified using the **Detailing menu** > **Edit Drawing Sheets List** command. Additional drawing sheets also may be added by right clicking on the *Drawing Sheets* node of the detailing tree in the *Model Explorer* window, and choosing the **Add Blank Drawing Sheet** command. Individual drawings can be deleted by right clicking on them in the *Model Explorer* window and choosing the **Delete Drawing Sheet** command.

The drawing component views can be rearranged on a drawing sheet by simply clicking on them and dragging them to a new location. Snap features aid in locating the component views on the drawing sheet. Alternatively, right clicking on a drawing and using the **Auto Arrange Views** command will automatically arrange the views on the drawing sheet, and if necessary, generate additional sheets if all of the views do not fit on a sheet. Drawing component views can be removed from a drawing sheet by right clicking on them either on the drawing sheet display or in the *Model Explorer* window and choosing the **Delete View** command.

Drawing component views can be quickly and easily added to drawing sheets using drag and drop methods. After a drawing sheet is displayed, component views can be dragged and dropped from the detailing tree in the *Model Explorer* window. Alternatively, multiple component views can be added by right clicking on the drawing and using the various **Add Views** commands for the different object types.

Each drawing component view on a drawing sheet can have its properties and text edited using the same methods described in the previous section. It should be noted that modifications made to the component views on drawing sheets are local to that particular component view and do not affect the global drawing component views described in the previous section. Changes to the global component views will not have an effect on drawing component views already placed on drawings.

Chapter 13
Display Results

After the model has been analyzed, designed, and detailed, results may be displayed graphically in the view windows or in forms, as well as in tabular format.

13.1 Graphical Display of Analysis Results

To display analysis results, click the **Display menu** and select the type of display desired, use the corresponding toolbar buttons, or switch to the Display tab in the *Model Explorer* window and double click on the desired named display. Table 13-1 identifies the display options for analysis results in SAFE, as well as the form that will display. The input on the form controls what and how analysis results are displayed.

TABLE 13-1 Display menu – Analysis commands

Command / Button	Action	Form
Show Undeformed Shape 	The Show Undeformed Shape command or button plots the undeformed shape in the display window.	No form

TABLE 13-1 Display menu – Analysis commands

Command / Button	Action	Form
Show Loads 	The **Show Loads** command or button displays loads assigned to point, beam, tendon, and slab objects.	*Show Loads*
Show Deformed Shape 	The **Show Deformed Shape** command or button plots a deformed shape in the display window based on the chosen load case, combination, or modal case. This plot can be animated in 3-D views.	*Deformed Shape*
Show Reaction Forces 	The **Show Reaction Forces** command or button displays reaction forces (or soil pressures) in the display window based on the chosen load case or combination.	*Reactions*
Show Beam Forces/Stresses 	The **Show Beam Forces/Stresses** command or button displays beam forces or stresses in the display window based on the chosen load case or combination.	*Beam* Forces/Stresses
Show Slab Forces/Stresses 	The **Show Slab Forces/Stresses** command or button displays slab forces or stresses in the display window based on the chosen load case or combination.	*Slab Forces/Stresses*
Show Strip Forces 	The **Show Strip Forces** command or button displays strip forces in the strip layers based on the chosen load case or combination.	*Strip Forces*

While a form is displayed, press the F1 function key on the keyboard to access context-sensitive help on the parameters in the form. Figure 13-1 shows sample displacement results displayed on a 3-D view of a model.

