SAFE™

Integrated Analysis and Design of Slab Systems

Tutorial



Computers and Structures, Inc.
Berkeley, California, USA

Version 8.0.0
August 2004

Copyright

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

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

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

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

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

DISCLAIMER

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

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



Contents

Tutorial

An Example Model

The Project		1
Step 1	Begin a New Model	2
	Set the Units	2
	Define the Grid	3
	Save the Model	6
Step 2	Define Properties	6
	Define Slab, Drop and Column Properties	6
	Define Beam Properties	9
Step 3	Define Supports	10
	Define Column Supports	10
	Define Wall Supports	11
Step 4	Define Static Load Cases	13

Step 5	Draw Objects	15
	Draw Slabs (Area Objects)	15
	Draw Column Supports (Point Objects)	17
	Draw Wall Supports (Line Objects)	18
	Draw Drop Panels (Area Objects)	20
	Draw Columns (Area Objects)	22
	Draw Beams (Line Objects)	23
	Draw Openings (Area Objects)	24
Step 6	Set Object Options	24
Step 7	Assign Loads	26
Step 8	Run the Analysis	28
Step 9	Graphically Review the Analysis Results	28
Step 10	Design	30

An Example Model

The intent of this tutorial is to give you hands-on experience using SAFE, likely the quickest way to become familiar with a program. Fundamentals of the model creation process are identified and various model construction techniques are introduced. The step-by-step instructions demonstrate how easy SAFE is to use. As you complete the tutorial, you will build the model shown in Figure 1.

The Project

The tutorial project is an irregularly shaped suspended concrete slab, with overall dimensions of 113 feet by 120 feet. A large opening exists in the interior for stair access.

The 10-inch-thick slab is supported by 12-inch-thick walls, 16-inch-thick drop panels on columns, and 12-inch by 24-inch beams on two perimeter sides. Columns are typically 18 inches square, and the story height below the slab is 12 feet. The model will be analyzed for a uniform dead load of 30 pounds per square foot (psf) plus the self-weight of the structure, and a live load of 50 psf.

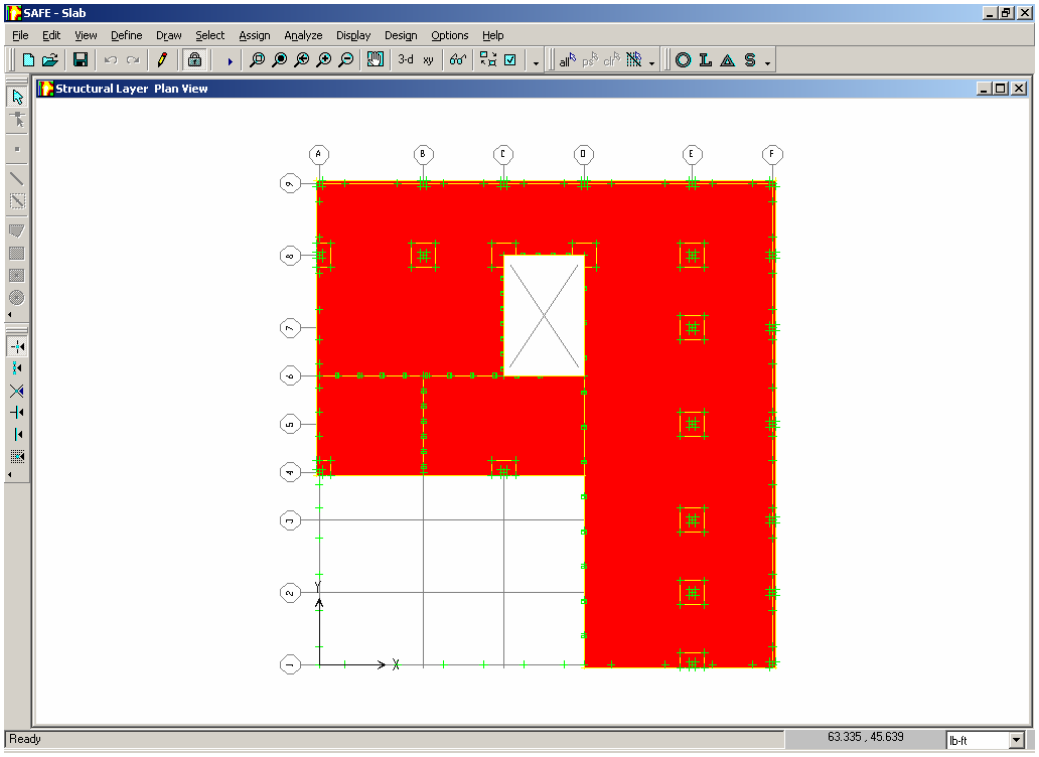


Figure 1
The Project Model

Step 1 Begin a New Model

In this Step, the dimensions and basic grid will be defined, which will serve as a guide for developing the model. This model will be built without using the automated template tools provided in SAFE so that you may see how to construct a model from scratch. However, as a general rule, we highly recommend that you start your models using templates whenever possible because they provide a quick, easy way of generating a model. Consult the SAFE Help topics for information about templates.

Set the Units

If the units shown in the drop-down list in the lower right-hand corner of the SAFE window are not lb-ft, scroll through the drop-down list to set the units to *lb-ft*.

Define the Grid

A. Click the **File menu > New Model** command or the **New Model** button  to access the Grid Definition form shown in Figure 2. This form is used to specify the number of grids and spacing in each direction. It is important that the grid be defined to represent accurately the geometry of the structure, so it is advisable to spend time carefully planning the number and spacing of the grid lines.

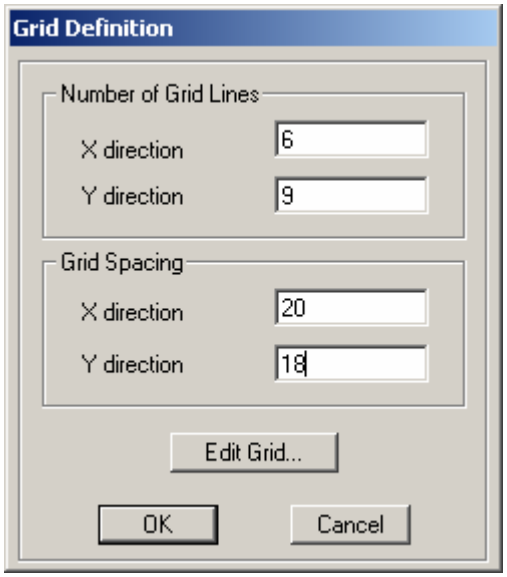


Figure 2
Grid Definition form

B. As shown in Figure 2, set the "**Number of Grid Lines**" in the X Direction to **6** and in the Y Direction to **9**. Set the "**Grid Spacing**" in the X Direction to **20** feet and in the Y Direction to **18** feet.

C. Select the **Edit Grid** button to display the form shown in Figure 3.

The Define Grid form is used to modify and edit the grid definitions in the x and y directions, as well as set the top of slab datum. It also allows the user to set the display options associated with the grids.

1. In the "**Display**" area, select the *X Grid* option.
2. In the "**Display Grid as"** area, select the *Ordinates* option.

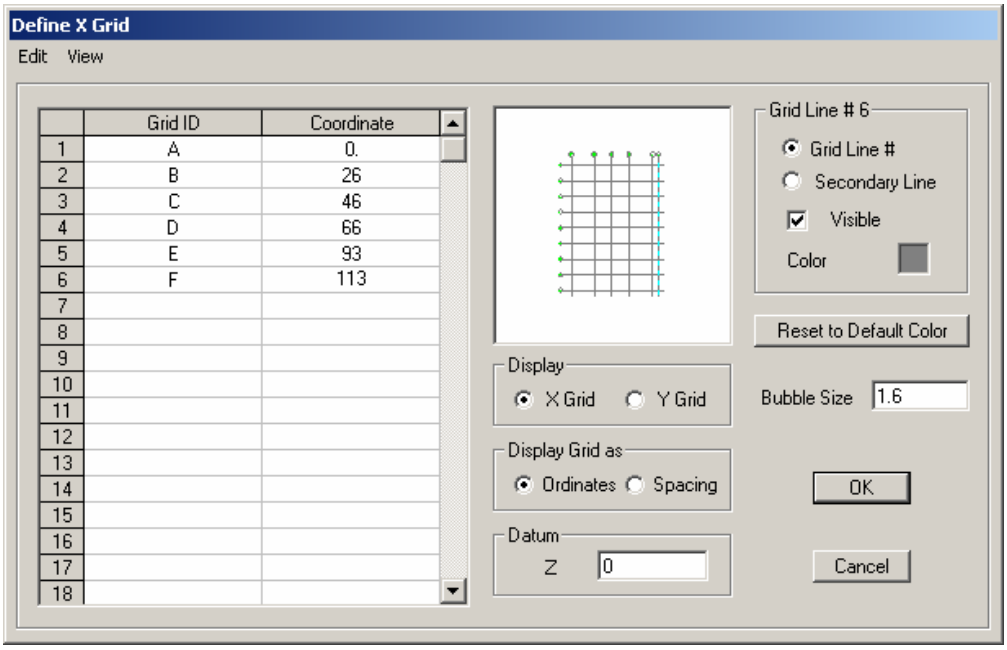


Figure 3
Define Grid form

3. In the grid table, change the X coordinates as follows:

Grid ID	Change X Coordinate to
B	**26**
C	**46**
D	**66**
E	**93**
F	**113**

4. In the **Display** area, select the *Y Grid* option.

5. In the grid table, change the Y coordinates as follows:

Grid ID	Change Y Coordinate to
4	**48**
5	**60**
6	**72**
7	**84**
8	**102**
9	**120**

6. Click the **OK** button to accept your changes.

When you click the **OK** button, by default, the grid appears on screen in the main SAFE window with two view windows tiled vertically, a Structural Layer Plan View on the left and a Structural Layer 3-D View on the right. The number of view windows can be changed using the **Options menu > Windows** command.

