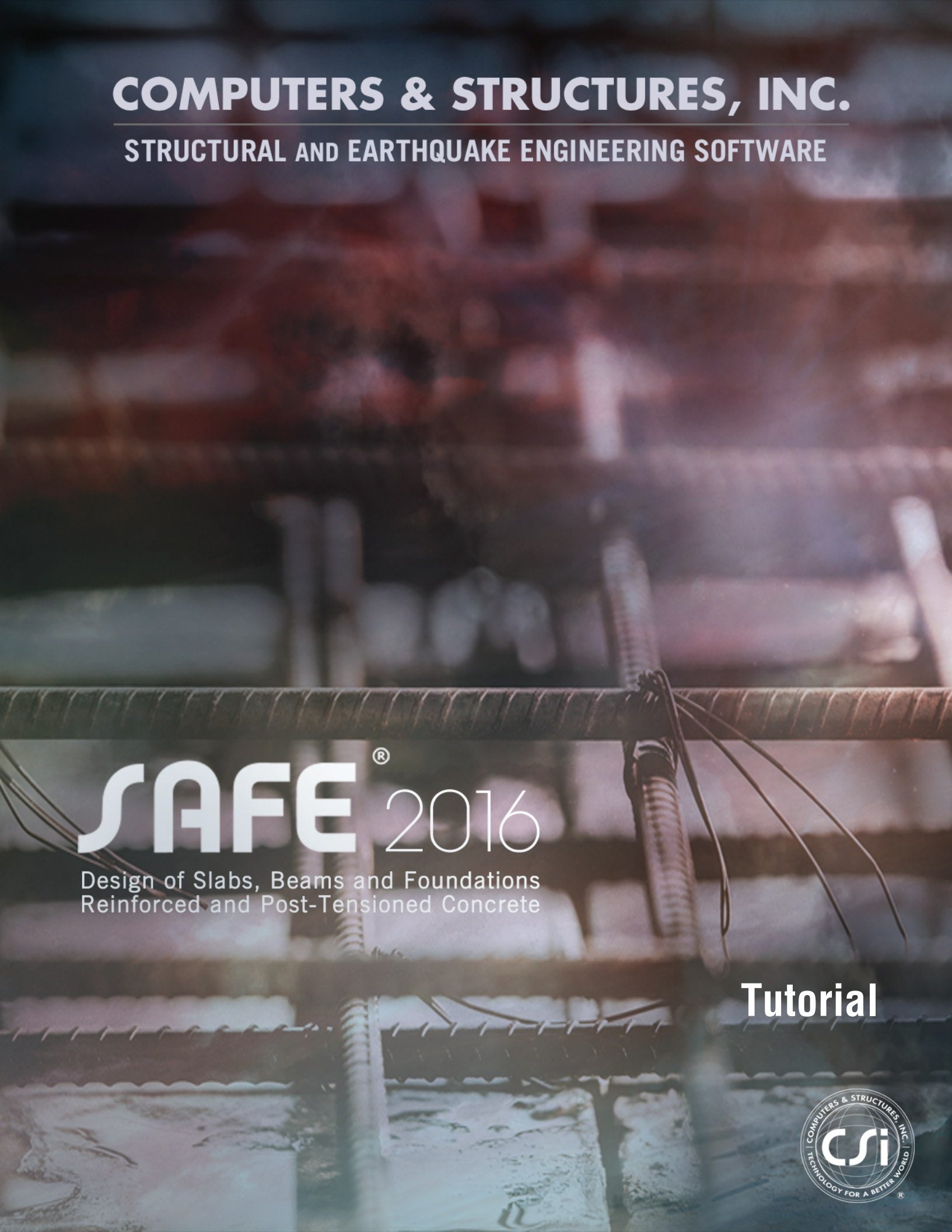


COMPUTERS & STRUCTURES, INC.

STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

SAFE® 2016

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

Tutorial

COMPUTERS & STRUCTURES INC.

SAFE®

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

Tutorial

ISO SAF112816M3 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

Introduction

The two examples in this tutorial provide step-by-step illustrations of how SAFE can be used to create, analyze, design and detail concrete slab systems. The first example is for a mild-reinforced concrete slab (R/C), and the second example deals with a post-tensioned structure (P/T). Although the general procedure is similar for both examples, the structures and specific details differ.

Reinforced Concrete

R/C Example

The intent of this tutorial is to give you hands-on experience via step-by-step instructions on how to use SAFE to model, analyze, design and detail mild reinforced concrete slabs. Fundamentals of the model creation process are identified and various model construction techniques are introduced. 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 18-inch by 24-inch beams on two perimeter sides. Columns are 18 inches square, drop panels are typically 6 feet 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.

Concrete Materials:

Concrete strength, f'c = 4000 psi

Unit weight of concrete = 150 pcf

Mild-steel reinforcing:

f_y = 60 ksi

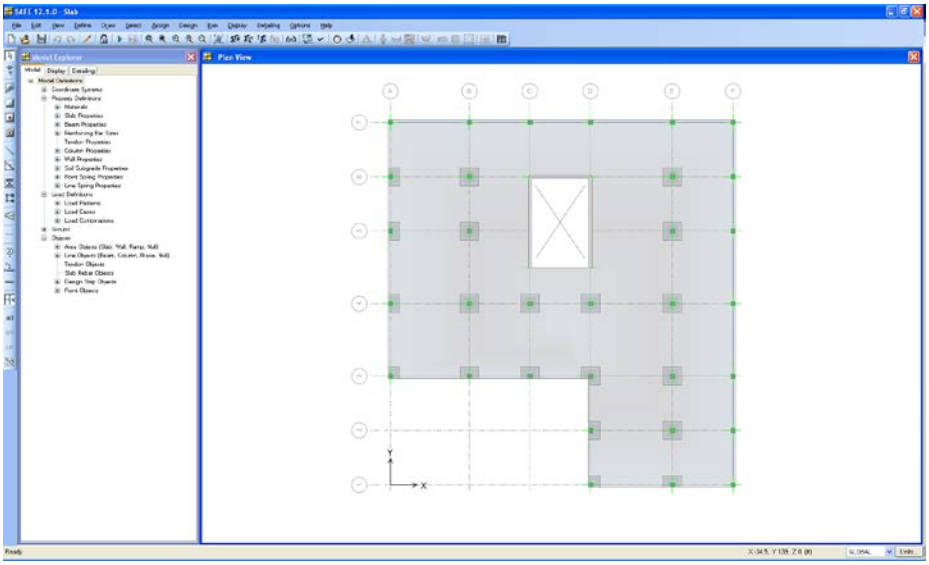


Figure 1 The Project Model

Navigating Through SAFE

The SAFE program provides the user with two principal ways to navigate through program commands: menu commands or toolbar buttons. All commands are available through the main menu bar (e.g., **Draw menu > Draw Columns**), and a majority of the menu commands are also available as buttons on toolbars (e.g., **Draw Columns**,). The availability of a button on the toolbar is indicated in the menus by the existence of an icon to the left of the command, as shown in Figure 2.

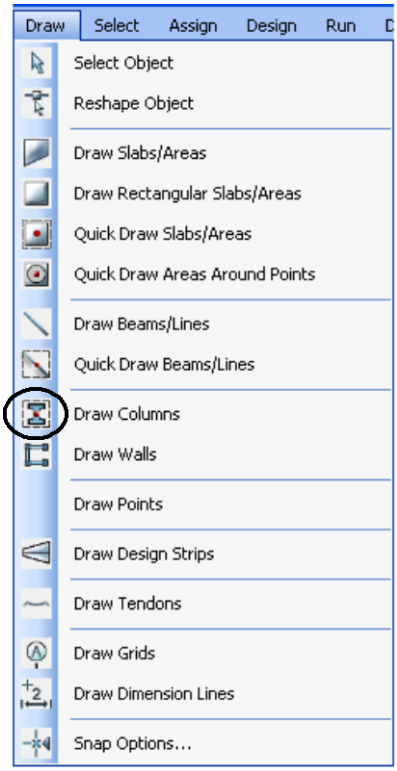


Figure 2 Draw Menu

In this tutorial, the reference to various commands will be given using the narrative description, i.e., **Draw menu > Draw Column** command, in lieu of the associated button.

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 to demonstrate how to construct a model from scratch. However, as a general rule, we highly recommend using templates to start models whenever possible because they provide a quick, easy way of generating a model. Consult the SAFE Help topics for information about templates.

Define the Grid

A. Click the **File menu > New Model** command to access the New Model Initialization form shown in Figure 3. This form is used to specify the starting point of the model creation: a Blank screen, a screen with a Grid Only, or one of eight templates. Default units also may be selected here, along with the design code and preferences.

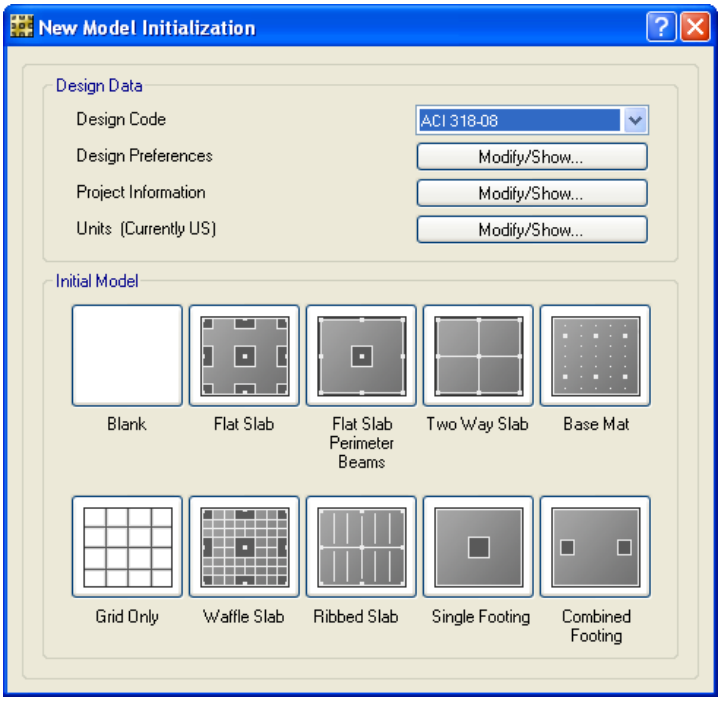


Figure 3 New Model Initialization form

B. In the Design Data area, select ACI 318-08 from the Design Code drop-down list.

C. In the Design Data area, verify that the Units are set to Currently US; if not, click the **Modify/Show** button and select the U.S. Defaults on the Units form.

D. In the Initial Model area, click the **Grid Only** button to display the Coordinate System Definition form shown in Figure 4. This

form is used to specify the number of grids and spacing in each direction. It is important that the grid is defined so as to accurately represent the geometry of the structure; so it is advisable to spend time carefully planning the number and spacing of the grid lines.

E. Select the *Cartesian* option.

F. As shown in Figure 4, set the Number of Grid Lines in the X Direction to **6** and in the Y Direction to **7**. Set the Spacing in the X Direction to **20** feet and in the Y Direction to **18** feet.

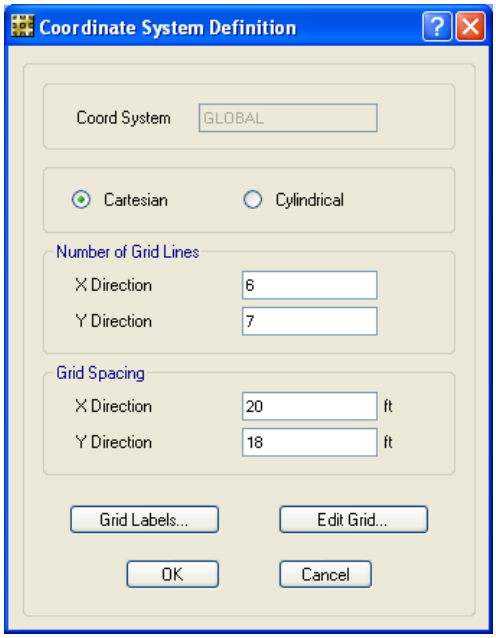


Figure 4 Coordinate System Definition form

G. Click the **Edit Grid** button to display the form shown in Figure 5. The Coordinate System form is used to modify and edit the grid definitions, as well as set the top of model datum. It also allows the user to set the display options associated with the grids.

1. In the *Display Grid Data as* area, select the *Spacing* option.

2. In the **X Grid Data** table, change the X spacing as follows:

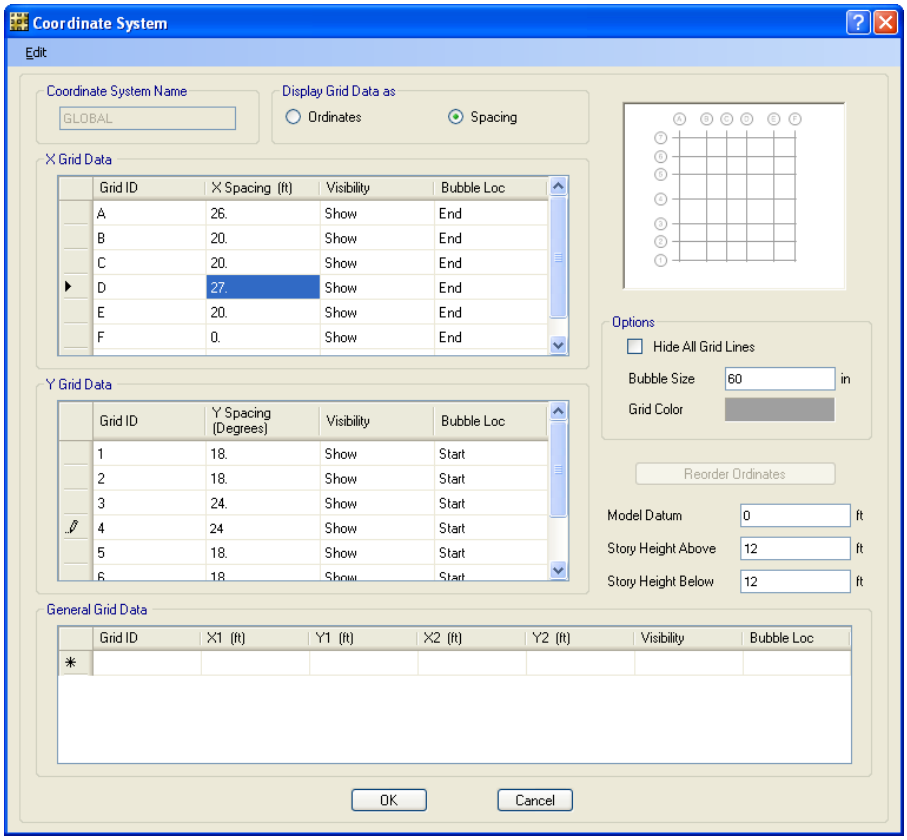


Figure 5 Coordinate System form

Grid ID	Change X Spacing to
A	26
D	27

3. In the **Y Grid Data** table, change the Y spacing as follows:

Grid ID	Change Y Spacing to
3	24
4	24

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

Upon closing the Coordinate System form, by default, the grid system displays in the main SAFE window, with two windows tiled vertically: a Model Explorer window on the left and a Plan View on the right. The number of view windows can be changed using the **Options menu > Windows** command.