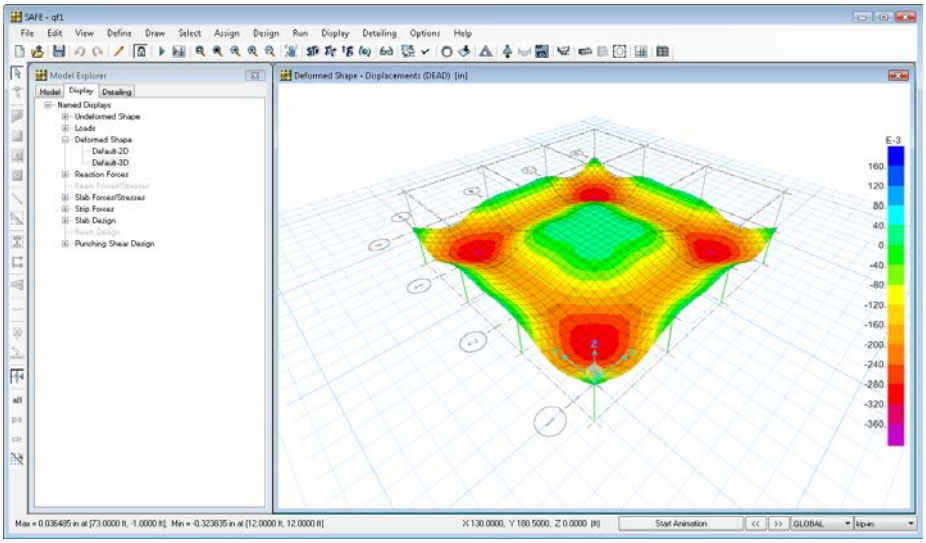


Figure 13-1 Deformed shape results display

13.2 Graphical Display of Design Results

To display design results, click the **Display menu** and select the type of display desired. Table 13-2 identifies the design display options available in SAFE, as well as the form that will display. The input on the form controls what and how design results are displayed.

TABLE 13-2 Display menu - Design commands

Command / Button	Action	Form
Show Slab Design	The Show Slab Design command or button plots the required slab reinforcement or post-tensioning stresses, as calculated by SAFE using design strips or finite element design, in the display window.	*Slab Design*
Show Beam Design	The Show Beam Design command or button plots the required beam reinforcement or post-tensioning stresses, as calculated by SAFE, in the display window.	*Beam Design*

TABLE 13-2 Display menu - Design commands

Command / Button	Action	Form
Show Punching Shear Design 	The **Show Punching Shear Design** command or button plots the punching shear ratios in the display window.	Immediate - No form

While a form is displayed, press the F1 function key on the keyboard to access context-sensitive help on the parameters in the form. Figure 13-2 shows sample strip required reinforcement results displayed on a plan view of a model.

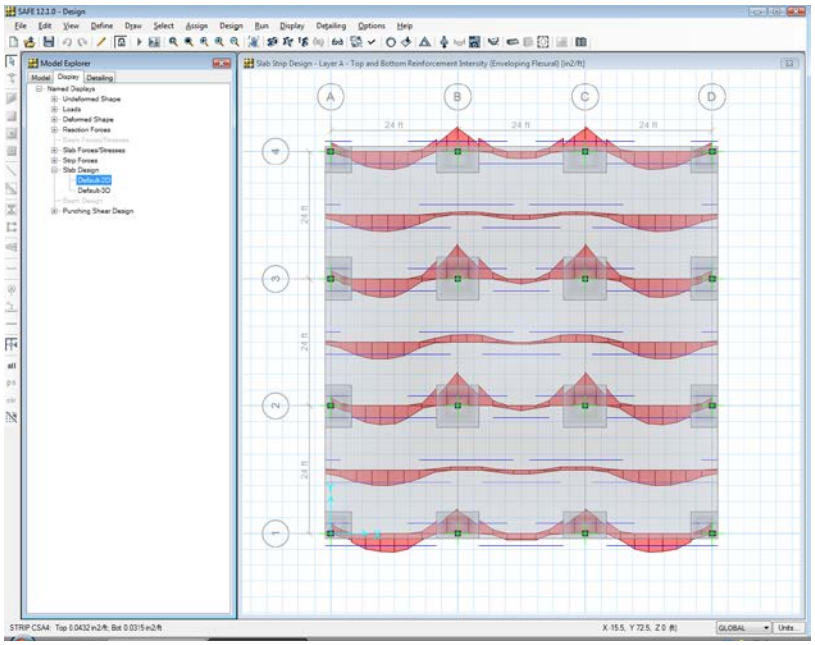


Figure 13-2 Strip reinforcement results display

Whenever design results (except for the finite element based slab design) are displayed, right clicking on the associated object will bring up a form with additional details about the design. That form allows different design combinations to be set, as well as toggling what items are displayed. Figure 13-3 shows the **Design Details** form for a sample beam. That

form allows the displayed design details to be sent to the printer or saved to an RTF (rich text format) file compatible with Microsoft Word.