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

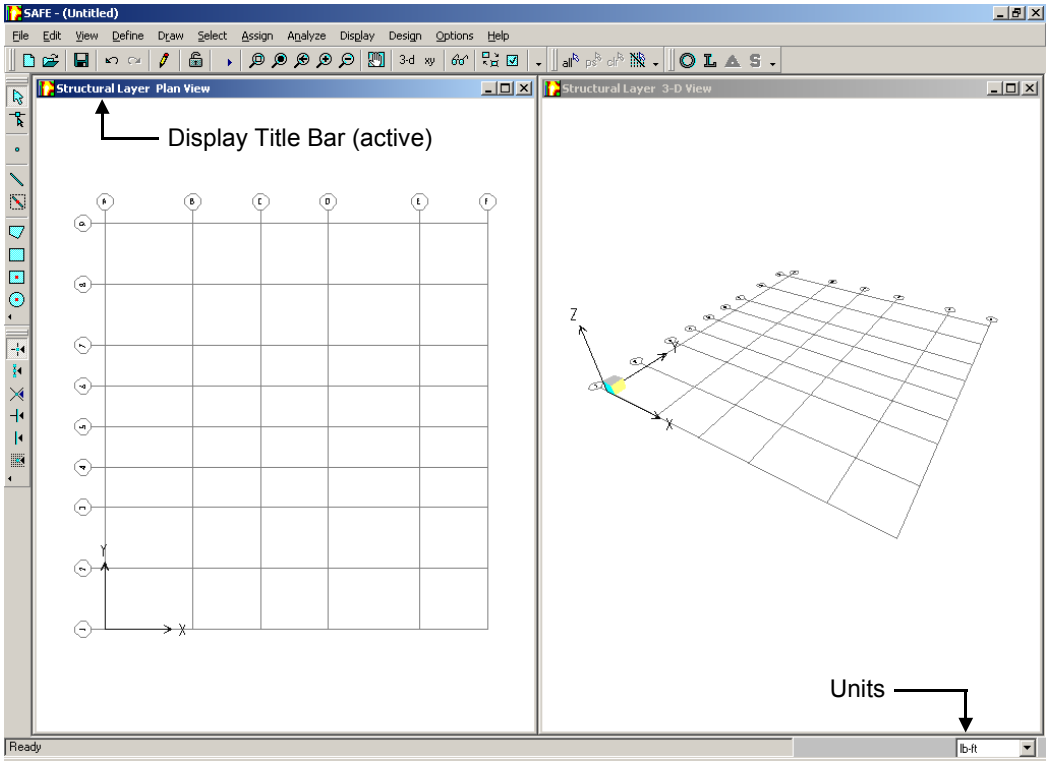


Figure 4
The SAFE main window

Save the Model

Save your model often! Click the **File menu > Save** command, or the **Save** button . Specify the directory in which you want to save the model. For this tutorial, specify the file name as *Slab*.

Typically you will save the model with the same name. However to record your work at various stages of development or as a backup, the **File menu > Save As** command can be used to save the file using another name.

Step 2 Define Properties

In this Step, properties for the slab (area object) and beams (line objects) are defined.

Define Slab, Drop and Column Properties

A. Click the **Define menu > Slab Properties** command to access the Slab Properties form shown in Figure 5.

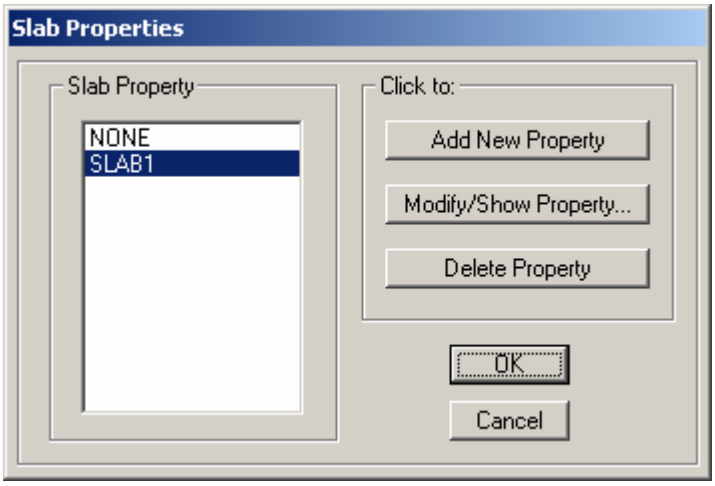


Figure 5
Slab Properties form

B. In the "**Slab Property**" area, highlight *SLAB1*.

C. Recall that for this tutorial project, the slab thickness is 10 inches. To adjust the default dimensions of *SLAB1*, click the **Modify/Show Property** button to access the Slab Property Data form shown in Figure 6.

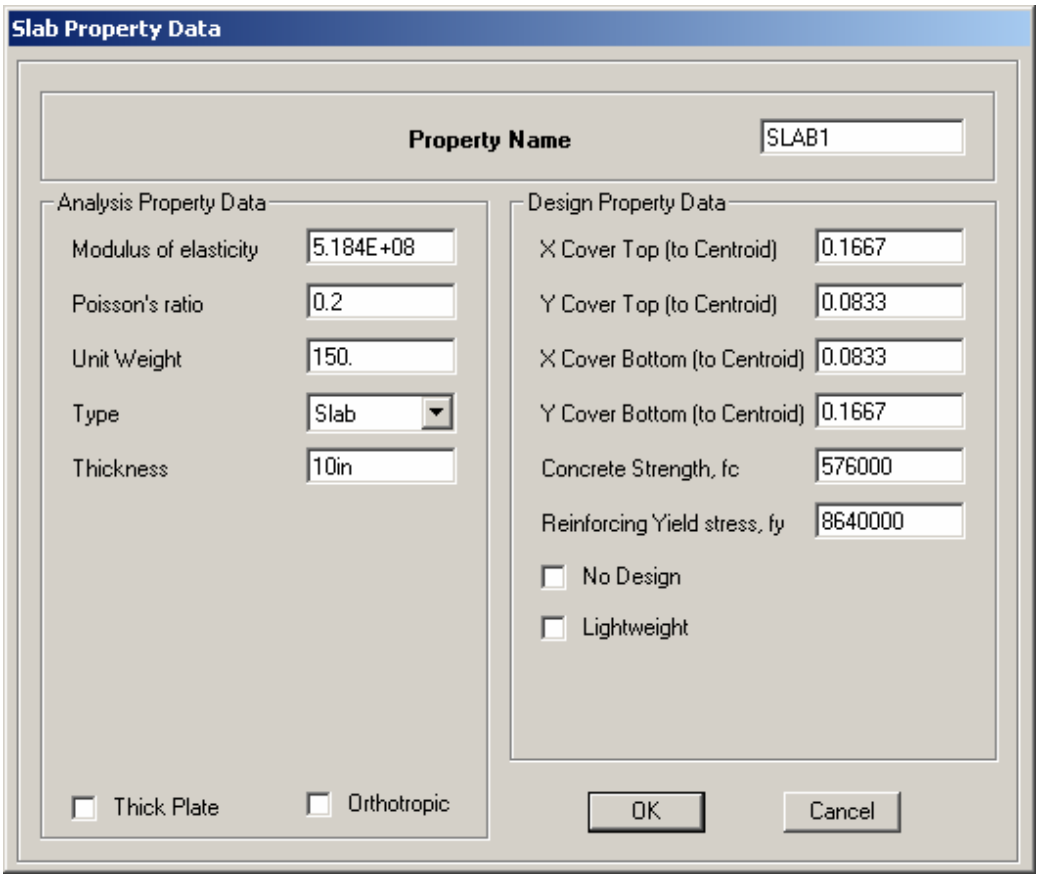


Figure 6
Slab Property Data form

1. Select *Slab* from the Type drop-down list; this ensures that any area object with this property assignment will be identified as a slab member.

   Note: When multiple area objects occupy the same location in plan, SAFE determines which property value to use in the stiffness formula based on the following hierarchy: a Column type

supersedes a Drop type, the Drop type has priority over a Slab type, and an Opening takes precedent over all other types.

2. Type **10in** in the Thickness edit box in the "**Analysis Property Data**" area.

   Note: When the units are explicitly stated, in this case inches (in), the program automatically converts the number input to be consistent with the database units (recall that the units have been set to lbs-ft). The value may also have been input as 10/12, without the inch identifier.