H. Click the **View menu > Set Display Options** command to display the Set Display Options form. Uncheck the *Horizon* option and click the **OK** button to exit that form. The Horizon option displays a plane that resembles an engineering calculation grid to illustrate the datum plane location; we are turning this option off to display our coordinate system grid better.

You should now have a display similar to that shown in Figure 6.

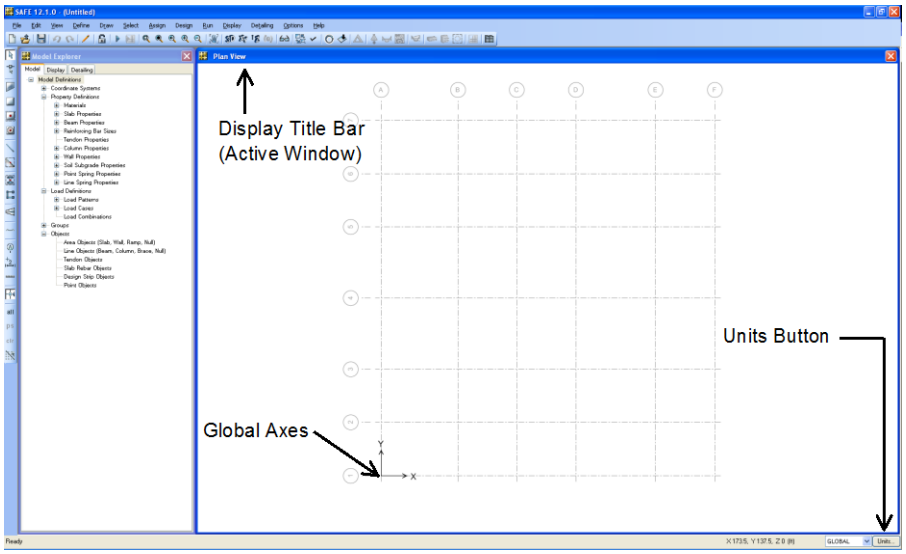


Figure 6 SAFE Main Window

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

Note that the Global Axes are displayed and that the Z positive is in the "up" direction. When SAFE refers to the direction of gravity, this is in the negative Z direction, or "down."

Save the Model

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

Typically a model would be saved with the same name. However to record 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, material and section properties for the slab (area object) and beams (line objects), columns, and walls are defined. Note that previously defined materials and properties may be reviewed and modified using the Model Explorer window (see Figure 7). To do this, expand any of the property items on the tree by clicking the + node, and then double click on the desired item to display the associated form.

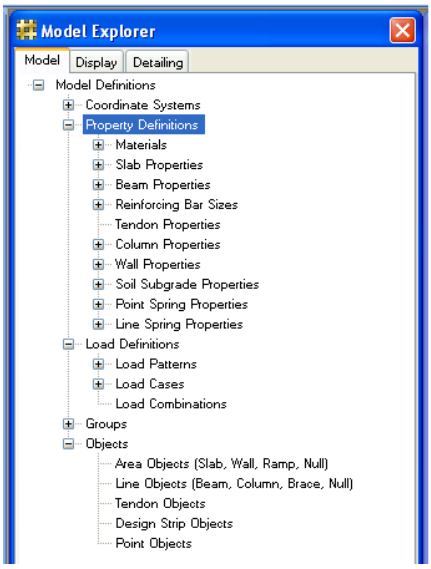


Figure 7 Model Explorer window

Define Material Properties

A. Click the **Define menu** > **Materials** command to access the Materials form shown in Figure 8.

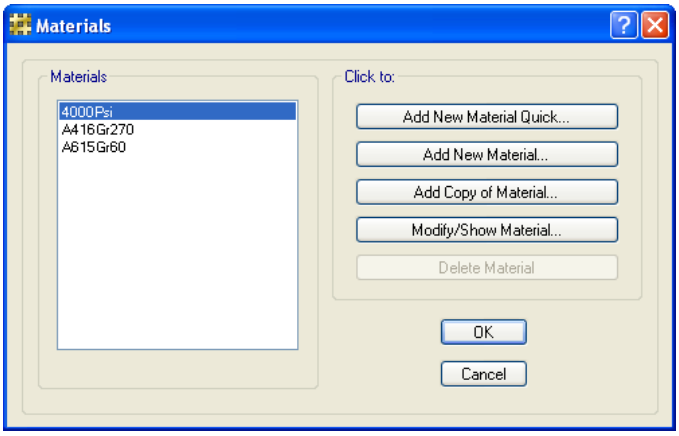


Figure 8 Materials form

B. Highlight *4000Psi* in the Materials area, and click the **Modify/Show Material** button to display the Material Property Data form shown in Figure 9. That form lists the properties associated with 4000psi concrete; this is the concrete property that will be used in our model.

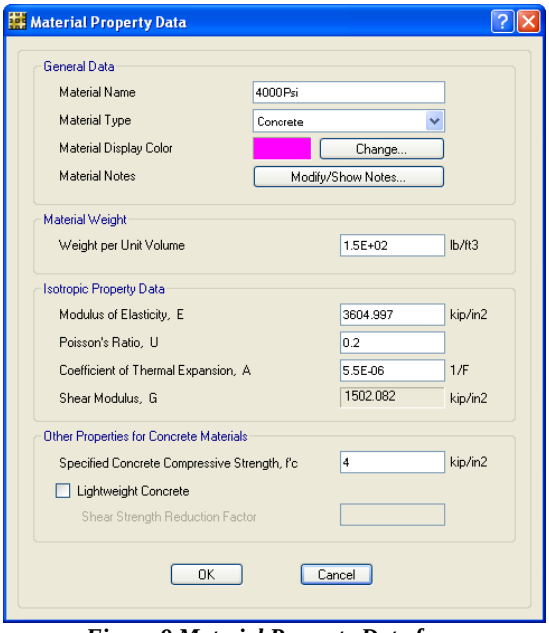


Figure 9 Material Property Data form

C. Click the **OK** button to accept this material as defined.

D. In the Materials area, highlight *A615Gr60*.

E. Click the **Modify/Show Material** button to display the Material Property Data form shown in Figure 10. This form lists the properties associated with Grade 60 reinforcing; this is the rebar property that will be used in our model.

F. Click the **OK** button to accept this material as defined.

G. Click the **OK** button on the Materials form to accept all of the defined materials.

H. Click the **File menu > Save** command to save your model.

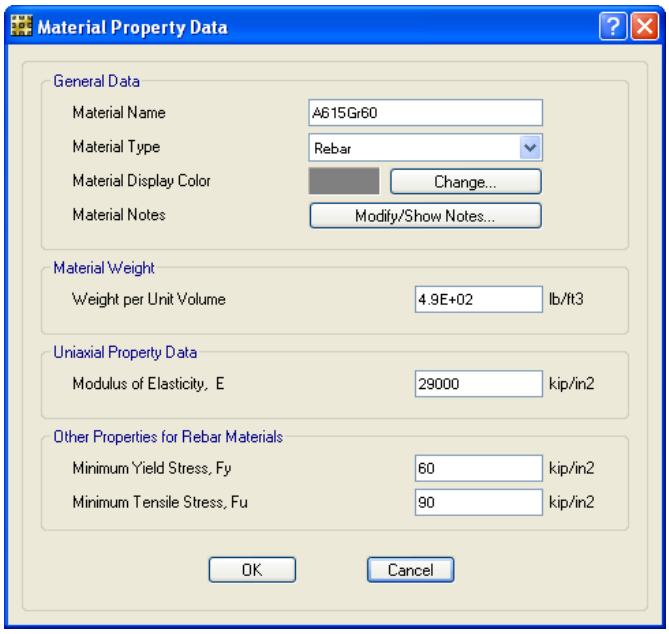


Figure 10 Material Property Data form

Define Slab and Drop Properties

A. Click the Define menu > Slab Properties command to access the Slab Properties form shown in Figure 11.

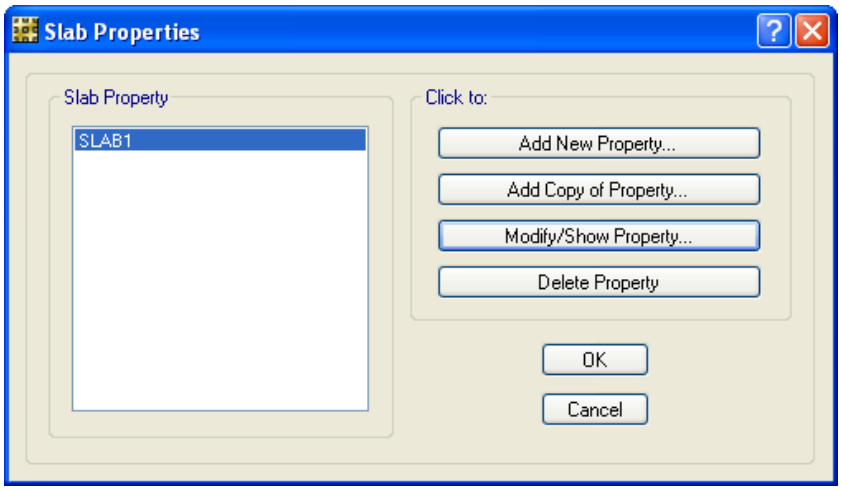


Figure 11 Slab Properties form

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 12.

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

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

   Note: Input may be done in units other than those shown on the form by explicitly stating the units. For example, if for this case the thickness was to be 18 inches, input could be 1.5ft, and the program automatically converts the number input to be consistent with the units shown on the form.

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

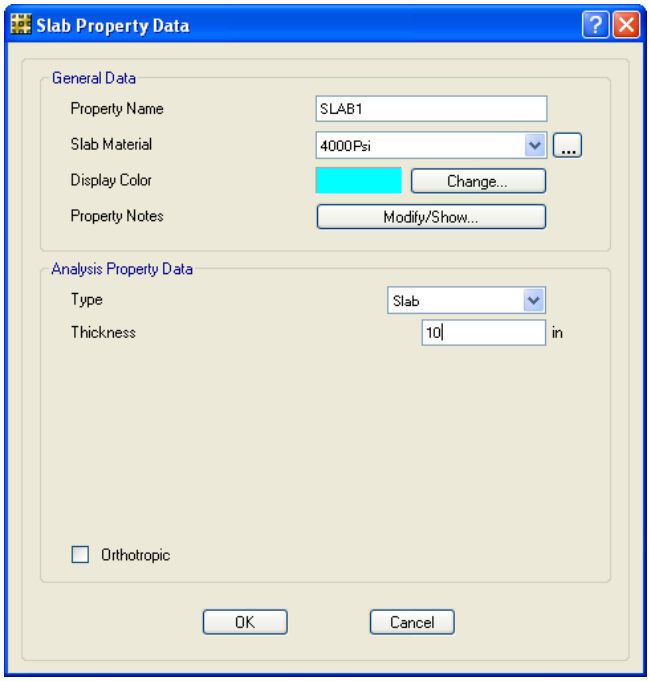


Figure 12 Slab Property Data form

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 in the Analysis Property Data area.

   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: the Drop type has priority over a Slab type.

3. Type **16** in the Thickness edit box.

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

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

Define Beam Properties

A. Click the **Define menu > Beam Properties** command to access the Beam Properties form shown in Figure 13.

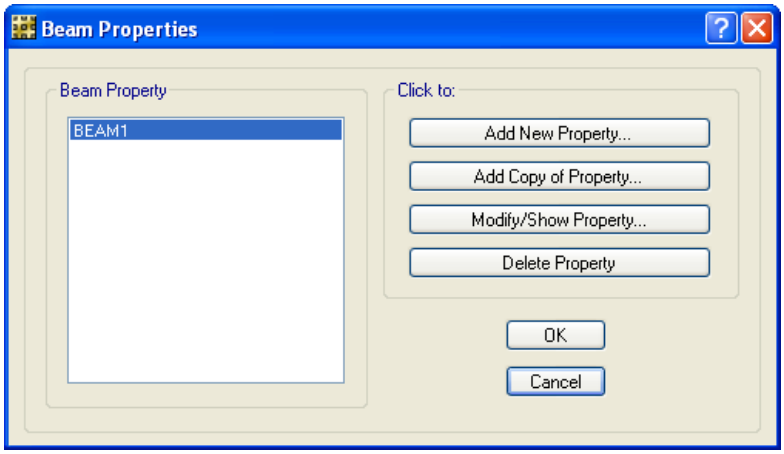


Figure 13 Beam Properties form

B. In the Beam Property area, highlight *BEAM1*.

C. Click the **Modify/Show Property** button to access the Beam Property Data form shown in Figure 14. Recall that the beams for the project are 18 inches by 24 inches.

1. In the Analysis Property Data area, select *L Beam* from the Beam Shape Type drop-down list.

2. In the Analysis Property Data area, type **18** into the Web Width at Top edit box, type **18** into the Web Width at Bottom edit box, type **24** into the Depth edit box, type **5ft** into the Flange Width edit box, and **10** into the Slab Depth edit box.

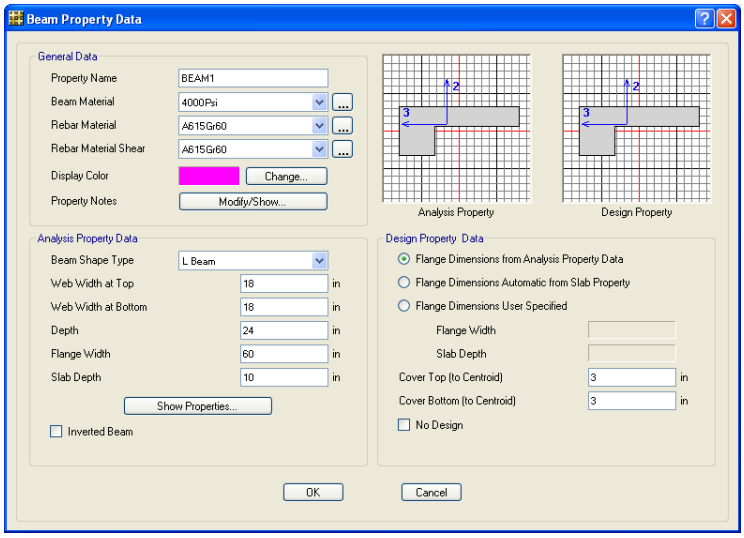


Figure 14 Beam Property Data form

3. In the Design Property Data area, select the *Flange Dimensions from Analysis Property Data* option. This option utilizes the flange width and depth provided in the analysis property data.

4. Click the **OK** button to end the beam property definition.

D. Click the **OK** button to leave the Beam Properties form. Click the **File menu > Save** command to save your model.

Define Column Properties

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

B. In the Column Property area, highlight *COL1*.

C. Click the **Modify/Show Property** button to access the Column Property Data form shown in Figure 15. Recall that the columns for the project are 18 inches square and that the drop panels are 6 feet square.