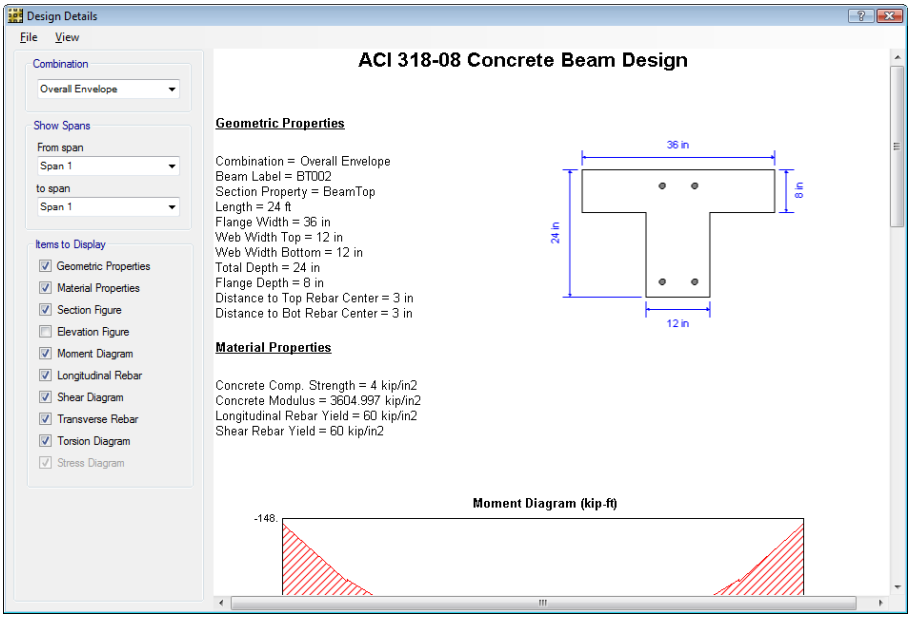


Figure 13-3 Design Details form

For beam and strip results, the **Design Details** form presents summary results on a span-by-span basis. The numerical values below the diagrams correspond to the maximums in the first quarter, middle half, and last quarter of each span.

13.3 Tabular Display of Results

Both the analysis and design results also can be displayed in tabular format in SAFE. The **Display menu > Show Tables** command or button presents the *Choose Tables for Display* form, shown in Figure 13-4. The tree structure can be expanded to reveal various results tables and allows the selection of one or more tables of analysis and design results, which subsequently are displayed in a table.

The displayed tables can be sorted and filtered based on user-defined criteria. While a table is displayed, right clicking on a column heading will present a description form, with a brief explanation as to what the column of data is.

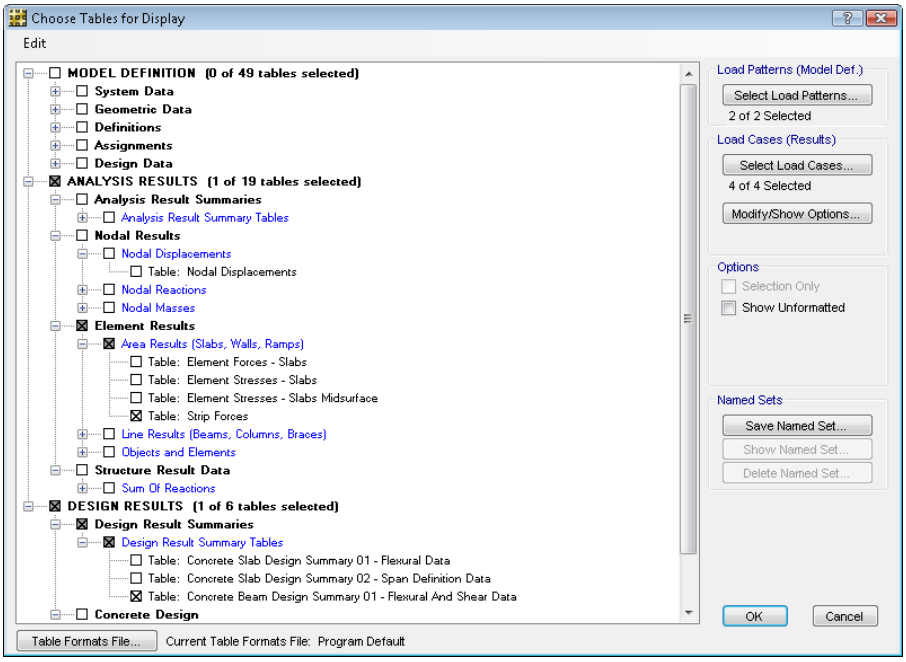


Figure 13-4 Choose Tables for Display form

It also is possible to save a Named Set of tables when choosing tables to be displayed. This allows for quick and easy retrieval of a set group of tables the next time that they are desired, without requiring each table in the tree structure to be individually reselected.