3. Click the **OK** button to accept the changes and return to the Slab Properties form.

D. Recall that the project has 16-inch-thick drop panels on columns. To specify a property for the drop panel, click the **Add New Property** button on the Slab Properties form and complete the following.

   1. Type **DROP** in the Property Name edit box on the Slab Property Data form.
   2. Select *Drop* from the Type drop-down list.
   3. Type **16in** in the Thickness edit box.
   4. Click the **OK** button to accept the changes and return to the Slab Properties form.

E. Recall that the columns in the project are typically 18 inches square. To specify a property for the column, click the **Add New Property** button in the "**Click To**" area of the Slab Properties form and complete the following.

   1. Type **COL** in the Property Name edit box on the Slab Property Data form.
   2. Select *Column* from the Type drop-down list.
   3. Type **50in** in the Thickness edit box. The plan dimensions of the columns will be modeled with area objects so that the geometry of the drops and slabs will be correct. An arbitrary thickness of

50 inches, or five times the slab thickness, is being assigned to the column property to ensure that it is at least a magnitude of two times stiffer than the surrounding slab. This will all but eliminate bending deformations in the column region

4. Check the *No Design* check box in the "**Design Property Data**" area because the column should not be designed using slab design techniques.

5. Click the **OK** button to accept the changes and return to the Slab Properties form.

Click the **OK** button to end the slab property definitions. Click the **File menu > Save** command, or the **Save** button, to save the model.

Define Beam Properties

A. Click the **Define menu > Beam Properties** command, which will display the Beam Properties form.

B. In the Beam Property area, highlight *Beam1*. Recall that the beams for the project are 12-inch by 24-inch (or 1 foot by 2 feet in the currently specified lb-ft units). To review the default dimensions of *Beam1*, click the **Modify/Show Property** button to access the Property Data for Rectangular Beam form shown in Figure 7.

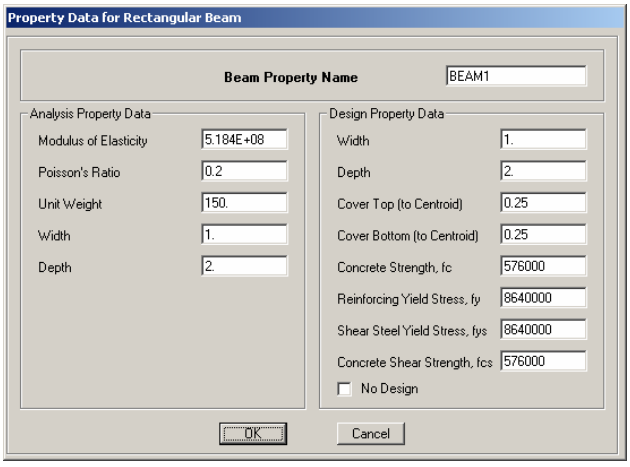


Figure 7
Property Data for Rectangular Beam form

When the form appears, it may have the appropriate values in the "**Analysis Property Data**" area (Width of 1 and Depth of 2) and in the "**Design Property Data**" area (Width of 1 and Depth of 2). If these values are not displayed, type them into the appropriate edit boxes.

C. Click the **OK** button twice to end the beam property definition. Click the **File menu > Save** command, or the **Save** button, to save your model.

Step 3 Define Supports

In this Step, properties for the column and wall supports are defined. From the data specified in the following Action Items, the program will calculate the properties of the springs used to model the column and wall supports.

Define Column Supports

A. Click the **Define menu > Column Supports** command to access the Support Properties form.

B. In the "**Support Property**" area, highlight *COL1*.

C. Click the **Modify/Show Property** button to access the Column Support Property Data form shown in Figure 8.

1. Ensure that *Rectangular Properties* is selected in the "**Define Column by**" area of the form.
2. Make sure that *Below Slab Only* is selected in the **"Activate Support Property"** area of the form.
3. Type **18in** in the X Dimension edit box and **18in** in the Y Dimension edit box.
4. Type **12** in the Column Height edit box.

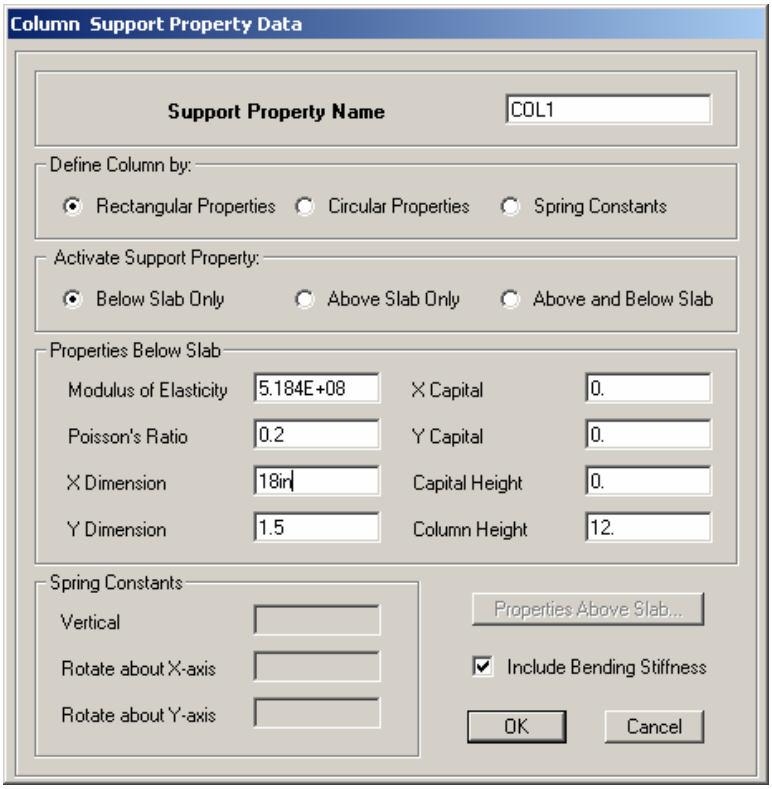


Figure 8 Column Support Property Data form

5. Click the **OK** button to accept the changes and return to the Support Properties form.

D. Click the **OK** button to close the Support Properties form.

Define Wall Supports

A. Click the **Define menu > Wall Supports** command to access the Support Properties form.

B. In the "**Support Property**" area, highlight *WALL1*.

C. Click the **Modify/Show Property** button to access the Wall Support Property Data form shown in Figure 9.

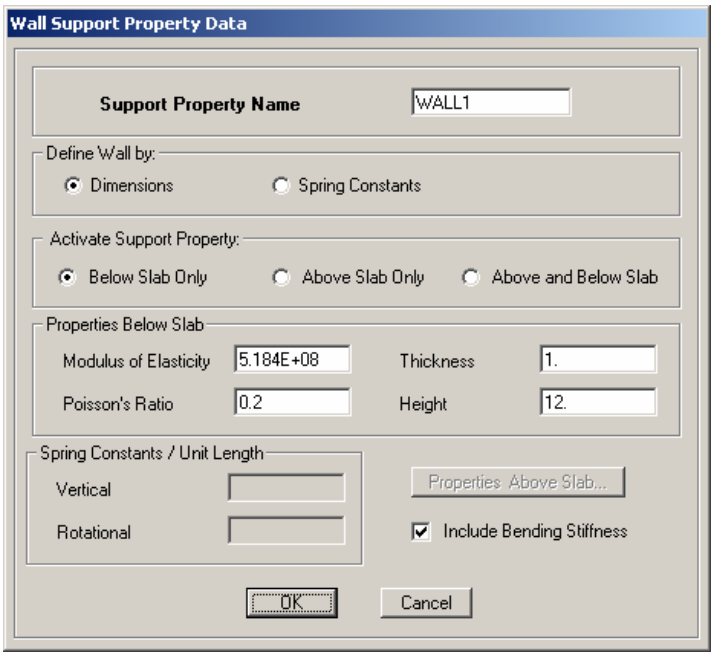


Figure 9
Wall Support Property Data form

1. Ensure that the *Dimensions* option is selected in the "**Define Wall by**" area of the form.

2. Make sure that *Below Slab Only* is selected in the **"Activate Support Property"** area of the form.

3. Recall that the project has walls 12 inches thick and that the units are lb-ft. Thus, type **1** in the Thickness edit box.

4. Also recall that the project's story height is 12 feet. Thus, type **12** in the Height edit box.

5. Click the **OK** button to accept the changes and return to the Support Properties form.

D. Click the **OK** button again to leave the Support Properties form. Click the **File menu > Save** command, or the **Save** button, to save your model.

Step 4 Define Static Load Cases

In this Step, the dead and live static load cases are defined. That is, we will name the load cases, identify the type (live or dead), and specify the self-weight multiplier and the long-term deflection multiplier for cracked deflection analysis. The loads will be assigned to objects, and the values for the loads specified (uniform dead load of 30 psf and live load of 50 psf), in Step 7.