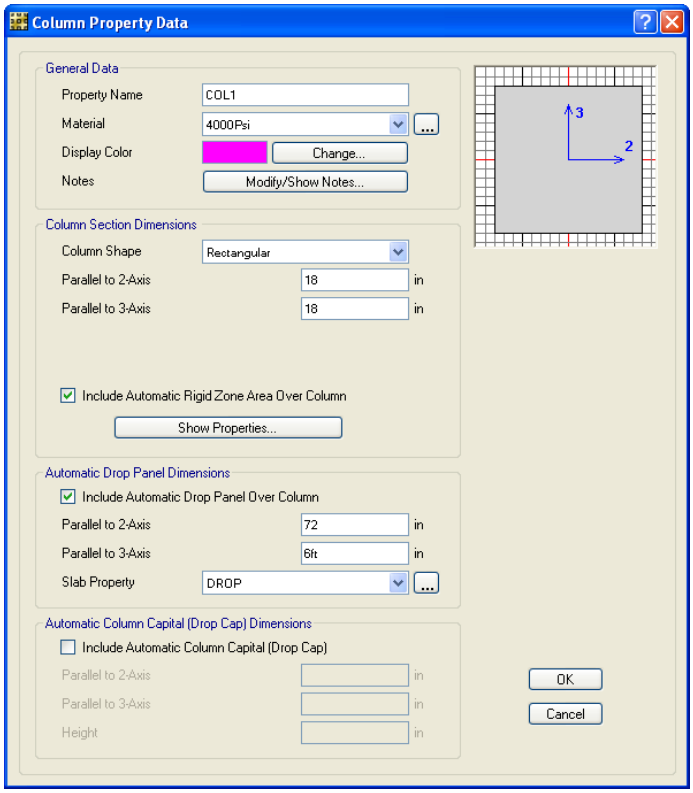


Figure 15 Column Property Data form

1. In the Column Section Dimensions area, select *Rectangular* from the Column Shape drop-down list.

2. In the Column Section Dimensions area, type **18** into the Parallel to 2-Axis edit box and type **18** into the Parallel to 3-Axis edit box.

3. Make sure that the *Include Automatic Rigid Zone Area Over Column* option is checked. This option restricts deformation of the slab at the column location, which prevents unrealistic peaks in moment distribution from occurring.

4. In the Automatic Drop Panel Dimensions area, check the *Include Automatic Drop Panel Over Column* option. This op-

tion automatically adds a drop panel when a column with this property is drawn.

5. In the Automatic Drop Panel Dimensions area, type **6ft** into the Parallel to 2-Axis edit box and type **6ft** into the Parallel to 3-Axis edit box.

6. In the Automatic Drop Panel Dimensions area, select *DROP* from the Slab Property drop-down list.

7. Click the **OK** button to leave the Column Property Data form.

D. Click the **Add New Property** button to access the Column Property Data form.

1. In the General Data area, type **COL-NODROP** in the Property Name edit box.

2. In the Column Section Dimensions area, select *Rectangular* from the Column Shape drop-down list.

3. In the Column Section Dimensions area, type **18** into the Parallel to 2-Axis edit box and type **18** into the Parallel to 3-Axis edit box.

4. Make sure that the Include Automatic Rigid Zone Area Over Column option is checked.

5. In the Automatic Drop Panel Dimensions area, make sure that the *Include Automatic Drop Panel Over Column* option is unchecked.

6. Click the **OK** button to leave the Column Property Data form.

E. Click the **OK** button to accept the Column Property definitions.

Define Wall Properties

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

B. In the Wall Property area, highlight *Wall1*.

C. Click the **Modify/Show Property** button to access the Wall Property Data form shown in Figure 16. Recall that the walls for the project are 12 inches thick.

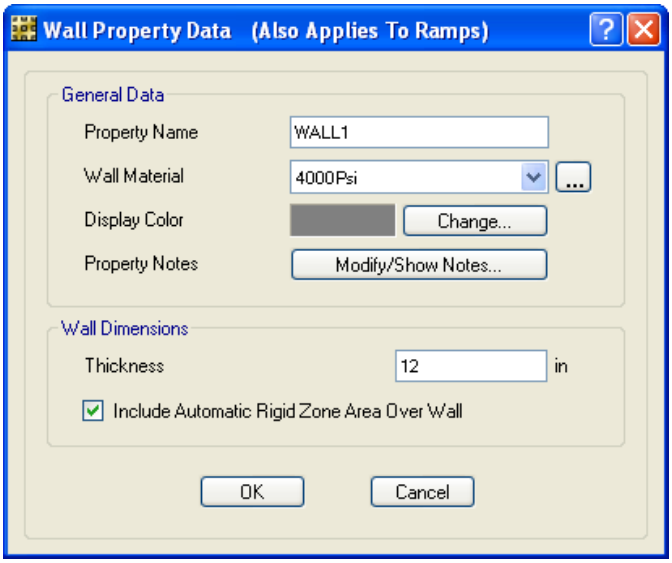


Figure 16 Wall Property Data form

1. In the Wall Dimensions area, type **12** into the Thickness edit box.

2. Make sure that the *Include Automatic Rigid Zone Area Over Wall* option is checked. This option restricts deformation of the slab at the wall location, which prevents unrealistic peaks in moment distribution from occurring.

3. Click the **OK** button to leave the Wall Property Data form.

D. Click the **OK** button to accept the Wall Property definition.

This completes the material and section property definition phase of the model creation. The slab and beam properties will be assigned in the model datum plane, while the columns and walls will be assigned as supports. Supports also can be assigned as point restraints, point springs or line springs. Support stiffnesses are calculated by SAFE based on the cross-sectional properties, material properties, and lengths specified when the columns and walls are drawn.

Step 3 Define Static Load Patterns

In this Step, the dead and live static load patterns are defined. That is, we will name the various types of loads and specify the self-weight multipliers. 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 8.

A. Click the **Define menu > Load Patterns** command to access the Load Patterns form shown in Figure 17.

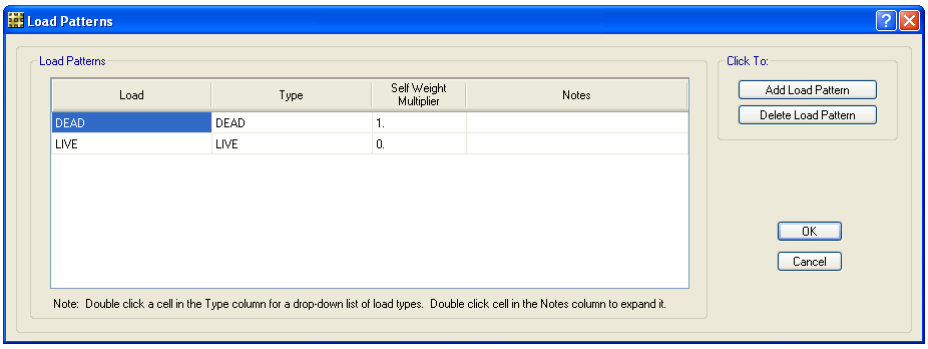


Figure 17 Load Patterns form

B. Note that load patterns *DEAD* and *LIVE* are defined by default.

C. 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) for the *DEAD* load. Only the *DEAD* load pattern should have a non-zero Self Weight Multiplier.

D. Click the **OK** button to accept the defined static load patterns.

E. Click the **File menu > Save** command.

Step 4 Define Load Cases

In this Step, the Load Cases are defined. This is where the type of analysis is specified.

A. Click the **Define menu > Load Cases** command to access the Load Cases form shown in Figure 18.

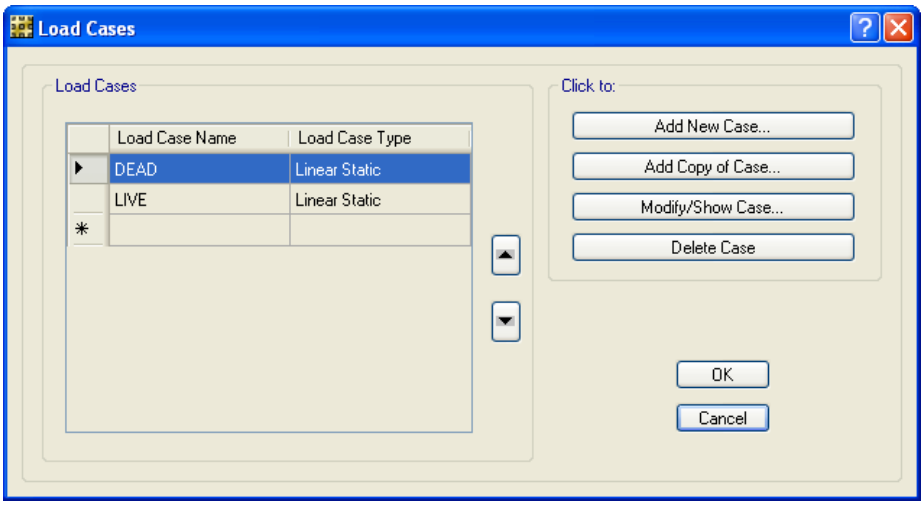


Figure 18 Load Cases form

B. With the *DEAD* Load Case Name highlighted, click the **Modify/Show Case** button to display the Load Case Data form shown in Figure 19. This data form changes based on the type of load case specified.

1. In the Load Case Type area, select *Static* from the drop-down list. Modal and Hyperstatic also are available as load case types.

2. In the Analysis Type area, select the *Linear* option. When working with a Static Load Case Type, the program offers the option to do Linear, Nonlinear (Allow Uplift), Nonlinear

(Cracked), or Nonlinear (Long Term Cracked) analysis. For our tutorial example, a Static, Linear analysis will be performed for DEAD and LIVE.

3. In the Loads Applied area, verify that the load pattern is *DEAD* with a scale factor of *1*.

4. Click the **OK** button to close the Load Case Data form.

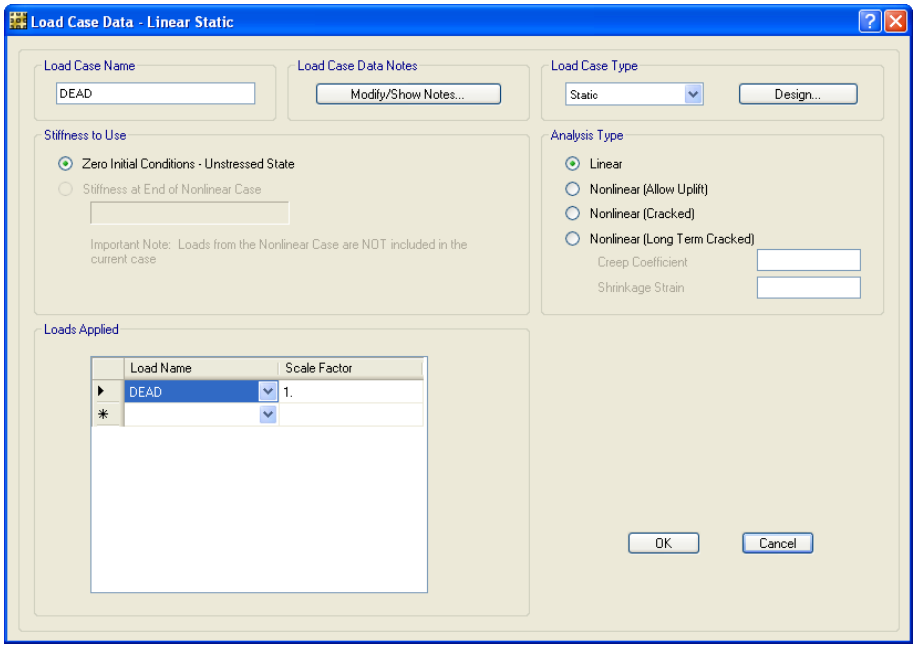


Figure 19 Load Cases Data form

C. Review the LIVE load case, if so desired, by selecting it and using the **Modify/Show Case** button as described for the DEAD load case.

D. Click the **OK** button to close the Load Cases form.

E. Click the **File menu > Save** command.

Step 5 Draw Objects

In this Step, slabs, columns with drops, beams, walls, and openings will be drawn.

Draw Slabs

Ensure that the Plan View is active (click anywhere in the display window; a window is active when the Display Title Bar, just below the horizontal toolbar, is highlighted). Now draw area objects to model the slab using the following Action Items.

A. Click the **Draw menu > Snap Options** command to display the Snap Options form shown in Figure 20.

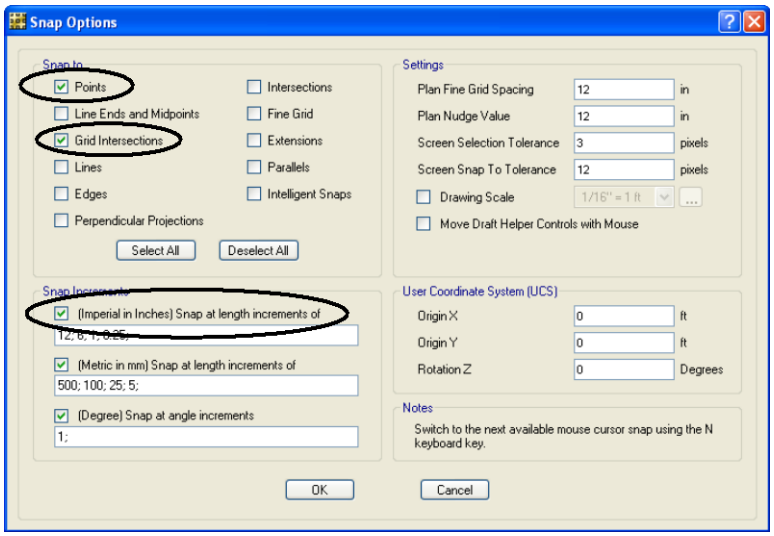


Figure 20 Snap Options form

B. In the *Snap to* area, make sure that the *Points* and *Grid Intersections* options are checked. These snap options will assist in accurately positioning objects when drawing.

C. In the *Snap Increments* area, make sure that the *(Imperial in Inches) Snap at length increments of* option is checked. When drawing slab edges, beams and walls, a dimension line will appear and the object will snap to the values specified in this edit box.

D. Click the **OK** button to close the Snap Options form.

E. Click the **Draw menu > Draw Slabs/Areas** command to access the Draw Slabs/Areas form shown in Figure 21. If the Draw Slabs/Areas form covers part of the model, click on the blue title bar, hold down the mouse button, and drag it out of the way.

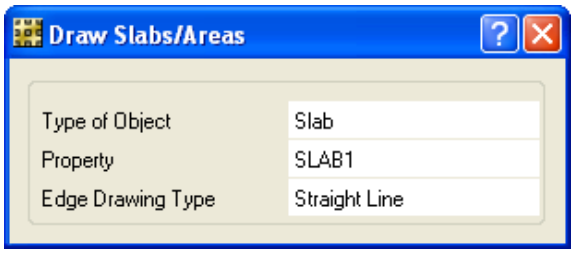


Figure 21 Draw Slabs/Areas form

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

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

H. Click in the Edge Drawing Type drop-down list and select *Straight Line*. Although not used in this project, slab edges also may be drawn with arcs and curves.

I. To draw the first corner of the slab, click once in the Plan View at the intersection of grid lines A and 7 (the cursor should display *Grid Point A 7* at the correct location). Then moving clockwise around the grid (note how the slab edge is dimensioned as you draw), click once at these grid intersections in this order to draw the outline of the slab: F7, F1, D1, D3, and A3. After clicking at grid A3, press the **Enter** key on the keyboard. The shaded slab object should now appear.