13.4 Detailing Results

The reinforcement detailing results are presented in the form of detailed drawing component views that can be accessed from the Detailing tab of the *Model Explorer* window or from the **Detailing menu > Show Detailing** command or button. Those views consist of framing plans, reinforcement plans, section cuts, reinforcement schedules, and rein-

forcement tables. After running the detailing, double clicking on one of the component views in the detailing tree will show it in the display window. Figure 13-5 shows a sample of a slab reinforcement drawing component view on a drawing sheet. Chapter 12 provides additional information on how to modify and customize these views.

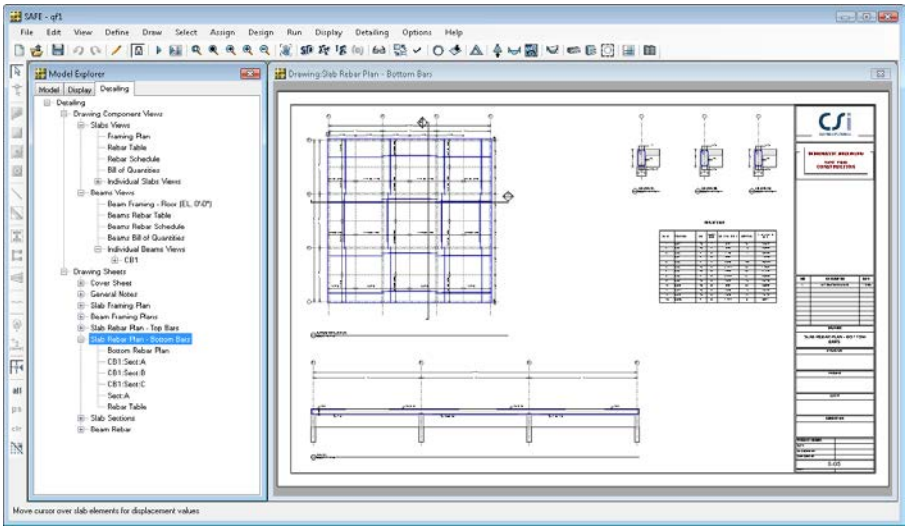


Figure 13-5 Detailed reinforcement component view on drawing sheet

Chapter 14
Output Results and Reports

SAFE provides a variety of methods for outputting analysis, design, and detailing results for further post-processing, presentations, or project submittals. This chapter describes the most commonly used methods for extracting results from a SAFE model.

14.1 Summary Report

A summary report is available at the click of a button using the **File menu > Create Report** command. This summary report is automatically created by SAFE, is compatible with Microsoft Word, and can contain the following items, depending on the make-up of the model:

- Title page containing project information defined from the **File menu > Modify/Show Project Information** command.
- Hyperlinked table of contents listing sections, figures, and tables, along with their associated page number.
- Model definition data in tabular format.
- Analysis results in tabular format.

- Summary design results in tabular format.
- Design detail summary sheets

The report also can be generated as HTML or plain text. The contents of the report can be filtered by specifying a predefined group or selecting specific load patterns and load cases. All of those options can be set using the **File menu > Report Setup** command, which presents the form shown in Figure 14-1.