A. Click the **Define menu > Static Load Cases** command to access the Static Load Case Names form shown in Figure 10.

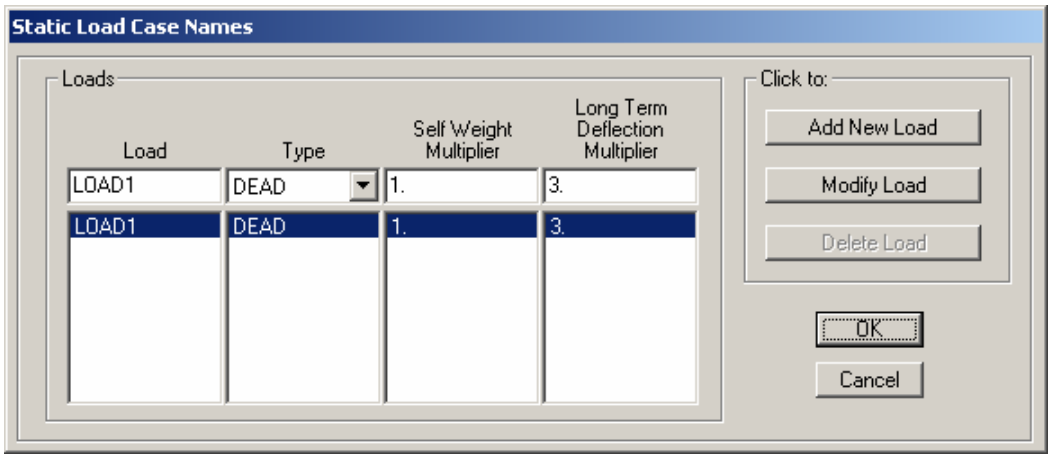


Figure 10
Static Load Case Names form

B. Note that dead load case *LOAD1* is defined by default. We will change the name of that load case to *DL* by clicking on *LOAD1* in the Load edit box, and then typing **DL**.

C. Because we need to define a dead load case, leave the Type set to *DEAD*.

D. Recall that the project will be analyzed for the dead load plus the self-weight of the structure. Thus, the Self Weight Multiplier should be set equal to 1 (this will include 1.0 times the self weight of all members).

E. The Long Term Deflection Multiplier is used by the program when doing a cracked deflection analysis and is meant to account for be-

havior such as shrinkage and creep. For this tutorial, a cracked deflection analysis will not be performed, and therefore the long-term deflection multiplier is not applicable.

F. Click the **Modify Load** button to accept the changes.

G. Now we will define the live load case. Click in the Load edit box and type **LL**.

H. Select *LIVE* from the Type drop-down list.

I. Type **0** (zero) in the Self Weight Multiplier edit box.

J. Type **1** in the Long Term Deflection Multiplier edit box.

K. Click the **Add New Load** button to add the LL load to the load list. The Static Load Case Names form should appear as shown in Figure 11.

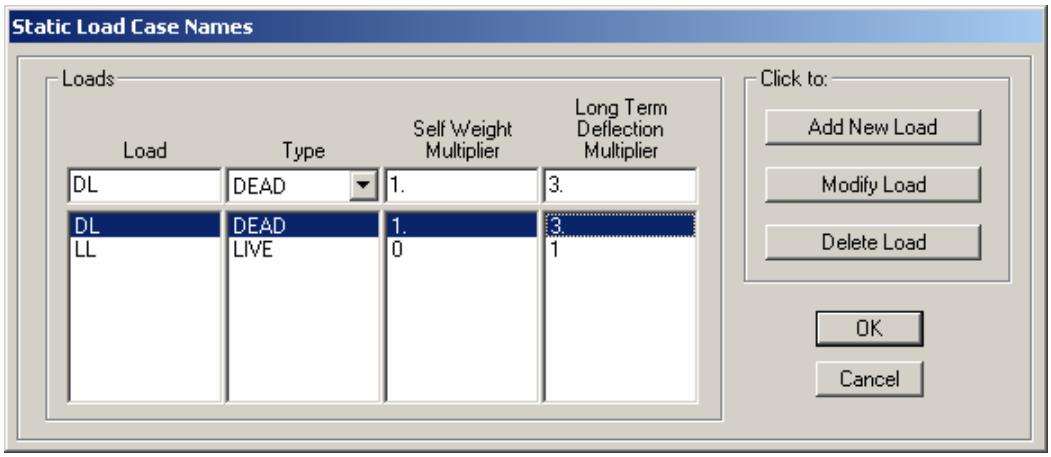


Figure 11
Static Load Case Names form with load cases defined

L. Click the **OK** button to accept all of the defined static load cases. Click the **File menu > Save** command, or the **Save** button, to save your model.

Step 5 Draw Objects

In this Step, area, point and line objects will be drawn to represent slabs, drops, columns, column supports and wall supports.

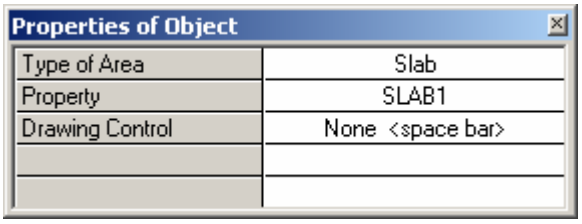
Note:
A display window is active when the Display Title Bar, just below the menu bar, is highlighted.

Draw Slabs (Area Objects)

With two windows tiled vertically (**Options menu > Windows** command), ensure that the left window is active and set to a Structural Layer Plan View (click anywhere in the left window and use the **View menu > Set Structural Layer** command to set this view). Also check that the snap to points/intersections feature is enabled (click the **Draw menu > Snap to > Points** command or use the **Snap to Grid Intersections and Points** button); the feature is enabled when the associated button appears depressed. This will assist in accurately positioning the area objects. Now draw area objects to model the slab using the following Action Items.

A. Click the **Draw menu > Draw Area Objects** command, or the **Draw Area Objects** button to access the Properties of Object pop-up form for areas shown in Figure 12.

Figure 12
Properties of Object form for area objects



Note:
If the Properties of Object form covers part of the model, click on the blue title bar, hold down the mouse button, and drag it out of the way.

Make sure that the Type of Area is set to *Slab*. If it is not, click once in the edit box opposite the Type of Area item and select *Slab* from the drop-down list.

B. Click in the Property edit box and select *SLAB1* from the drop-down list. This is the slab property defined in Step 2.

C. To draw the first corner of the area object, click once in the Structural Layer Plan View at the intersection of grid lines A and 9. Then moving clockwise around the grid, click once at these grid intersec-

tions in this order to draw the outline of the slab: F9, F1, D1, D4, and A4.

If you have made a mistake while drawing this object, click the **Select Object** button to leave the Draw mode and go to the Select mode. Then click the **Edit menu > Undo Area Add** command, and repeat Items A through C.

D. Click on the **Select Object** button, or press the **Esc** key on the keyboard to exit the **Draw Area Objects** command.

E. **Right** click on the point object located at grid intersection A9 to access the Point Object Information form shown in Figure 13.

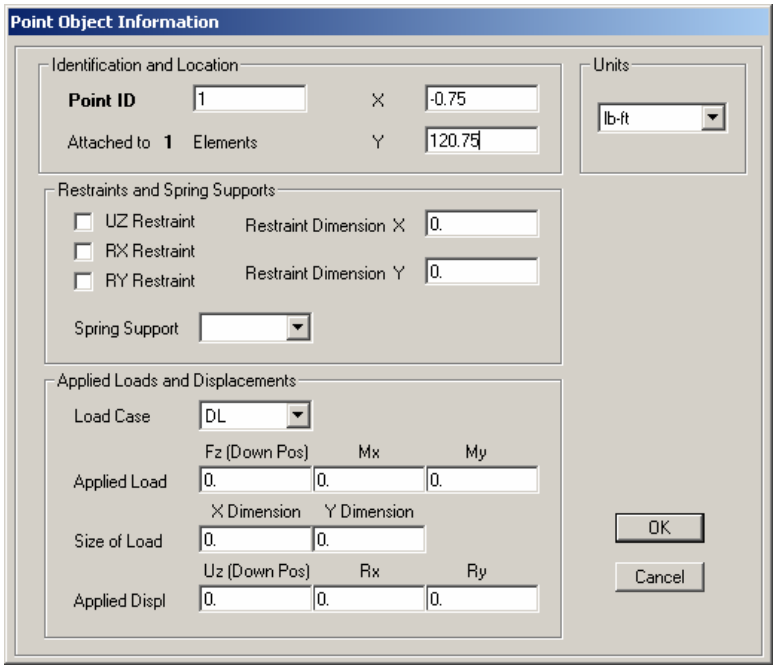


Figure 13
Point Object Information form

This form displays the status of the items currently assigned to the selected object. We will use the form to move the corner points of the slab to account for the 9-inch deep perimeter overhang that is needed to accommodate the width of the columns (1/2 of 18 inches).

1. Click in the X edit box in the "**Identification and Location**" area and type **–9/12**. Note that the units are feet.