If you have made a mistake while drawing this object, click the **Select menu > Select > Pointer/Window** command to leave the

Draw mode and go to the Select mode. Then click the **Edit menu > Undo Area Add** command, and repeat Items E through I.

J. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Slabs/Areas command.

K. Select the slab by clicking on it anywhere. The status bar in the lower left-hand corner should show "1 Areas, 6 Edges selected." If you make a mistake in selecting, press the **Select menu > Clear Selection** command and try again.

L. Click the **Edit menu > Edit Areas > Expand/Shrink Areas** command to display the Expand/Shrink Areas form shown in Figure 22.

M. Type **9** into the Offset Value edit box. We will use this form to expand (a positive value expands) the slab by 9 inches at each corner to create the perimeter overhang that is needed to accommodate the width of the columns (1/2 of 18 inches).

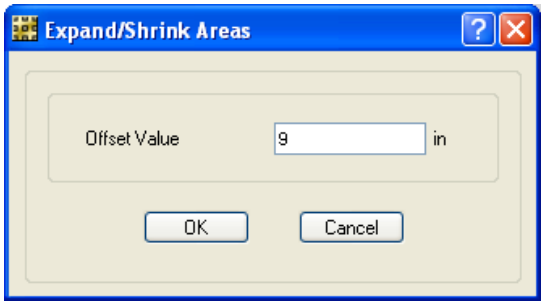


Figure 22 Expand/Shrink Areas form

N. Click the **OK** button to finish the slab.

O. Click the **File menu > Save** command to save your model.

Draw Columns

With the active window set as described in the preceding *Draw Slabs* section (i.e., Plan View window active and the snap to points and grid intersections enabled), use the following Action Items to draw columns.

A. Click the Draw menu > Draw Columns command to access the Draw Columns form shown in Figure 23.

B. Click in the Property Below drop-down list and select COL1. This is the property for the 18-inch by 18-inch column with a drop panel defined in Step 2.

C. Click in the Property Above drop-down list and select NONE. There are no columns above the slab.

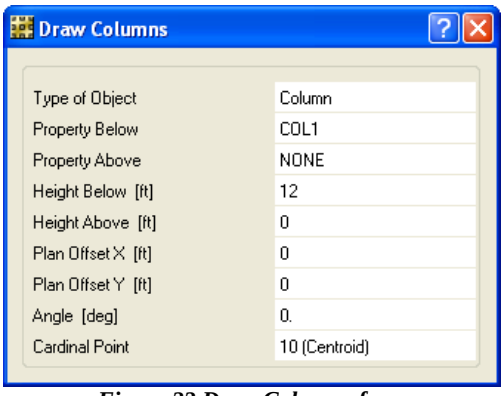


Figure 23 Draw Columns form

D. Click in the Height Below edit box and type **12**, and click in the Height Above edit box and type **0**. Note that the units are feet.

E. Click in the Cardinal Point drop-down list and select *10 (Centroid)*.

F. Locate the mouse cursor just above and to the left of grid intersection A6, hold down the left mouse button, and drag diagonally to just below and to the right of E3 and release the mouse button. Columns and drop panels should be placed at every grid intersec-

tion enclosed by the window just drawn. Note how the drop panels are trimmed at the slab edges.

G. Locate the mouse cursor just above and to the left of grid intersection D2, hold down the left mouse button, and drag diagonally to just below and to the right of E1 and then release. Again, columns and drop panels should be added to the grid intersections enclosed by the window.

H. Click in the Property Below drop-down list and select *COL-NODROP*. This is the property for the 18-inch by 18-inch column without a drop panel for use on the perimeter.

I. Left click once at each of the following grid locations to draw the perimeter columns: A7, B7, C7, D7, E7, F7, F6, F5, F4, F3, F2, and F1.

J. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Columns command.

K. Locate the mouse cursor just above and to the left of grid intersection C6, hold down the left mouse button, and drag diagonally to just below and to the right of D5 and release the mouse button. The status bar in the lower left-hand corner should show "24 Points, 4 Lines, 4 Areas, 16 Edges selected." If the selection is not correct, simply click the **Select menu > Clear Selection** command and try again.

L. Click the **Edit menu > Delete** command or press the **Delete** key on the keyboard to remove the columns enclosed in the window.

M. Click the **File menu > Save** command to save your model.

N. Click the **View menu > Set Default 3D View** command to display the model in 3D. Note how the columns extend below the slab.

O. Click the **View menu > Set Plan View** command to return to the Plan View before continuing the project.

Draw Walls

Similar to the preceding two sections, ensure that the Plan View is active and that the snap to points, grid intersections and the snap increments options are enabled. Now use the following Action Items to draw walls.

A. Click the **Draw menu > Draw Walls** command to access the Draw Walls form shown in Figure 24.

B. Click in the drop-down list opposite the Wall Property Below item and select *WALL1*. This is the wall property for the 12-inch-thick wall defined in Step 2.

C. Click in the Wall Property Above drop-down list and select *NONE*. There are no walls above the slab

D. Click in the Wall Height Below edit box and type **12**. Click in the Wall Height Above edit box and type **0**. Note that the units are feet.

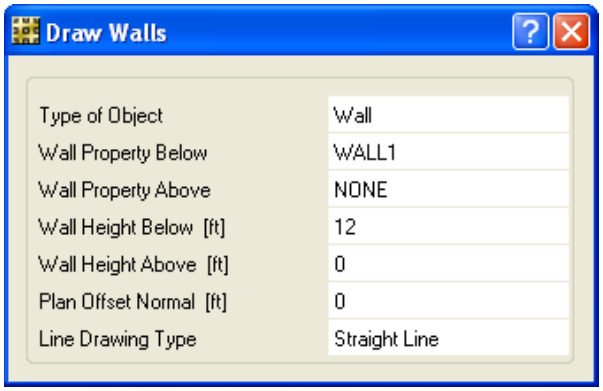


Figure 24 Draw Walls form

E. Click in the drop-down list opposite the Line Drawing Type item and select *Straight Line*. Although not used in this project, walls may also be drawn with arcs and curves.

F. Left click at grid intersection C6 to begin drawing the first wall. Draw the wall along grid line C (the wall is on grid line C if no angle measure is shown) past grid line 5 until the snap increments dimension line shows *30ft* and then click again. Hit the **Enter** key on the keyboard to complete the wall. This places a wall 30 feet long along grid line C.

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

G. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Walls command.

H. Hold down the **Shift** key on your keyboard and **right** click once at the end point of the wall just drawn. A selection list similar to the one shown in Figure 25 appears because multiple objects exist at that location. In this example, two point objects and two area objects exist at the same location. Note that the selection list will appear only when the **Shift** key is used with the click.

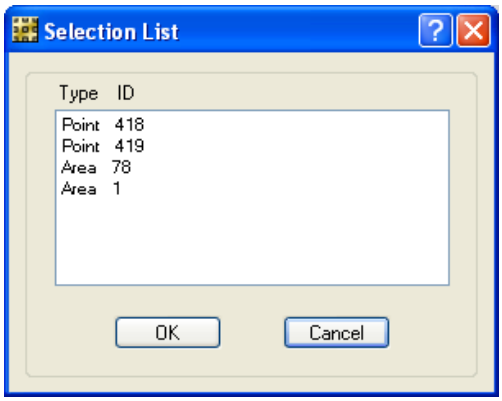


Figure 25 Selection List form

I. Highlight the first point object and click the **OK** button. Because a right click action initiated the selection process, a Point Object Information form similar to the one shown in Figure 26, will display.

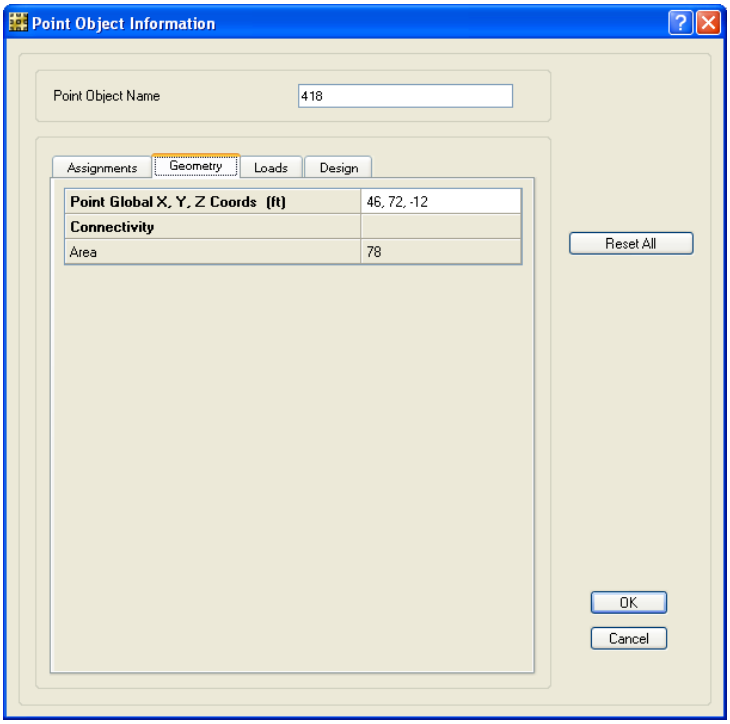


Figure 26 Point Object Information form

1. Click on the *Geometry* tab. Verify that the Point Global X and Y Coordinates are *46* and *72*, respectively. If the coordinates are not correct, delete and re-draw the wall following the instructions described in Item F above.

2. Click the **OK** button to leave the Point Object Information form.

J. Click the **Draw menu > Draw Walls** command to access the Draw Walls form.

K. For the next wall, located along grid line 6, left click at grid intersection C6 to begin drawing the wall and at intersection D6 (the snap increment dimension line should show *20ft*) to designate the end of the wall.

L. With the draw mode still active, draw the last wall along grid line D past grid line 5 until the snap increments dimension line shows 30 and then click again. Hit the **Enter** key on the keyboard to complete the wall. This wall should be parallel to the first wall drawn.

M. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Walls command. Your model should look similar to Figure 27.

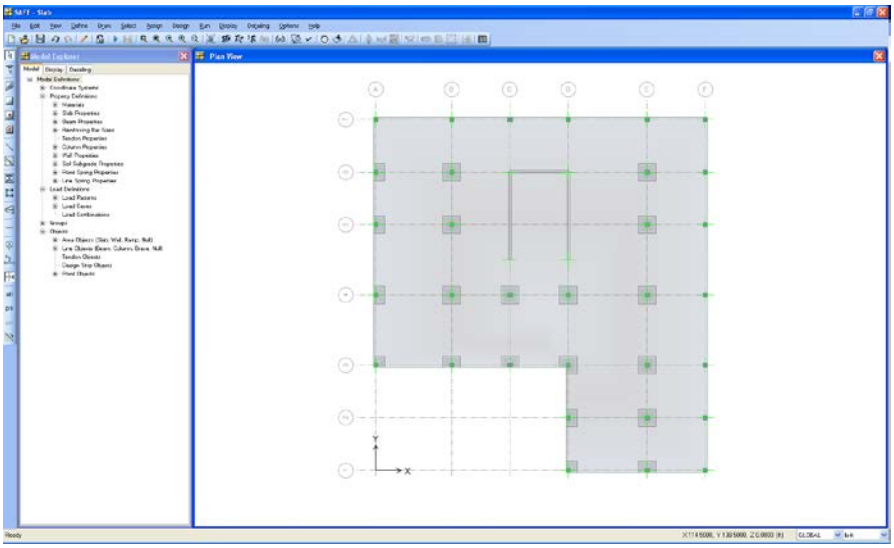


Figure 27 The model after drawing columns and walls

Draw Beams

Similar to the previous sections, ensure that the Plan View is active and the snap to points and grid intersections features are enabled. Draw the beams as follows.

A. Click the **Draw menu > Draw Beams/Lines** command to access the Draw Beams/Lines form shown in Figure 28.

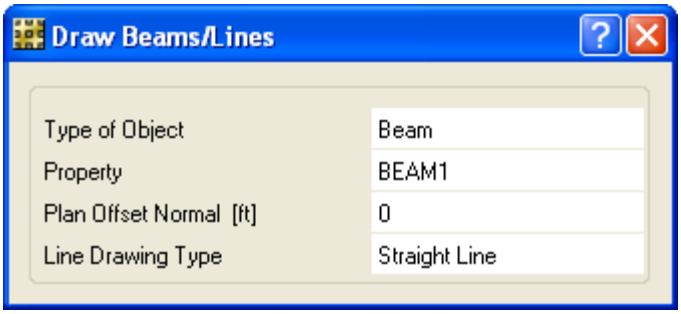


Figure 28 Draw Beams/Lines form

B. Click in the drop-down list opposite the Property item and select *BEAM1*. Recall that BEAM1 is the 18-inch by 24-inch beam defined in Step 2.

C. Click in the Line Drawing Type drop-down list and select *Straight Line*.

D. Left click once at the grid intersection A7. Click again at grid intersections F7, followed by F1. Although only one beam 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 menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Beams command.

F. Click the **File menu > Save** command to save your model.

Draw Openings (Area Objects)

Similar to the previous sections, ensure that the Plan View is active and the snap to points and grid intersections features are enabled. Draw an area object to model the opening as follows:

A. Click the **Draw menu > Draw Rectangular Slabs/Areas** command to display the Draw Rectangular Slabs/Areas form shown in Figure 29.

B. Click once in the drop-down list opposite the Type of Object item and select *Opening*.

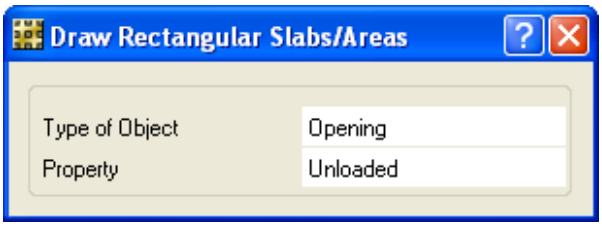


Figure 29 Draw Rectangular Slabs/Areas form

C. Left click at the intersection of grid lines C and 6, and while holding the left mouse button down, move diagonally down to the wall end point located along grid line D between grids 4 and 5 and release the button. An area object with different shading should appear, indicating that an opening for the stairs has been drawn. An opening takes priority over an object with assigned slab properties when the program determines the stiffness formulation.

D. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Rectangular Slabs/Areas command.

E. Click the **File menu > Save** command to save your model.

Step 6 Add Design Strips

In this step, design strips will be added to the model. Design strips determine how reinforcing will be calculated and positioned in the slab. Forces are integrated across the design strips and used to calculate the required reinforcing for the selected design code. Typically design strips are positioned in two principal directions: Layer A and Layer B.

Similar to the previous sections, ensure that the Plan View is active and the snap to points and grid intersections features are enabled. Add design strips to the model as follows:

A. Click the **Edit menu > Add/Edit Design Strips > Add Design Strips** command to display the Add Design Strips form shown in Figure 30.