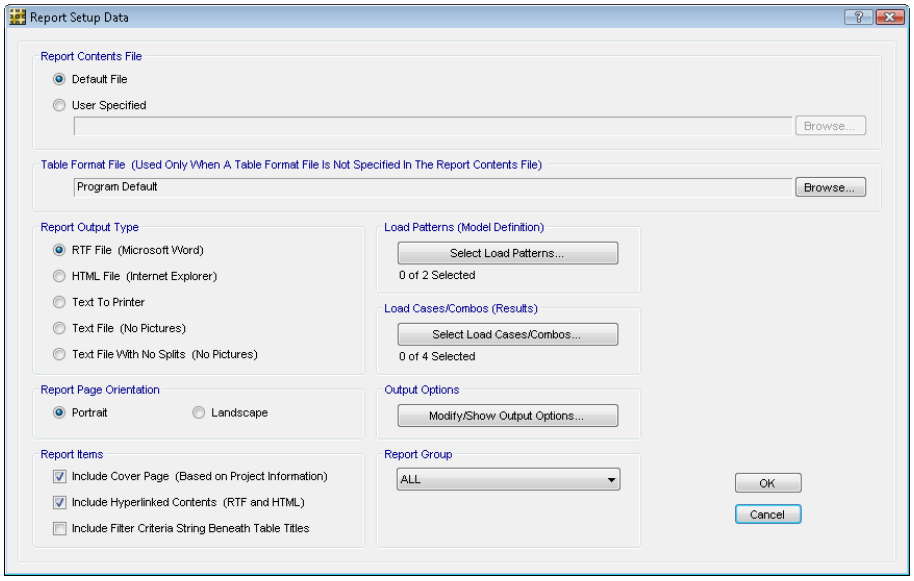


Figure 14-1 Report Setup Data form

The contents of the report are defined in the *SafeReportContents.xml* XML file that is installed in the SAFE installation directory. A user-defined contents file can be created by simply taking a copy of the default XML file provided with SAFE, renaming it, and editing the contents. The XML can be edited in any text browser or XML editor. For more information on editing or creating an XML report contents file, please refer to the documentation describing the report contents file, available from the **Help menu > Documentation** command. After a new contents file has been generated, the **File menu > Report Setup** command can specify that the revised file be used instead of the default file.

In addition to the summary report that is automatically generated by SAFE, there is also an advanced report writer tool that can be accessed from the **File menu > Advanced Report Writer** command. That tool allows custom reports, including text, pictures, tables, and more, to be defined, saved, and exported to RTF, HTML, or text format.

14.2 Print Graphics and Tables

The graphic displays and tables described in the previous chapter also can be printed directly to a printer or captured to various file formats.

The display in the currently active window can be printed directly to the printer using the **File menu > Print Graphics** command. This will provide a print preview form that allows for adding text or graphical annotations before going to the printer. Graphic displays also may be captured to a file using the **File menu > Capture Picture** commands. There are different options for defining the region to be captured and the image file type.

The **File menu > Print Tables** command presents a similar form to the one from the **Display menu > Show Tables** command. However, the File menu command provides the capability to print a selection of tables directly to the printer or send the tables to an RTF (Word compatible), HTML, or text file. Again, the data can be filtered based on a selection in the model, specified filter criteria, or selected load patterns and load cases.

14.3 Export Results

Analysis and design results also can be exported from SAFE for further post-processing or use in other applications. The **File menu > Export Model** command has three main options for exporting results.

A display showing results can be exported to a DXF/DWG file compatible with CAD applications. The exported data will depend on the current display. The tabular data described in the previous section can be export-

ed to either Microsoft Excel (*.xls) or Microsoft Access (*.mdb). The desired tables are chosen and filtered in the same way as previously described.

14.4 Print and Export Drawings

The drawing sheets generated during the detailing can be printed directly from SAFE. The sheets can be sent to a normal printer or a full size plotter. The **File menu > Print Drawings** command brings up a printing form that allows typical print settings to be defined, as well as the desired drawing sheets to be selected.

If further manipulation of the drawings is desired, they can be exported easily to DWG or DXF format for import into CAD applications, using the **File menu > Export Drawings** command. Similar to the print command, the export form, shown in Figure 14-2, allows the desired drawings to be selected and the output format to be specified.

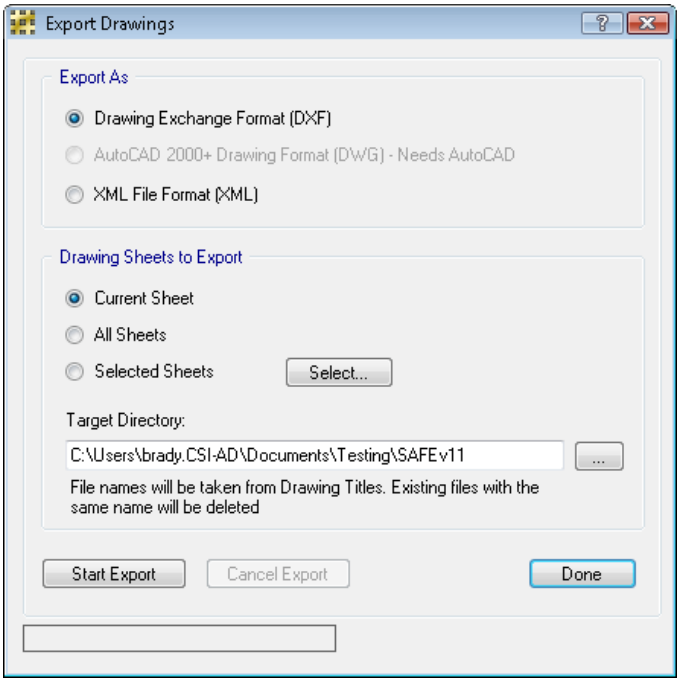


Figure 14-2 Export Drawings form

As an alternative to using the **File menu** commands, it is also possible to print or export drawing component views or drawing sheets by right clicking on them in the display window or the *Model Explorer* window. Right clicking will bring up a context menu that has both print and export commands.