2. Click in the Y edit box and type **120.75**.

3. Click the **OK** button to accept the changes. The slab area object has now been expanded by 9 inches at this corner.

Repeat Items 1 through 3 for the following grid intersection points using the X and Y values provided to move the corner points of the slab to model the 9-inch overhang:

Right click this Intersection	Use this X Value	Use this Y Value
F9	113.75	120.75
F1	113.75	-9/12
D1	No Change	-9/12
D4	No Change	47.25
A4	-9/12	47.25

Draw Column Supports (Point Objects)

With the active window set as described in the preceding *Draw Area Objects* section (i.e., left window active with a Structural Layer Plan View displayed and the snap to points/intersection feature enabled), use the following Action Items to draw point objects to model the column supports.

A. Click the **Draw menu > Draw Point Objects** command, or the **Draw Point Objects** button to access the Properties of Object pop-up form for points shown in Figure 14.

Ensure that the Type of Point is set to *Column*. If it is not, click once in the edit box opposite the Type of Point item and select *Column* from the drop-down list.

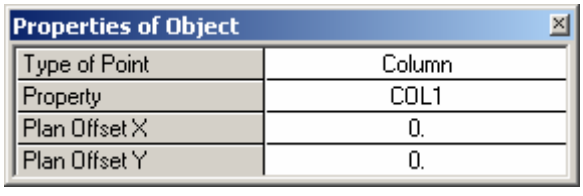


Figure 14
Properties of Object form for point objects

Click in the Property edit box and select *COL1* from the drop-down list. This is the column support (spring) property for the 18-inch by 18-inch by 12-foot tall columns defined in Step 3.

B. Left click once at each of the following grid locations to position the column supports: A9, B9, C9, D9, E9, F9, A8, B8, E8, F8, E7, F7, E5, F5, A4, C4, E3, F3, E2, F2, E1, and F1.

C. Click on the **Select Object** button, or press the **Esc** key on the keyboard to exit the **Draw Point Objects** command.

Draw Wall Supports (Line Objects)

Similar to the preceding two sections, ensure that the Structural Layer Plan View is active and that the snap to points/intersections feature is active. Now draw line objects to model the wall supports as follows.

A. Click the **Draw menu > Draw Line Objects** command, or the **Draw Line Objects** button to access the Properties of Object pop-up form for lines shown in Figure 15.

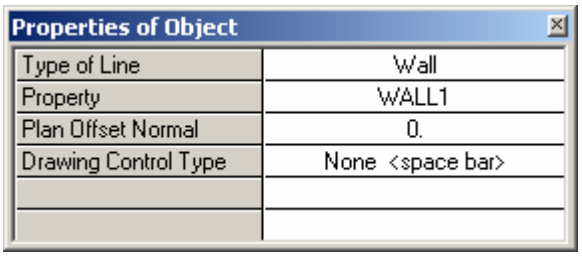


Figure 15
Properties of Object form for line objects

B. Click in the edit box opposite the Type of Line item and select *Wall* from the drop-down list.

C. Click in the edit box opposite the Property item and select *WALL1* from the drop-down list. This is the wall support (spring) property for the 12-inch-thick by 12-foot-tall walls defined in Step 3.

D. Left click at grid intersection A6 to begin drawing the line and at intersection D6 to designate the end of the line. Hit the **Enter** key on the keyboard to complete the line. Repeat this process at the following intersections:

Click here to begin line	Click here to end line	Use this key to end drawing
D8	D1	Enter
C8	D8	Enter
C8	C6	Enter
B6	B4	Enter

E. Click on the **Select Object** button, or press the **Esc** key on the keyboard to exit the **Draw Line Objects** command.

our model should look similar to Figure 16.

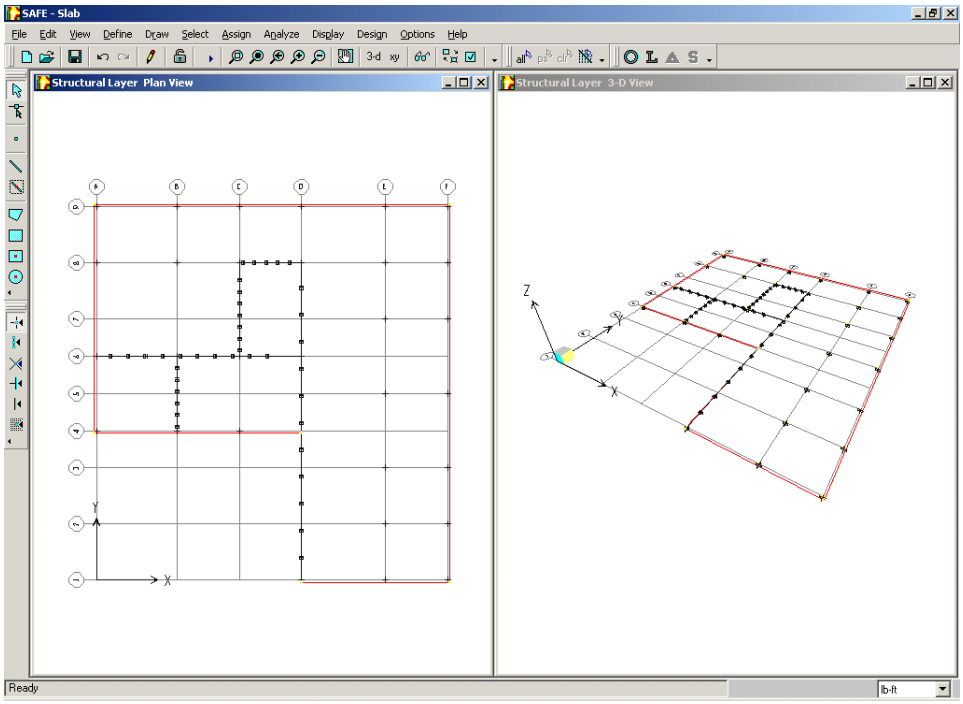


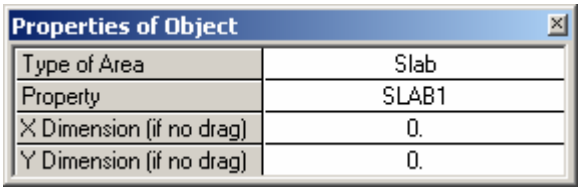
Figure 16
The model after drawing column and wall supports

Draw Drop Panels (Area Objects)

Similar to the previous sections, ensure that the Structural Layer Plan View is active and the snap to points/intersections feature is enabled. Now draw area objects to model the drop panels as follows.

A. Click the **Draw menu > Draw Rectangular Area Objects** command, or the **Draw Rectangular Area Objects** button to access the Properties of Object pop-up form for areas shown in Figure 17.

Figure 17
Properties of Object form for area objects



Make sure that the Type of Area is set to *Slab*.

B. Click in the edit box opposite the Property item and select *DROP* from the drop-down list.

C. Type **6** in the edit box opposite the X Dimension item. The edit box for the Y Dimension should change automatically to 6. This allows a 6-foot by 6-foot area object (drop panel) to be drawn with a single click (i.e., no dragging of the mouse is required).

D. Left click once at the grid intersection A8. SAFE will draw a rectangular area object at this location. Add additional drop panels by clicking at grid intersections B8, C8, D8, E8, E7, E5, A4, C4, E3, E2, and E1.

E. Note that four of the drop panels extend beyond the slab (A8, A4, C4 and E1). To adjust them to match the slab edge, begin by clicking on the **Select Object** button, or press the **Esc** key on the keyboard to exit the **Draw Rectangular Area Objects** command.

F. Hold down the **Ctrl** key on your keyboard and **right** click once on grid intersection A8. A selection list similar to the one shown in Figure 18 appears because multiple objects exist at that location. In this example, a point object and two area objects exist at the same loca-

tion. Note that the selection list will only appear when the **Ctrl** key is used with the click.

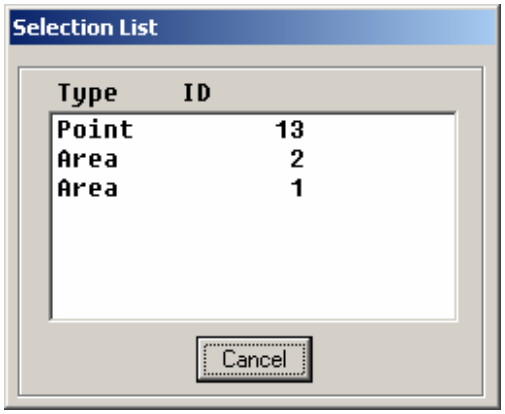


Figure 18 Selection List form

G. Select the area object that is the drop panel (Area 1 should be the primary slab, so the other area object should be the drop panel). Because a right click action initiated the selection process, the Rectangular Area Object Information form for the drop panel, shown in Figure 19, will appear.