B. In the Options area, select the *Add Design Strips Along Cartesian Grid Lines* option. Make sure that the *Include Middle Strip* option is checked

C. In the Parameters area, click in the Grid Direction drop-down list and select *X*.

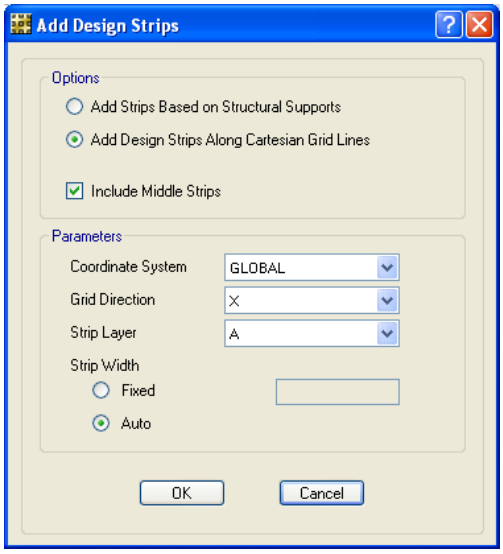


Figure 30 Add Design Strips form

D. Select *A* from the Strip Layer drop-down list.

E. Select the *Auto* option. The added design strips will automatically adjust their width to align with adjacent strips.

F. Click the **OK** button to leave the Add Design Strips form. Design strips in the X-axis direction should now appear as solid lines.

G. Left click on the design strips that lie below grid line 3 to select them; the status bar in the lower left-hand corner should show "4 Design Strips selected." If the selection is not correct, simply click the **Select menu > Clear Selection** command and try again.

H. Left click on the slab (anywhere except at a column, drop panel, beam or design strip location) to select it; the status bar in the lower left-hand corner should now show "1 Areas, 6 Edges, 4 Design Strips selected."

I. Left click at the left ends of the 4 selected design strips; the status bar in the lower left-hand corner should now show "4 Points, 1 Areas, 6 Edges, 4 Design Strips selected."

J. Click the **Edit menu > Align Points/Lines/Edges** command to display the Align Points/Lines/Edges form shown in Figure 31.

K. Select the Trim Line/Edge/Tendon/Strip Objects option.

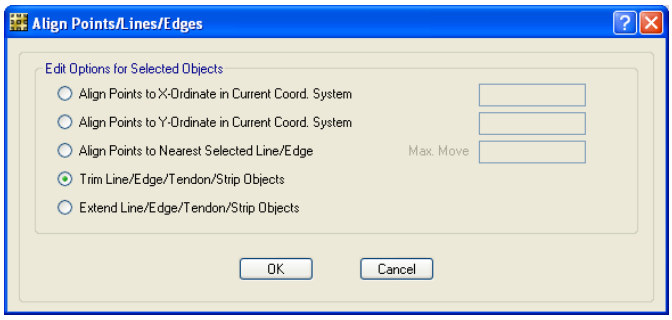


Figure 31 Align Points/Lines/Edges form

L. Click the **OK** button to leave the Align Points/Lines/Edges form. The design strips below grid line 3 should now be trimmed to the edge of the slab.

M. Left click on the design strip that lies on grid line 5 to select it; the status bar in the lower left-hand corner should show "1 Design Strips selected."

N. Press the **Delete** key on the keyboard to remove the selected design strip from the model.

O. Click the **Draw menu** > **Draw Design Strips** command to display the Draw Design Strips form shown in Figure 32.

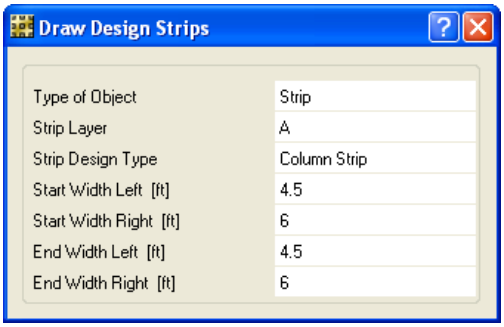


Figure 32 Draw Design Strips form

P. Select *A* from the Strip Layer drop-down list.

Q. Select *Column Strip* from the Strip Design Type drop-down list.

R. Type **4.5** into the Start Width Left edit box.

S. Type **6** into the Start Width Right edit box.

T. Type **4.5** into the End Width Left edit box and type **6** into the End Width Right edit box.

U. Left click at grid intersection A5 and at C5, and then click the right mouse button to stop drawing.

V. Left click at grid intersection D5 and at F5.

W. Press the **Esc** key on the keyboard to leave the Draw command.

X. Click the **Edit menu** > **Add/Edit Design Strips** > **Add Design Strips** command to display the Add Design Strips form.

Y. Select *Y* from the Grid Direction drop-down list.

Z. Click in the Strip Layer edit box and select *B* from the drop-down list.

AA. Click the **OK** button to leave the Add Design Strips form. Design strips in the Y-axis direction should now appear as solid lines.

BB. Left click on the design strips that lie to the left of grid line D to select them; the status bar in the lower left-hand corner should show "6 Design Strips selected."

CC. Left click on the slab (anywhere except at a column, drop panel, beam or design strip location) to select it; the status bar in the lower left-hand corner should now show "1 Areas, 6 Edges, 6 Design Strips selected."

DD. Left click at the bottom ends of the selected design strips; the status bar in the lower left-hand corner should now show "6 Points, 1 Areas, 6 Edges, 6 Design Strips selected."

EE. Click the **Edit menu** > **Align Points/Lines/Edges** command to display the Align Points/Lines/Edges form.

FF. Select the Trim Line/Edge/Tendon/Strip Objects option.

GG. Click the **OK** button to leave the Align Points/Lines/Edges form. The Y direction design strips to the left of grid line D should now be trimmed to the edge of the slab.

The trimming of the design strips was done for display purposes only; the program will automatically ignore the portion of a design strip that extends beyond a slab edge.

HH. Click the **File menu** > **Save** command to save your model.

Step 7 Set Display Options

In this Step, the set display options will be used to alter the objects displayed.

A. Click the **View menu > Set Display Options** command. When the Set Display Options form displays, uncheck the *Design Strip Layer A* and *Design Strip Layer B* check boxes in the Items Present in View area, as shown in Figure 33. This action will turn off the display of the design strips.

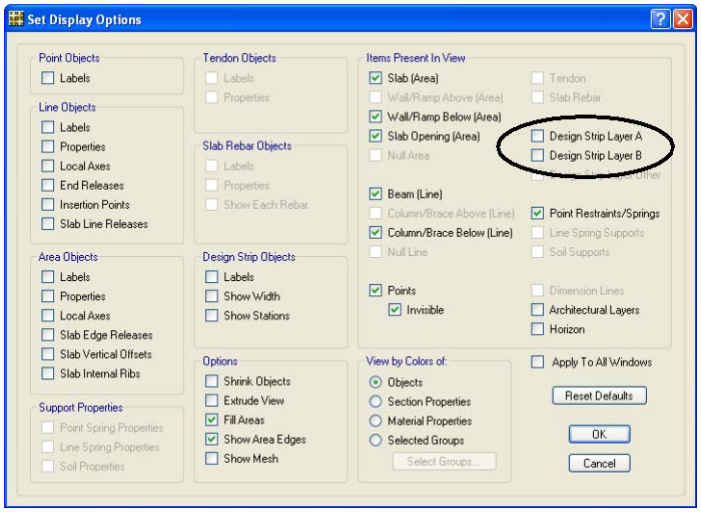


Figure 33 Set Display Options form

B. Click the **OK** button to accept the changes, and the model now appears as shown in Figure 34.

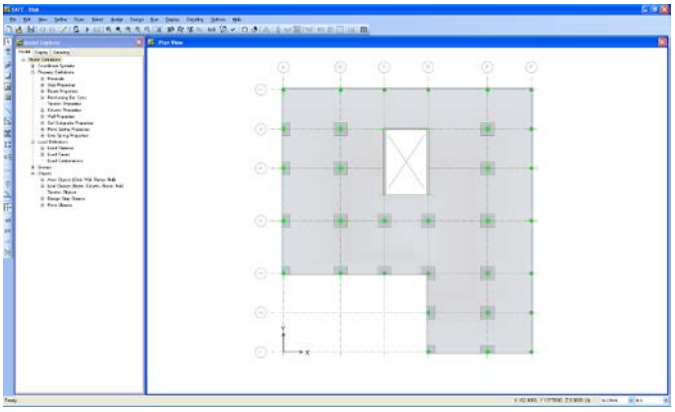


Figure 34 Model after all objects have been drawn

Step 8 Assign Loads

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

A. Select the slab by clicking on it anywhere that is ***not*** a beam, wall, column, drop panel or opening. The status bar in the lower left-hand corner should show "1 Areas, 6 Edges selected." If you make a mistake in selecting, click the **Select menu > Clear Selection** command, and try again.

B. Click the **Assign menu > Load Data > Surface Loads** command to access the Surface Loads form shown in Figure 35.

C. If it is not selected already, select DEAD from the Load Pattern Name drop-down list.

D. Select *Gravity* from the Direction drop-down list in the Load Direction area.

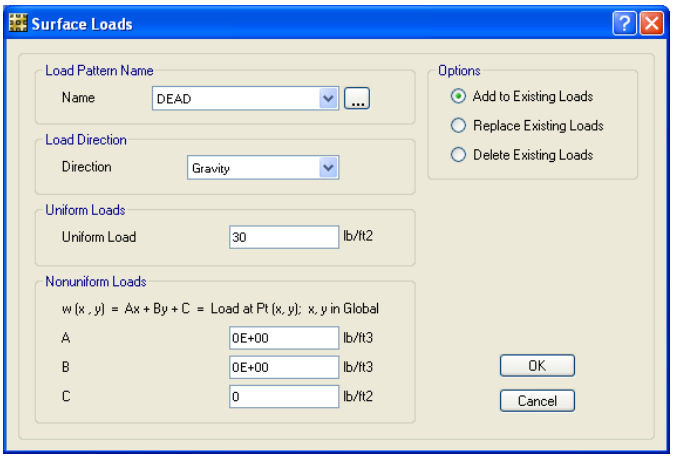


Figure 35 Surface Loads form

E. In the Uniform Loads area, type **30** in the Uniform Load edit box.

Note: Additional load patterns may be defined by clicking on the "**…**" button next to the load pattern name. A "**…**" button returns you to the form used to define the item in the adjacent drop-down list or edit box, which in this case is the Load Patterns form.

F. 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.

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

H. Click the **Assign menu > Load Data > Surface Loads** command to again access the Surface Loads form.

I. Select *LIVE* from the Load Pattern Name drop-down list.

J. Type **50** in the Uniform Load edit box in the Uniform Loads area.

K. 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.

L. To review the assignments to the slab, **right** click on the slab anywhere that is ***not*** a beam, wall, column, drop-panel or opening to access the Slab-Type Area Object Information form shown in Figure 36.

M. Select the *Loads* tab and note that the DEAD Load Pattern has a Load Value of 30lb/ft2, and that the LIVE Load Pattern has a Load Value of 50lb/ft2.

N. Click the **OK** button to close the Slab-Type Area Object Information form.

O. Click the **File menu > Save** command to save your model.

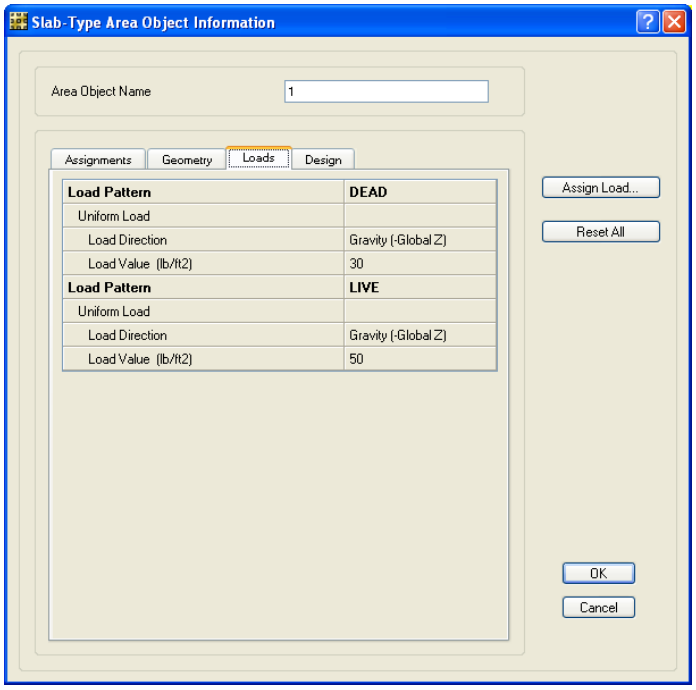


Figure 36 Slab-Type Area Object Information form

Step 9 Run the Analysis and Design

In this Step, the analysis and design will be run.

A. Click the **Run menu > Run Analysis & Design** command to start the analysis. The program will create the analysis model from your object-based SAFE model and will display information in the status bar in the lower left-hand corner as the analysis and design proceeds. Additional information about the run may be accessed at a later time using the **File menu > Show Input/Output Text Files** command and selecting the filename with a *.LOG* extension.

B. When the analysis and design are finished, the program automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the **Options menu > Lock/Unlock Model** icon appears depressed. Locking the

model prevents any changes to the model that would invalidate the analysis results.

Step 10 Graphically Review the Analysis Results

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

A. Click the **View menu > Set Default 3D View** command to display the deformed shape for the DEAD load case in 3D.

B. Click the **Start Animation** button in the lower right-hand corner of the display to animate the deformed shape. Speed of the animation may be adjusted by using the slider control adjacent to the button. Click the **Stop Animation** button to end the animation.

C. Click the **Display menu > Show Deformed Shape** command to access the Deformed Shape form shown in Figure 37.

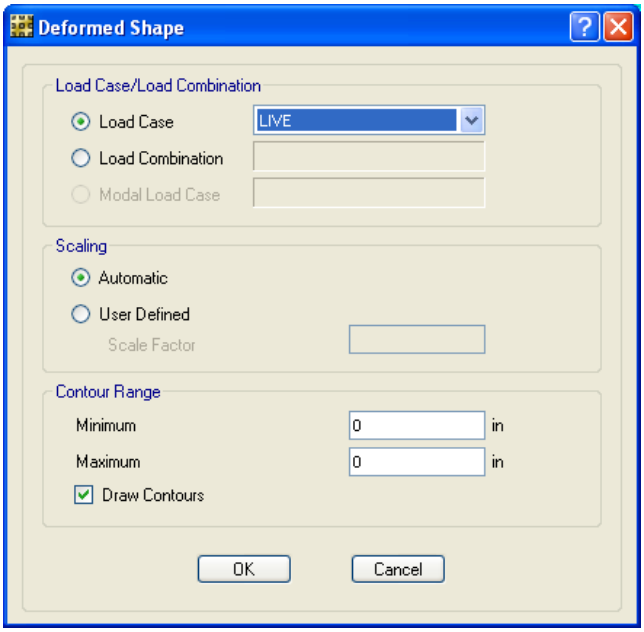


Figure 37 Deformed Shape form

D. In the Load Case/Load Combination area, select the *Load Case* option.

E. Select *LIVE* from the Load Case drop-down list.

F. Select the *Automatic* option in the Scaling area.

G. Check the *Draw Contours* checkbox in the Contour Range area.

H. Click the **OK** button to generate a 3-D deformed shape with contours for the LIVE load case.

I. Click the **Display menu > Show Slab Forces/Stresses** command to bring up the Slab Forces/Stresses form shown in Figure 38.