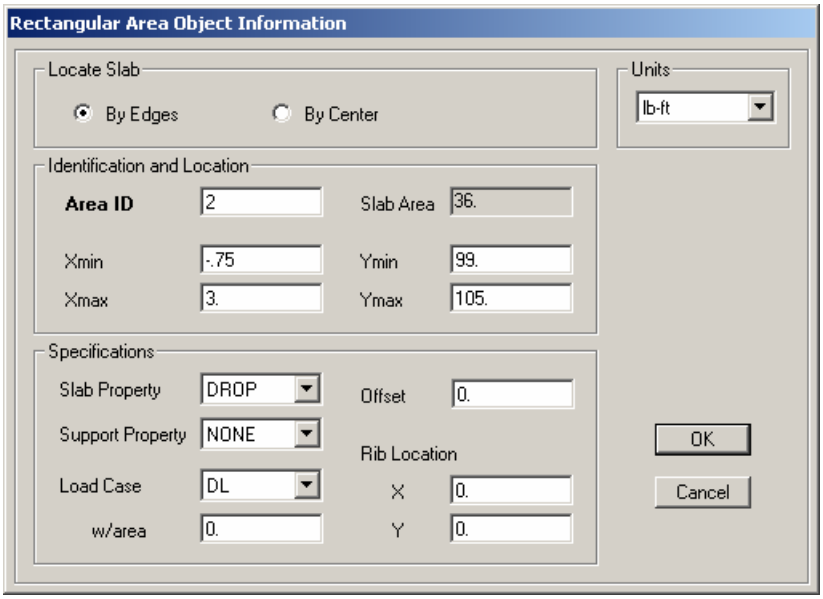


Figure 19 Rectangular Area Object Information form

1. Make sure that the *By Edges* option is selected in the "**Locate Slab**" area.
2. Type **-.75** in the Xmin edit box.
3. Click the **OK** button to accept the changes.

H. Repeat Steps F and G and Action Items 1, 2, and 3 using the following grid intersections and Xmin and Ymin values:

Hold down the Ctrl key; click this grid location	Select this area	Type this Xmin value	Type this Ymin value
A4	drop panel area	-.75	47.25
C4	drop panel area	No change	47.25
E1	drop panel area	No change	-.75

Draw Columns (Area Objects)

Similar to the previous sections, ensure that the Structural Layer Plan View is active and the snap to points/intersections feature is enabled. Remember that the primary purpose for modeling the columns as area objects is to ensure that the drop panels and slabs connected to them have the correct spans. Draw the area objects to model the columns as follows.

A. Click the **Draw menu > Draw Rectangular Area Objects** command, or the **Draw Rectangular Area Objects** button to access the Properties of Object pop-up form for areas shown in Figure 20. Make sure that the Type of Area is set to *Slab*.

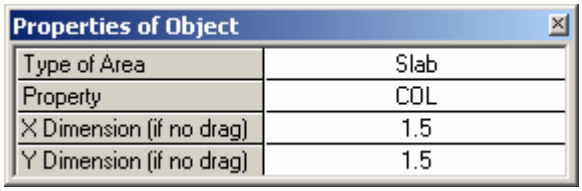


Figure 20 Properties of Object form for areas

B. Click in the edit box opposite the Property item and select *COL* from the drop-down list.

C. Type **1.5** in the edit box opposite the X Dimension item and **1.5** in the edit box opposite the Y Dimension item. This allows an 18-inch by 18-inch area object (column) to be drawn with a single click (i.e., no dragging of the mouse is required).

D. Left click once at grid intersection A9. A rectangular area object should be drawn at this location. Add additional columns by clicking at grid intersections B9, C9, D9, E9, F9, A8, B8, E8, F8, E7, F7, E5, F5, A4, C4, E3, F3, E2, F2, E1 and F1.

E. Click on the **Select Object** button, or press the **Esc** key on the keyboard to exit the **Draw Rectangular Area Objects** command.

Draw Beams (Line Objects)

Similar to the previous sections, ensure that the Structural Layer Plan View is active and the snap to points/intersections feature is enabled. Draw line objects to model the beams as follows.

A. Click the **Draw menu > Draw Line Objects** command, or the **Draw Line Objects** button to access the Properties of Object pop-up form for lines shown in Figure 21.

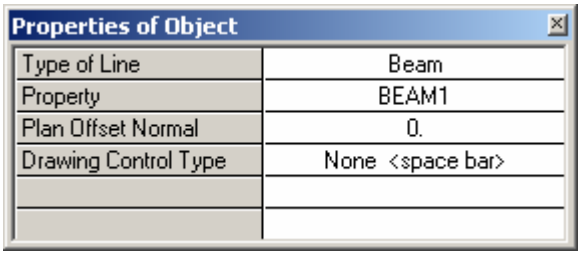


Figure 21
Properties of Object form for lines objects

B. Click in the edit box opposite the Type of Line item and select *Beam* from the drop-down list.

C. Click in the edit box opposite the Property item and select *BEAM1* from the drop-down list. Recall that *BEAM1* is the 12-inch by 24-inch beam defined in Step 2.

D. Left click once at the grid intersection A9. Click again at grid intersections F9, followed by F1. Although only one line object per side

was drawn, the program will automatically mesh this single object internally into multiple beam elements to provide the correct connectivity to the supporting columns and slab elements.

E. Click on the **Select Object** button, or press the **Esc** key on the keyboard to exit the Draw Line Objects command.

Draw Openings (Area Objects)

Similar to the previous sections, ensure that the Structural Layer Plan View is active and the snap to points/intersections feature is enabled. Draw an area object to model the opening as follows.

A. Click the **Draw menu > Draw Rectangular Area Objects** command, or the **Draw Rectangular Area Objects** button. The Properties of Object pop-up form for areas will appear.

B. Click once in the edit box opposite the Property item and select *OPENING* from the drop-down list.

C. Left click in the Structural Layer Plan View at the intersection of grid lines C and 6, and while holding the left mouse button down, move diagonally up to grid intersection D8 and release the button. An area object with an "X" should appear, indicating that an opening for the stairs has been drawn. As stated earlier, an opening (or null slab) takes priority over an object with assigned slab properties when the program determines the stiffness formulation.

D. Click on the **Select Object** button, or press the **Esc** key on the keyboard to exit the Draw Rectangular Area Objects command.

Click the **File menu > Save** command, or the **Save** button, to save your model.

Step 6 Set Object Options

In this Step, the set object options will be used to better view the slab outline.

A. Click the **Set Object Options** button, or the **View menu > Set Object Options** command. When the Set Objects form appears, check the *Fill Elements* check box as shown in Figure 22.

B. Click **OK** to accept the changes, and the model now appears as shown in Figure 23.

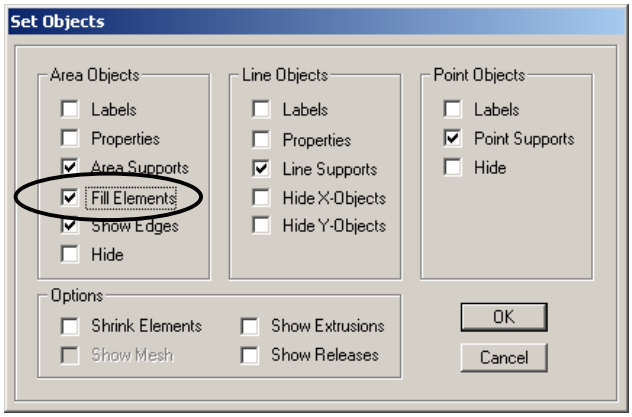


Figure 22
Set Objects form

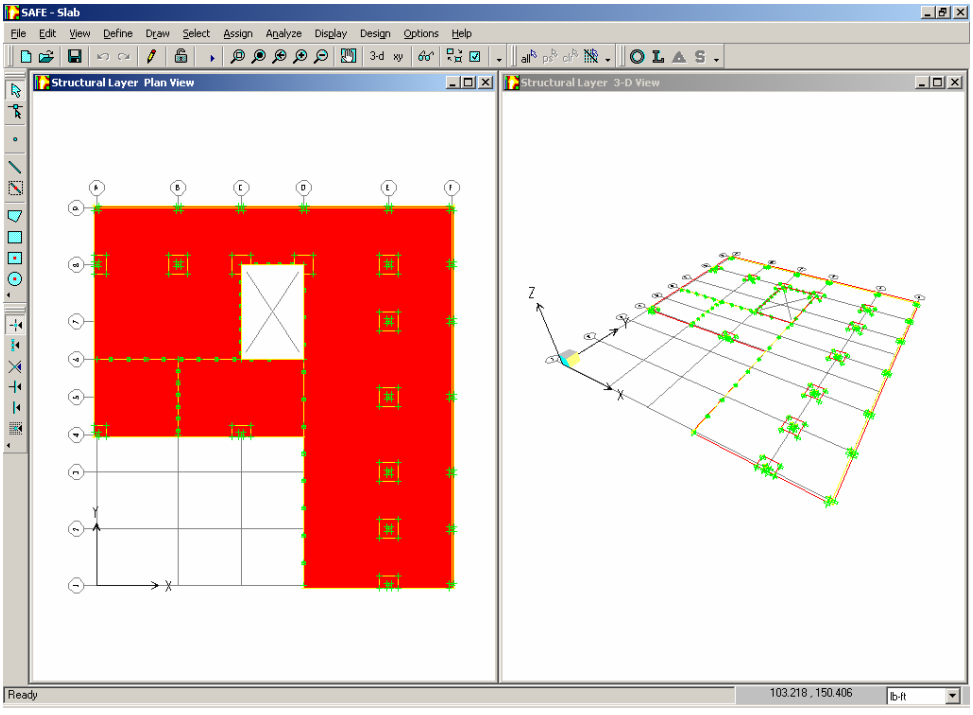


Figure 23
Model after all objects have been drawn

Step 7 Assign Loads

In this Step, the dead and live loads will be assigned to the slab objects. Ensure that the Structural Layer Plan View is still active, and that the program is in the select mode (**Draw menu > Select** command).

A. Select the slab by clicking on it anywhere that is ***not*** a beam, wall support, column or opening. The status bar in the lower left-hand corner should show "1 Areas selected." If you make a mistake in selecting, press the **Clear Selection** button , and try again.

B. Click the **Assign menu > Surface Loads** command to access the Surface Loads form shown in Figure 24.

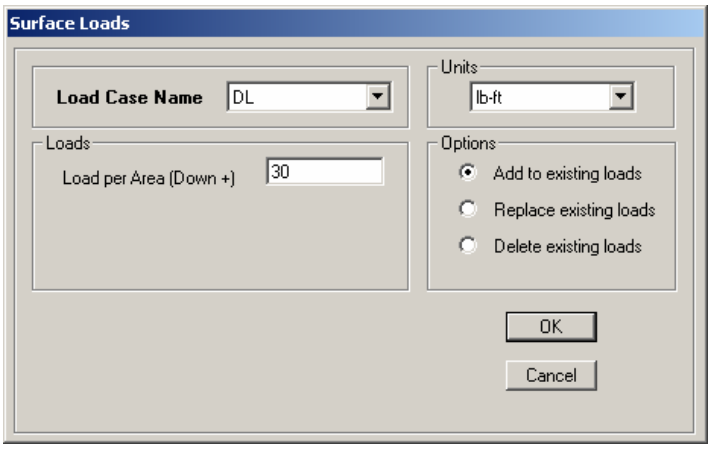


Figure 24
Surface Loads form

1. If it is not already displayed, select *DL* from the Load Case Name drop-down list.
2. Make sure *lb-ft* is displayed in the Units area.
3. Type **30** in the Load per Area (Down +) edit box.
4. Click the **OK** button to accept the dead load assignment. SAFE will display the loads on the model. Use the **Assign menu > Clear Display of Assigns** command to remove the assignments from the display, if desired.

C. Click anywhere on the main area object to reselect the slab, or click the **Select menu > Get Previous Selection** command or the **Get Previous Selection** button to select the slab.

D. Click the **Assign menu > Surface Loads** command to again access the Surface Loads form.

1. Select *LL* from the Load Case Name drop-down list.

2. Type **50** in the Load per Area (Down +) edit box.

3. Click the **OK** button to accept the live load assignment. Again use the **Assign menu > Clear Display of Assigns** command to remove the assignments from the display.

To review the assignments to the slab, **right** click on the slab anywhere that is ***not*** a beam, wall support, column or opening to access the Area Object Information form shown in Figure 25.

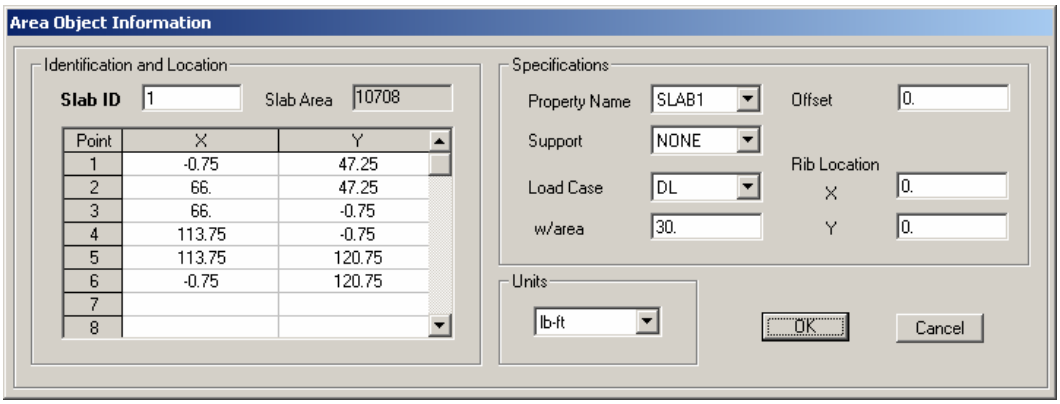


Figure 25
Area Object Information form

Note that the DL Load Case has a w/area (weight/area) of 30. Select the LL Load Case from the drop-down list and the w/area changes to 50. Click the **OK** button to close the Area Object Information form. Click the **File menu > Save** command, or the **Save** button, to save your model.

Step 8 Run the Analysis

In this Step, the analysis will be run.

A. Click the **Analyze menu > Run Analysis** command, or the **Run Analysis** button to start the analysis.

   The program will create the analysis model from your object-based SAFE model, and will display an "Analyzing, Please Wait" window. Data will scroll in this window as the program runs the analysis. This information may be accessed at a later time using the **File menu > Display Input/Output Text Files** command and selecting the file-name with a *.LOG* extension.

B. When the analysis is finished, the message "ANALYSIS COMPLETE" will display. Click **OK** to close the analysis window. The program automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the **Lock/Unlock Model** button appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results.

Step 9 Graphically Review the Analysis Results

In this Step, the analysis will be reviewed using graphical representation of the results.

A. Click once in the right window to make the Structural Layer 3-D View active.

B. Click the **Display menu > Show Deformed Shape** command or the **Show Deformed Shape** button to access the Deformed Shape form shown in Figure 26.

   1. Select the *LL Load Case* from the Load drop-down list.

   2. Select the *Auto* Option from the "**Scaling**" area.

   3. Check the *Displacement Contours* box in the "**Display Options**" area.

4. Click the **OK** button to generate a 3-D deformed shape with contours.

C. Click once in the left window to make the Plan View active.

D. Click the **Display menu > Show Slab Forces** command or the **Show Slab Forces** button to bring up the Slab Forces form shown in Figure 27.

1. Select *LL Load Case* from the Load drop-down list.

2. Select the *Mxx* option from the "**Component**" area.

3. Check the *Display Contours on Deformed Shape* box in the "**Display Options**" area.

4. Click the **OK** button to generate the moment diagram shown in Figure 28.

Note that as you move the cursor over the moment diagram, the values are displayed in the lower left-hand corner of the window.

E. Ensure that the Plan View is active and then click the **Display menu > Show Undeformed Shape** command or the **Show Undeformed Shape** button to clear the display of the moment diagram.