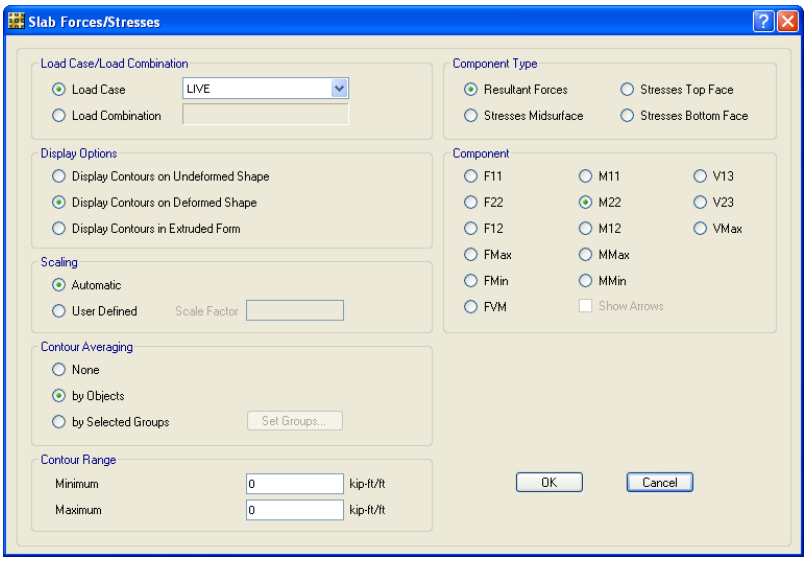


Figure 38 Slab Forces/Stresses form

J. Select *LIVE* from the Load Case drop-down list.

K. Select the *Resultant Forces* option in the Component Type area.

L. Select the *M22* option in the Component area.

M. Select the *Display Contours on Deformed Shape* option in the Display Options area.

N. Click the **OK** button to generate the moment diagram shown in Figure 39.

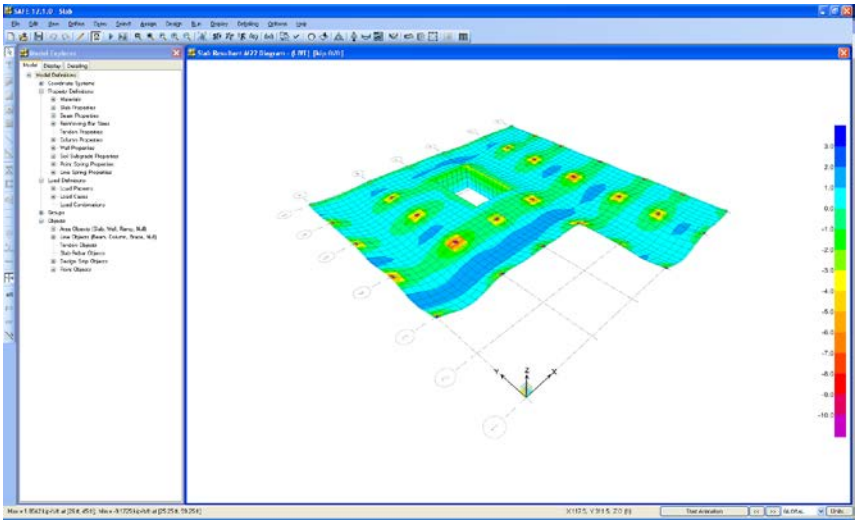


Figure 39 M22 Moment Diagram

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

O. Click the **Display menu > Show Undeformed Shape** command to clear the display of the moment diagram.

P. Click the **View menu > Set Plan View** command to return to the Plan View.

Step 11 Design Display

In this Step, design results for the slab and beams will be displayed. Note that the design was run along with the analysis in Step 9. Design results are for the ACI 318-08 code, which was selected in Step 1. Design preferences may be reviewed or changed by going to the **Design menu >**

Design Preferences command (some design preferences are also set on the section property data forms); be sure to re-run the analysis and design (Step 9) if changes to the preferences are made.

A. Click the **Display menu > Show Slab Design** command to access the Slab Design form shown in Figure 40.

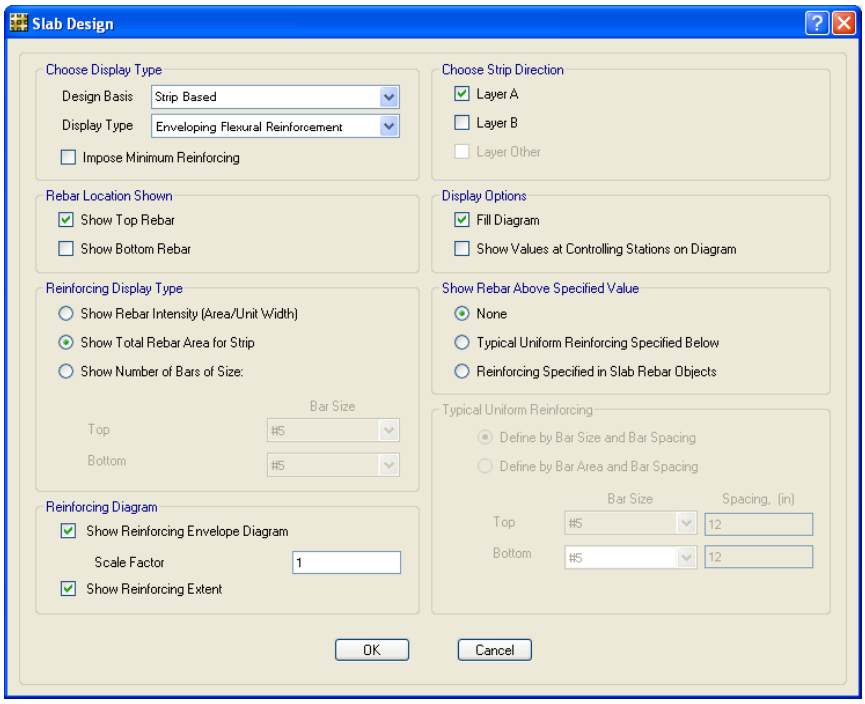


Figure 40 Slab Design form

B. In the Choose Display Type area, select *Strip Based* from the Design Basis drop-down list and *Enveloping Flexural Reinforcement* from the Display Type drop-down list.

C. In the Choose Strip Direction area, check the *Layer A* checkbox and uncheck the *Layer B* checkbox. This will display the design results in the Layer A (X) direction only.

D. In the Rebar Location Shown area, check the *Show Top Rebar* checkbox and uncheck the *Show Bottom Rebar* checkbox.

E. In the Reinforcing Display Type area, select the *Show Total Rebar Area for Strip* option.

F. Click the **OK** button to leave the Slab Design form and display the slab design results. The top flexural reinforcing required in the Layer A direction is displayed for both column and middle strips.

Positioning the cursor at any location on a Layer A design strip causes the required top and bottom reinforcing values to be displayed at the cursor and in the lower left corner of the window.

G. To view the required reinforcing in the other direction, click the **Display menu > Show Slab Design** command to display the Slab Design form.

H. In the Choose Display Type area, select *Finite Element Based* from the Design Basis drop-down list. This option displays the required reinforcing calculated on an element-by-element basis as intensity contours - integration across the defined design strips is not performed.

I. In the Reinforcing Direction and Location area, select the *Direction 2 – Bottom Rebar* option. Direction 2 refers to the object local axis 2 direction.

J. In the Show Rebar Above Specified Value area, select the *None* option.

K. Click the **OK** button to leave the Slab Design form and display the slab design results for the local axis 2 direction. The view will be updated to that shown in Figure 41. Again, positioning the cursor anywhere on the slab will result in the display of the reinforcing values at the cursor and in the lower left-hand corner of the SAFE window.

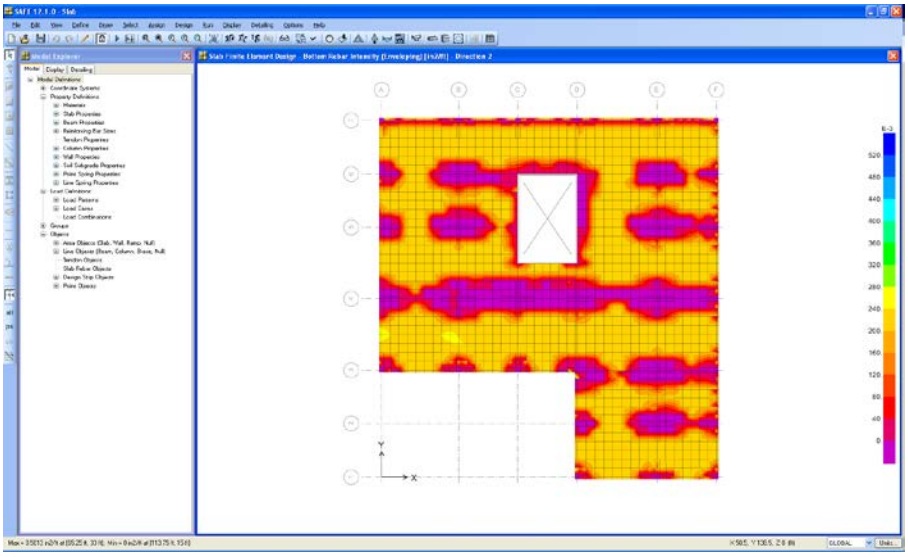


Figure 41 Direction 2 reinforcing

L. To view the required reinforcing in the beams, click the **Display menu > Show Beam Design** command. The Beam Design form shown in Figure 42 displays.

M. In the Choose Display Type area, select *Longitudinal Rebar* from the Display Type drop-down list and *Flexure* from the Rebar Type drop-down list.

N. Review the other selected options and then click the **OK** button to close the Beam Design form.

The 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 and at the cursor.

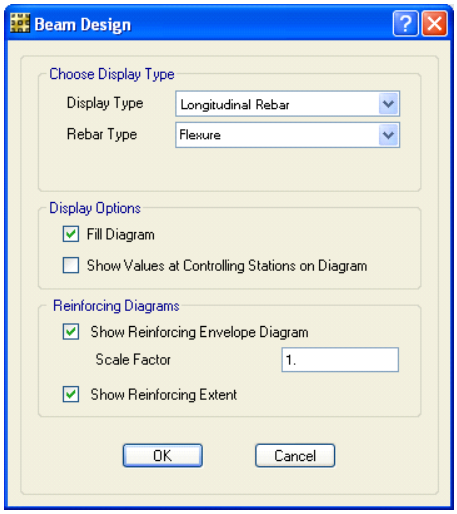


Figure 42 Beam Design form

O. **Right** click on a beam to display the Design Details form shown in Figure 43.

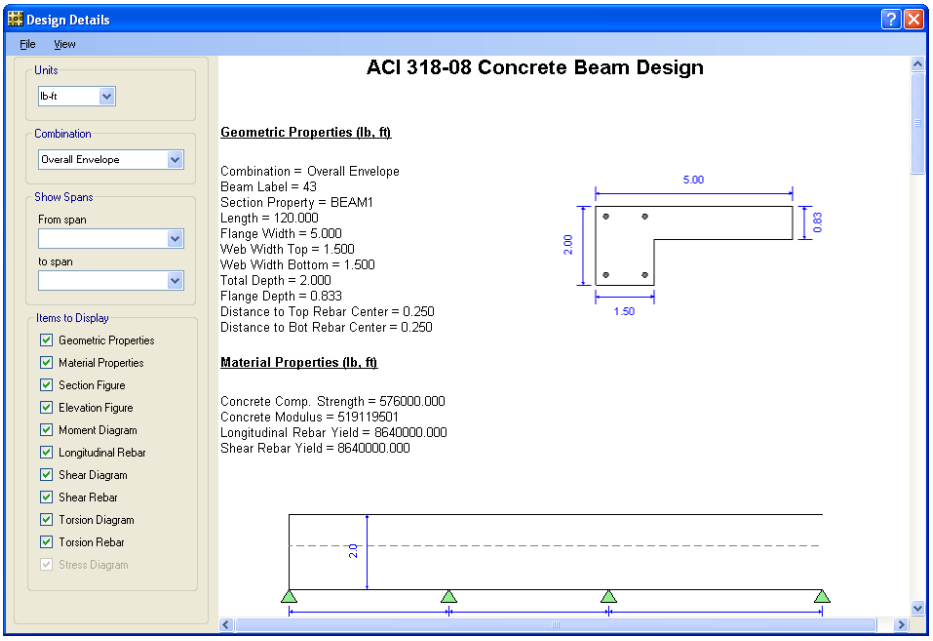


Figure 43 Design Details form

Scroll through the form to see all of the design information associated with the selected beam. Click the **X** button in the top-right corner of the form to close the form.

Step 12 Run Detailing

In this Step, detailing will be run and displayed. Detailing may be run only after analysis and design are complete.

A. Click the **Detailing menu > Detailing Preferences** command to display the Detailing Preferences form shown in Figure 44. Use this form to set the regional standards, to control how dimensioning is displayed, to manage reinforcing bar notation, and to select the units for material quantity takeoffs.

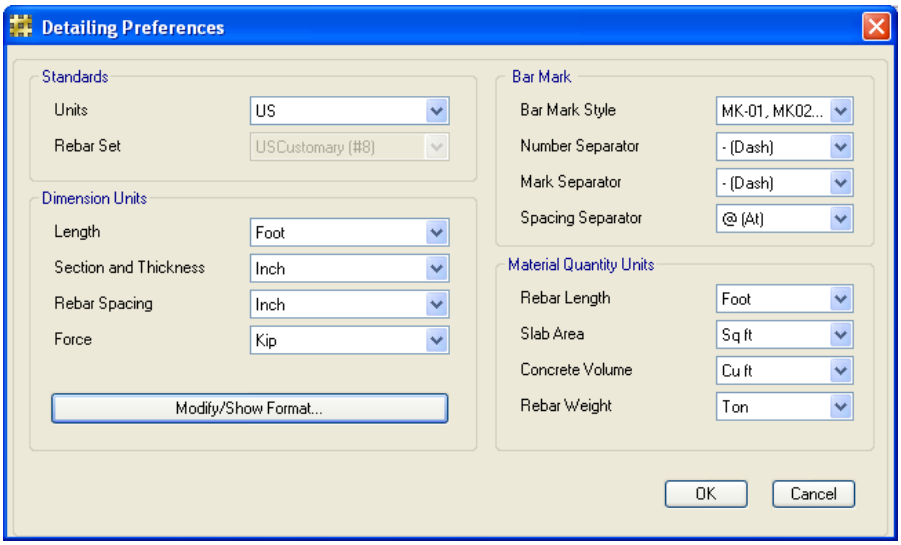


Figure 44 Detailing Preferences form

B. Review the settings on this form (we will accept the default selections), and then click the **OK** button to close the form.

C. Click the **Detailing menu > Slab/Mat Reinforcing Preferences** to display the Slab/Mat Detailing Preferences form shown in Figure 45.

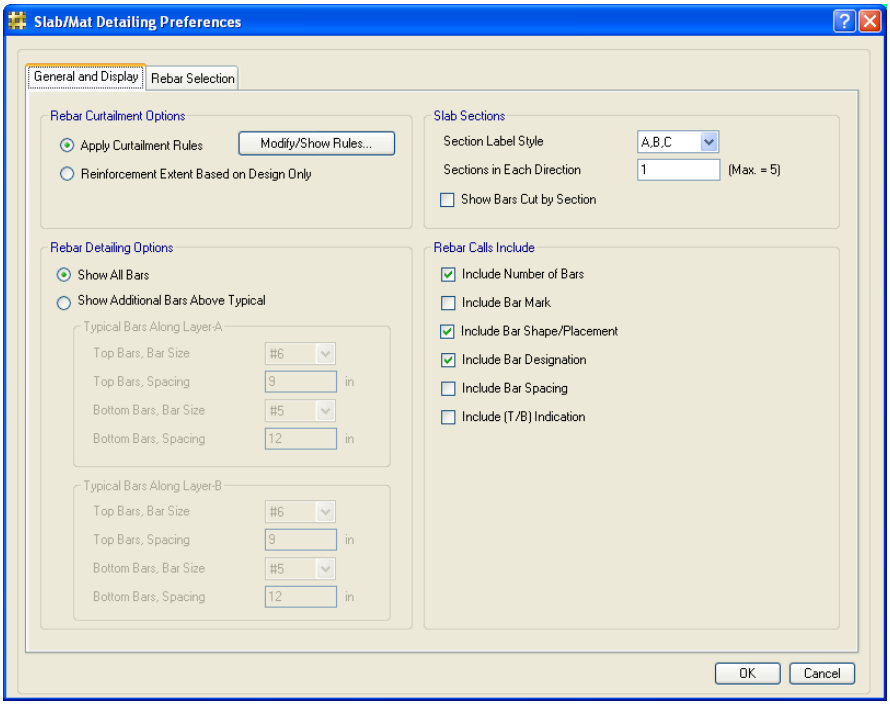


Figure 45 Slab/Mat Detailing Preferences form

D. Click the *General and Display* tab. On this tab review or alter the rebar curtailment, detailing and callout options, as well as set how sections should be cut. We will accept the default settings.

E. Click the *Rebar Selection* tab and review or change the rebar selection rules, preferred sizes, minimums and reinforcing around openings. We will accept the default settings.

F. Click the **OK** button to accept the selections and close the form.

G. Click the **Detailing menu > Drawing Sheet Setup** command to display the Drawing Sheet Setup form. The sheet size, scales, title block and text sizes can be reviewed and changed using this form. We will accept the default settings.

H. Click the **OK** button to close the form.

I. Review the line thicknesses and styles by clicking the **Detailing menu > Drawing Format Properties** command.

J. Click the **OK** button to accept the selections and close the form.

K. Now that the detailing preferences and drawing setup options have been reviewed, click the **Run menu > Run Detailing** command to generate the detailing drawings. A framing plan is displayed when detailing is complete.

L. Click the **Detailing menu > Show Detailing** command to access the Display Detailing Item form shown in Figure 46.

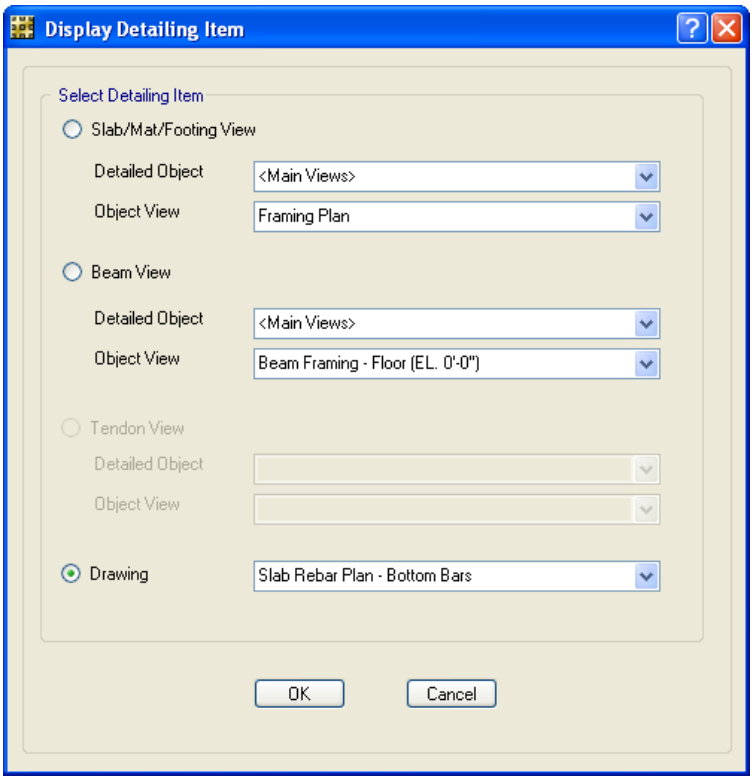


Figure 46 Display Detailing Item form

M. Select the *Drawing* option.

N. Select *Slab Rebar Plan – Bottom Bars* from the Drawing drop-down list.

O. Click the **OK** button to leave the Display Detailing Item form and display the selected drawing shown in Figure 47.

P. Clicking on the *Detailing* tab in the Model Explorer and expanding the Views and Drawing Sheets trees also provides access to detailing drawings and component views.

Q. Click the **Display menu > Show Undeformed Shape** command to return to the model.

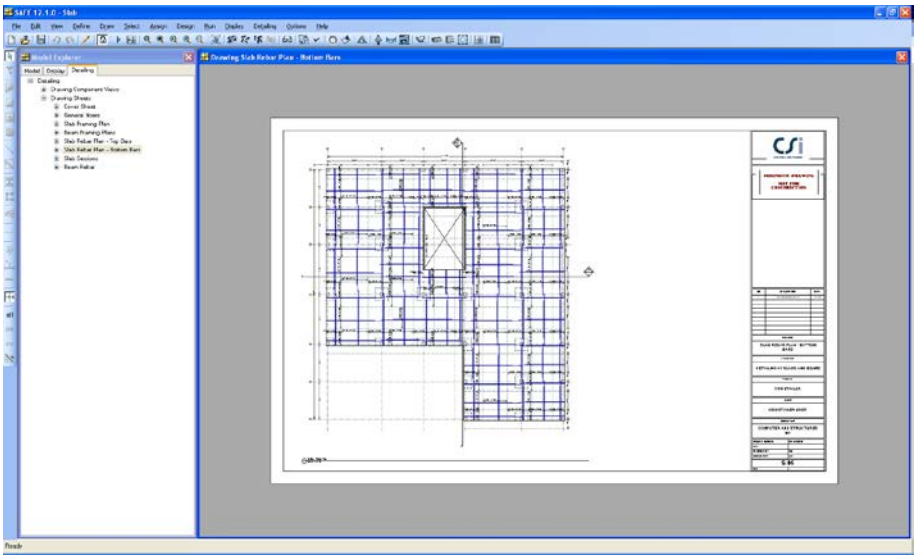


Figure 47 Slab Rebar Plan

Step 13 Create Report

In this Step, a report describing model input and output results will be created.

A. Click the **File menu > Report Setup** command to display the Report Setup Data form.

B. In the Report Output Type area, be sure that the *RTF File* option is selected.

C. In the Report Items area, uncheck the *Include Hyperlinked Contents* checkbox.

D. Click the **OK** button to leave the Report Setup Data form.

E. Click the **File menu > Create Report** command to display the Microsoft Word Rich Text File Report form.

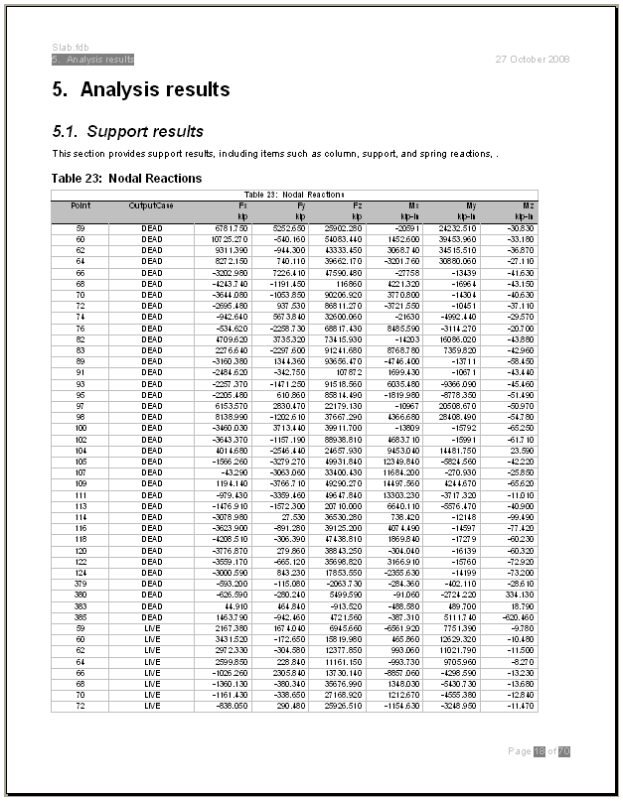
F. Type **Slab** in the File name edit box and click the **Save** button. A report, with a cover similar to that shown in Figure 48 should be displayed in your word processor, and will be saved to your hard disk.

Figure 48 Cover Page for Report

G. Scroll through the report to find tables that list geometry and properties, analysis results and design information, such as that shown if Figure 49.

H. Close your word processor and return to the SAFE program.

I. Click the **File menu > Save** command to save your model one last time.



Slab.fdb
5. Analysis results 27 October 2008

5. Analysis results

5.1. Support results

This section provides support results, including items such as column, support, and spring reactions, .

Table 23: Nodal Reactions

Table 23: Nodal Reactions

Point	OutputCase	F1 kip	F2 kip	F3 kip	M1 kip-in	M2 kip-in	M3 kip-in
59	DEAD	6781.750	8252.690	25902.260	-2091	24232.510	-305.930
60	DEAD	10725.270	-540.160	54083.440	1452.600	39453.960	-33.180
62	DEAD	9311.390	-944.300	43333.450	3068.740	34515.510	-36.870
64	DEAD	827.150	740.110	39662.170	-3201.760	30880.060	-27.110
66	DEAD	-3262.980	7226.410	47590.480	-2758	-15439	-41.630
68	DEAD	-4243.740	-1191.450	116860	422.1320	-16964	-43.150
70	DEAD	-3644.080	-1053.850	90206.920	377.0800	-14304	-40.630
72	DEAD	-2695.480	937.530	86811.270	-3721.550	-10451	-37.110
74	DEAD	-942.640	667.3840	32600.060	-21630	-4992.440	-29.570
76	DEAD	-534.620	-2298.730	68917.430	8485.590	-3114.270	-20.700
82	DEAD	4709.620	3735.320	73415.930	-1423	16086.020	-43.880
83	DEAD	2276.640	-2297.600	91241.680	8568.780	7359.820	-42.960
89	DEAD	-3160.380	1344.360	93656.470	-4746.400	-13711	-58.450
91	DEAD	-2484.620	-342.750	107857.2	1699.430	-1061	-43.440
93	DEAD	-2257.370	-1471.250	91518.560	6035.480	-9366.090	-45.490
95	DEAD	-2295.480	610.860	88514.490	-1819.980	-8778.350	-51.490
97	DEAD	6153.570	2830.470	22179.130	-10967	20508.670	-50.970
98	DEAD	8138.990	-1202.610	37667.290	4366.680	28408.490	-54.780
100	DEAD	-3460.030	3713.440	39911.700	-13809	-15792	-66.250
102	DEAD	-3643.370	-1157.190	88938.810	4683.710	-15991	-61.710
104	DEAD	4014.680	-2546.440	24657.930	9453.040	14481.750	23.590
105	DEAD	-1566.260	-3279.270	49931.840	12349.840	-5824.560	-42.220
107	DEAD	-43.290	-3063.060	33400.430	11684.200	-270.930	-25.800
109	DEAD	1194.140	-3766.710	49290.270	14497.560	624.4670	-65.620
111	DEAD	-979.430	-3359.460	49647.840	13303.230	-3717.320	-11.010
113	DEAD	-1476.910	-1572.300	20710.000	6640.110	-5576.470	-40.900
114	DEAD	-3078.980	27.530	36530.260	738.420	-12148	-99.490
116	DEAD	-3623.900	-891.260	39125.200	407.4490	-14597	-77.420
118	DEAD	-4286.510	-306.390	47438.810	1869.840	-17279	-66.230
120	DEAD	-3776.870	279.860	38843.250	-304.040	-16139	-66.320
122	DEAD	-3559.170	-665.120	35696.820	3166.910	-15760	-72.920
124	DEAD	-3000.590	843.230	17853.550	-2355.630	-14199	-73.200
379	DEAD	-593.200	-115.080	-2063.730	-284.360	-402.110	-25.610
380	DEAD	-626.590	-280.240	5499.590	-91.690	-2724.220	334.130
383	DEAD	44.910	464.840	-913.520	-488.580	489.700	18.790
385	DEAD	1463.790	-942.460	4721.560	-387.310	5111.740	-620.460
59	LIVE	2167.380	167.4040	6945.660	-6561.920	7751.390	-9.780
60	LIVE	3431.520	-172.650	18519.980	465.860	12629.320	-10.480
62	LIVE	2972.330	-304.580	12377.850	983.060	11021.790	-11.500
64	LIVE	2599.850	228.840	11161.150	-993.730	9705.960	-8.270
66	LIVE	-1026.260	2305.840	13730.140	-8857.060	-4298.590	-13.230
68	LIVE	-1360.130	-380.340	35676.990	1348.030	-5430.730	-13.680
70	LIVE	-1161.430	-338.650	27168.920	1212.670	-4555.380	-12.840
72	LIVE	-838.650	290.480	28926.510	-1164.630	-3248.980	-11.470

Page 16 of 70

Figure 49 Typical Report information

Congratulations! You've successfully created, analyzed, designed, detailed, and reviewed a SAFE reinforced concrete model.

Post-Tensioned Concrete

P/T Example

The intent of this tutorial is to give you hands-on experience via step-by-step instructions on how to use SAFE to model, analyze, design and detail post-tensioned concrete slabs. Fundamentals of the model creation process are identified and various model construction techniques are introduced. As you complete the tutorial, you will build the model shown in Figure 50.

The Project

The tutorial project is an irregularly shaped post-tensioned concrete slab as might be used for residential construction, with overall dimensions of 108 feet by 92 feet. Several openings exist in the interior for stair access. The 8-inch thick slab is supported by a combination of 8-inch-thick walls and 12-inch-square columns. The floor-to-floor height is 10 feet. The model will be analyzed for a uniform dead load (partitions + mechanical) of 35 pounds per square foot (psf) plus the self weight of the structure, and a live load of 50 psf.