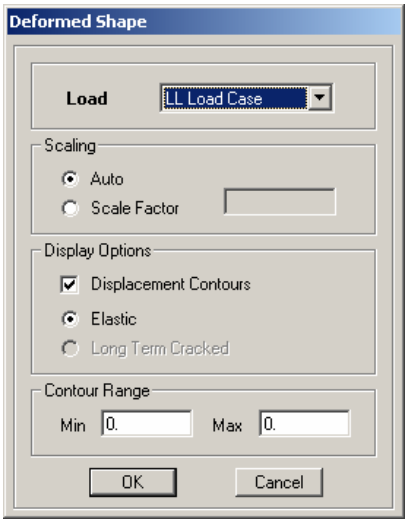


Figure 26
Deformed Shape form

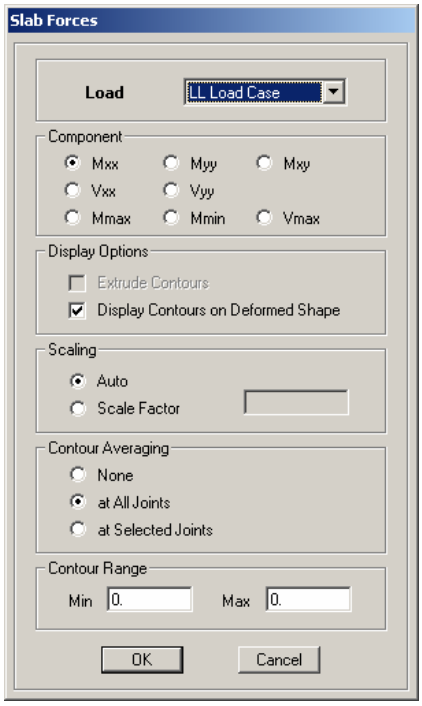


Figure 27
Slab Forces form

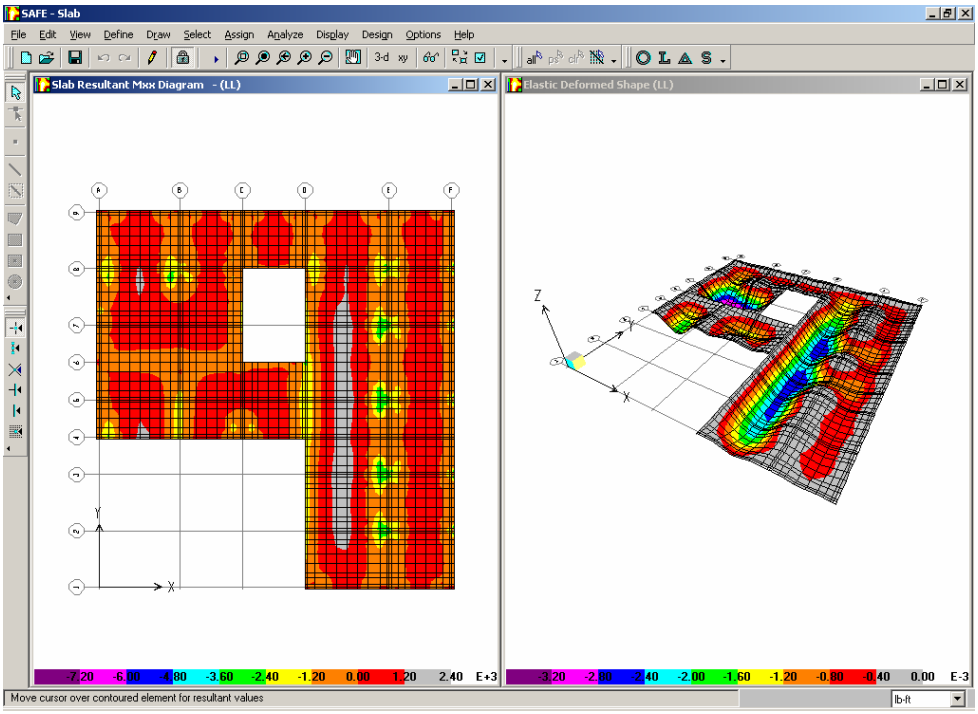


Figure 28
Mxx moment diagram

F. Click the **Set Object Options** button and uncheck the *Fill Elements* check box in the Set Objects form, and then click **OK**.

Step 10 Design

In this Step, the area objects with slab and drop assignments, along with the line objects with beam assignments, will be designed. Note that the analysis should be successfully run before completing the following Action Items.

A. Click the **Options menu > Preferences** command and click on the **Design** tab. The Preferences form shown in Figure 29 appears.

   1. Select the *ACI 318-95* option from the drop-down list in the "**Concrete Design code**" area.

2. Select the *Use Nodal Moments* option in the "**Design Method**" area.

3. Review the information contained in the other edit boxes, and then click **OK** to accept the selections.

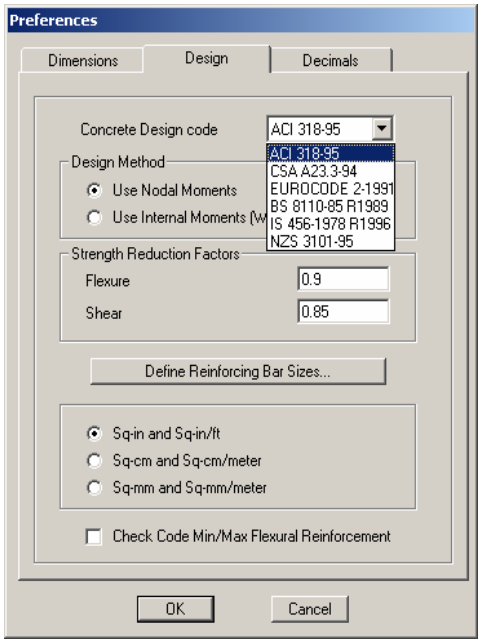


Figure 29
Preferences form

B. Click the **Design menu > Start Design** command to start the design process. The program will calculate the required reinforcing needed for both the slabs and beams.

When the design is complete, the reinforcing required in the X-Strip direction is displayed, as shown in Figure 30.

Positioning the cursor at any location on the X-Strip reinforcing plan causes the required top and bottom reinforcing values to be displayed in the lower left corner of the window.

C. To view the required reinforcing in the Y-Strip direction, click the **Design menu > Display Slab Design Info** command. The Slab Reinforcing form shown in Figure 31 appears.

1. Select the *Y Direction Strip* option in the "**Choose Strip Direction**" area.

2. Review the information contained in the other edit boxes, and then click **OK** to accept the selections.

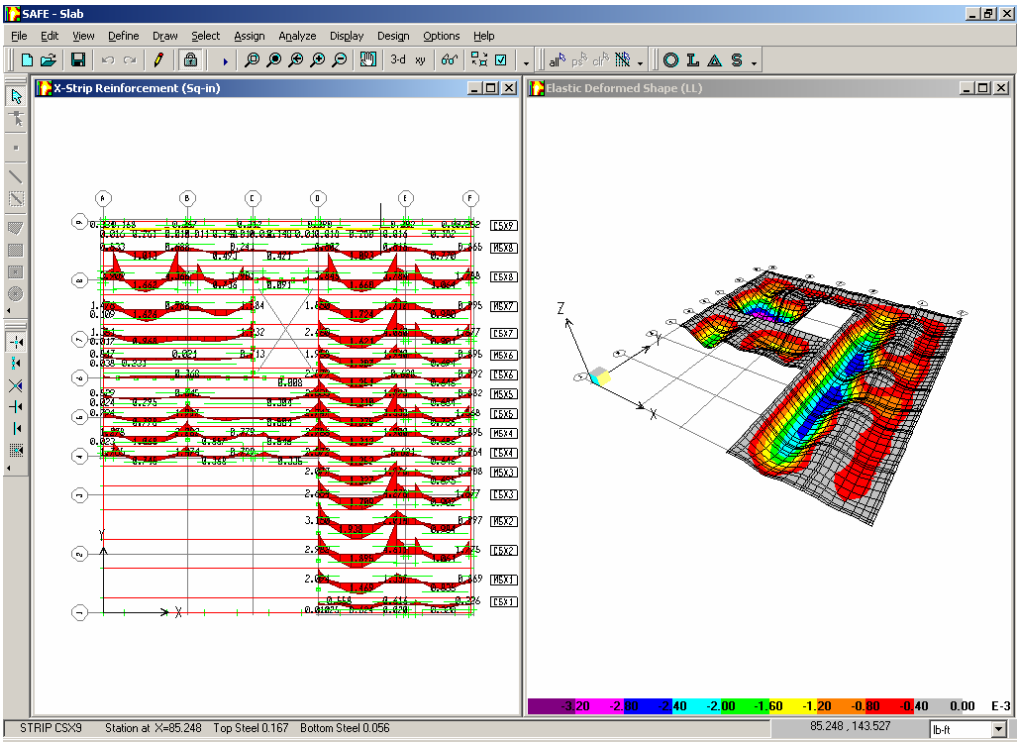


Figure 30
X-Strip reinforcing

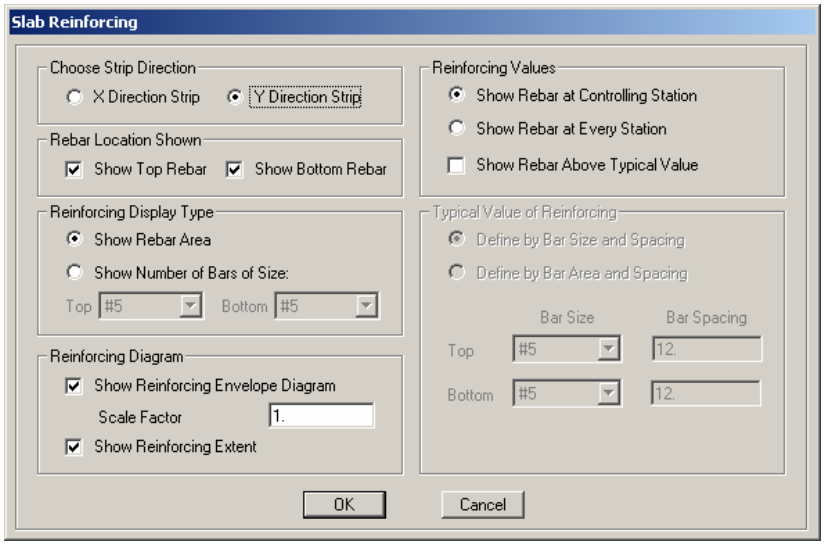


Figure 31
Slab Reinforcing form

The Plan View will be updated to show the required reinforcing in the Y-Strip direction. Again, positioning the cursor anywhere on the Plan View will result in the display of the reinforcing values in the lower left-hand corner of the SAFE window.

D. To view the required reinforcing in the beams, click the **Design menu > Display Beam Design Info** command. The Beam Reinforcing form shown in Figure 32 appears.

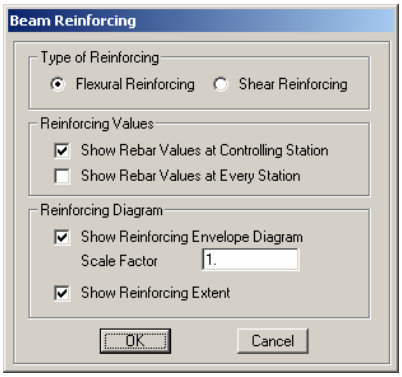


Figure 32
Beam Reinforcing form

1. Select the *Flexural Reinforcing* option in the "**Type of Reinforcing**" area.

2. Review the other selected options and then click **OK** to accept the selections.

   The Plan View will be updated to show the flexural reinforcing required in the beams along the two perimeter sides. Positioning the cursor on the beams will result in the display of the reinforcing values in the lower left-hand corner of the SAFE window.

Click the **File menu > Save** command, or the **Save** button, to save your model.

Congratulations! You've successfully created a SAFE model.