Concrete Materials:

Concrete strength, f'c = 5000 psi

Unit weight of concrete = 150 pcf

Prestressing Tendons:

Unbonded mono-strand

½" diameter strand; 0.153 square-inch area

f_{se} = 175.5 ksi (216 ksi @ stressing – 40.5 ksi losses)

Reinforced Concrete:

f_y = 60ksi

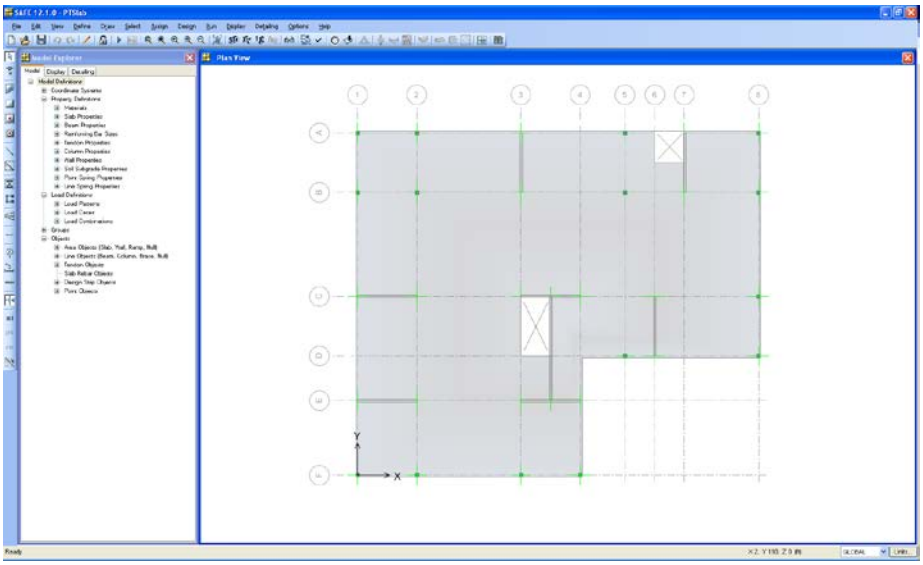


Figure 50 The Project Model

Navigating Through SAFE

The SAFE program provides the user with two principal ways to navigate through program commands: menu commands or toolbar buttons. All commands are available through the main menu bar (e.g., **Draw menu > Draw Columns**), and a majority of the menu commands are also available as buttons on toolbars (e.g., **Draw Columns**, ). The availability of a button on the toolbar is indicated in the menus by the existence of an icon to the left of the command, as shown in Figure 51.

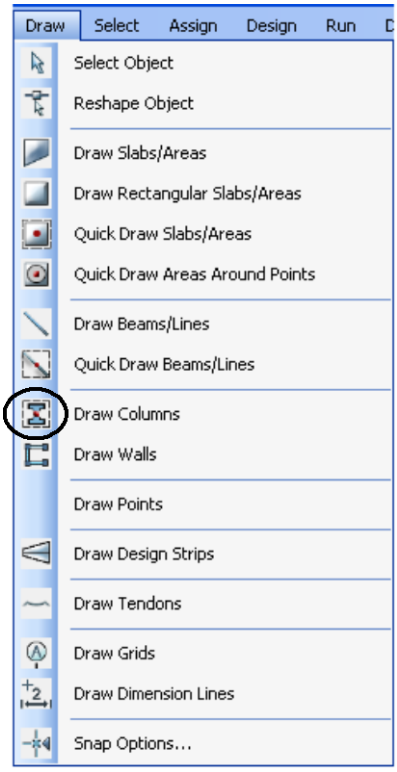


Figure 51 Draw Menu

In this tutorial the reference to various commands will be given using the narrative description, i.e., **Draw menu > Draw Column** in lieu of the associated button.

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 to demonstrate how to construct a model from scratch. However, as a general rule, we highly recommend using templates to start models whenever possible because they provide a quick, easy way of generating a model. Consult the SAFE Help topics for information about templates.

Define the Grid

A. Click the **File menu** > **New Model** command to access the New Model Initialization form shown in Figure 52. This form is used to specify the starting point of the model creation: a Blank screen, a screen with a Grid Only, or one of eight templates. Default units may also be selected here, along with the design code and preferences.

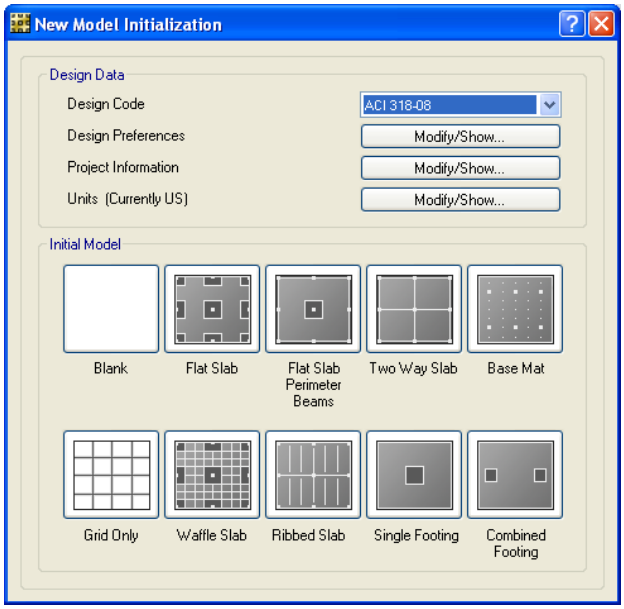


Figure 52 New Model Initialization form

B. In the Design Data area, select *ACI 318-08* from the Design Code drop-down list.

C. In the Design Data area, verify that the Units are set to *Currently US*; if they are not, click the **Modify/Show** button and select the U.S. Defaults on the Units form.

D. In the Initial Model area, click the **Grid Only** button to display the Coordinate System Definition form shown in Figure 53. This form is used to specify the number of grids and spacing in each direction. It is important that the grid be defined such as to accu-

rately represent the geometry of the structure, so it is advisable to spend time carefully planning the number and spacing of the grid lines.

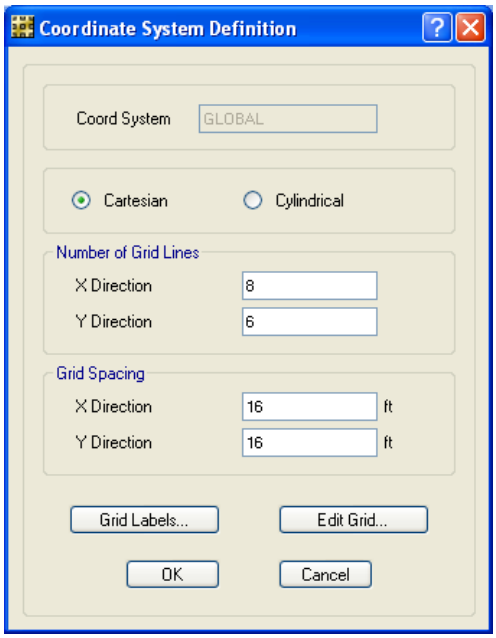


Figure 53 Coordinate System Definition form

1. Select the *Cartesian* option.

2. As shown in Figure 53, set the Number of Grid Lines in the X Direction to **8** and in the Y Direction to **6**.

3. In the Grid Spacing area, type **16** in the X Direction edit box.

   Note: Input may be in units other than those shown on the form by explicitly stating the units. For example, if for this case the grid spacing was to be 10 feet, input could be 120in, and the program would convert the number input automatically to be consistent with the units shown on the form.

4. Type **16** in the Y Direction edit box in the Grid Spacing area.

5. Click the **Grid Labels** button to display the Grid Labeling Options form shown in Figure 54.

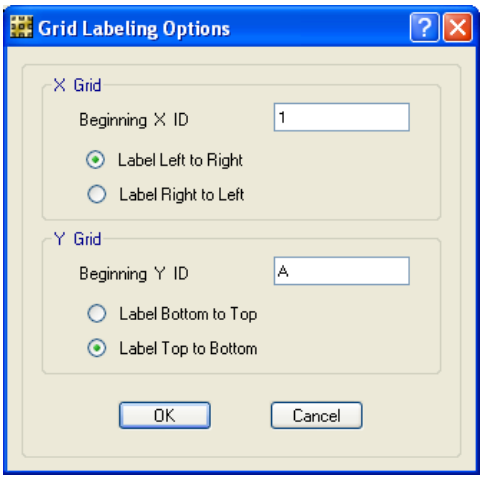


Figure 54 Grid Labeling Options form

   a. In the X Grid area, type **1** in the Beginning X ID edit box. This will number the grids along the X-axis starting with 1.

   b. In the Y Grid area, type **A** in the Beginning Y ID edit box. This will label the grids along the Y-axis starting with A.

   c. In the Y Grid area, select the *Label Top to Bottom* option; grid A will start at the top of the model.

   d. Click the **OK** button to accept the changes and leave the Grid Labeling Options form.

6. Click the **Edit Grid** button on the Coordinate System Definition form to display the form shown in Figure 55. The Coordinate System form is used to modify and edit the grid definitions, as well as set the top of model datum. It also allows the user to set the display options associated with the grids.

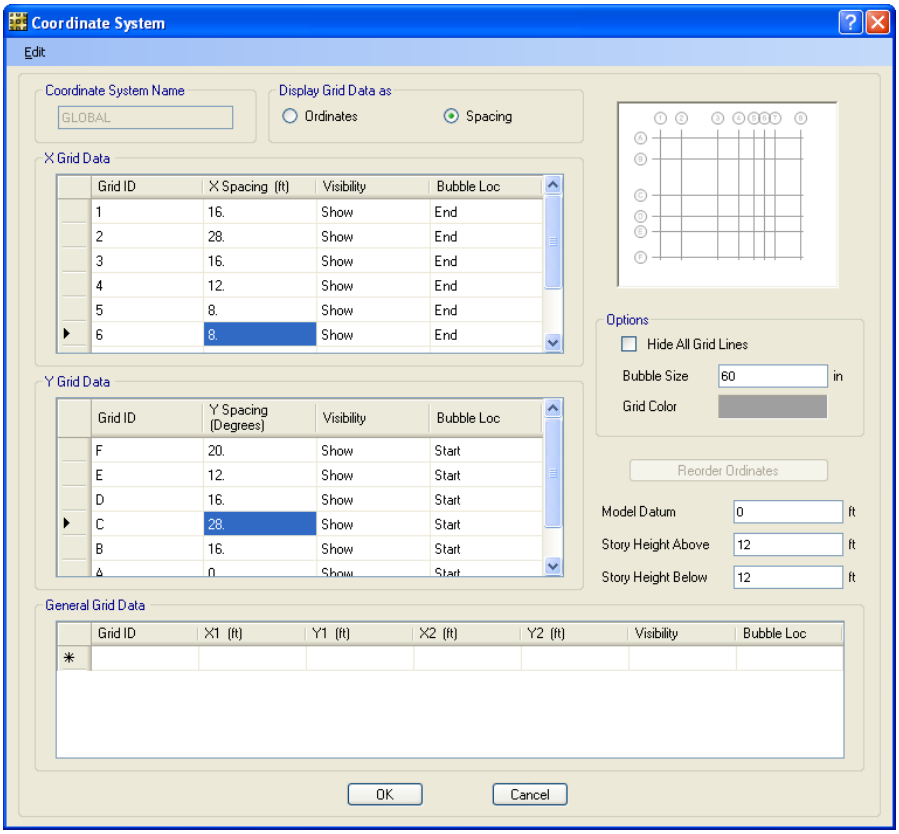


Figure 55 Coordinate System form

a. In the Display Grid Data as area, select the *Spacing* option.

b. In the **X Grid Data** table, change the X spacing as follows:

Grid ID	Change X Spacing to
2	28
4	12
5	8
6	8
7	20

c. In the **Y Grid Data** table, change the Y spacing as follows:

Grid ID	Change Y Spacing to
F	20
E	12
C	28

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

When you leave the Coordinate System form, by default, the grid displays on screen in the main SAFE window with two windows tiled vertically: a Model Explorer window on the left and a Plan View on the right. The number of view windows can be changed using the **Options menu > Windows** command.

E. Click the **View menu > Set Display Options** command to display the Set Display Options form. Uncheck the *Horizon* option in the Items Present In View area and click the **OK** button to exit this form. The Horizon option displays a plane that resembles an engineering calculation grid to illustrate the datum plane location; we are turning this option off to display our coordinate system grid better.

You should now have a display similar to that shown in Figure 56.

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

Note that the Global Axes are displayed as well, and that the Z positive is in the “up” direction. When SAFE refers to the direction of gravity, this is in the negative Z direction, or “down.”

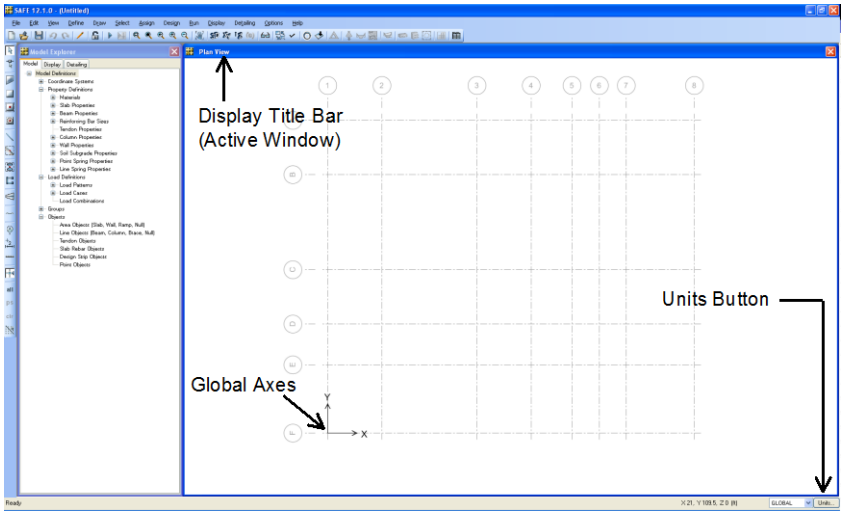


Figure 56 SAFE Main Window

Save the Model

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

As you build the model, each time you save it, you typically will save the file using 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, material and section properties for the slab, tendons, columns, and walls are defined.

Although we are going to define our materials and sections using menu commands, previously defined materials and properties also may be reviewed and modified using the Model Explorer window (see Figure 57). To do this, expand any of the property items on the tree by clicking the + node, and then double click on the desired item to display the associated form.

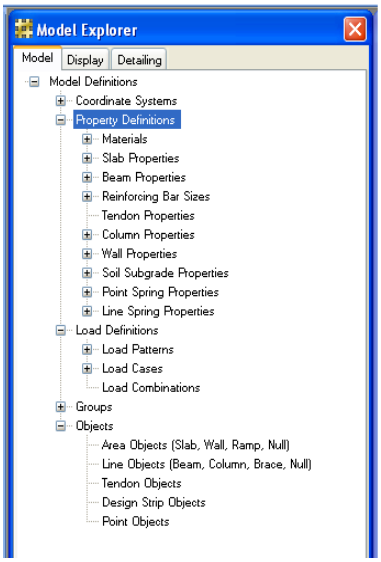


Figure 57 Model Explorer window

Define Material Properties

A. Click the **Define menu > Materials** command to access the Materials form shown in Figure 58.

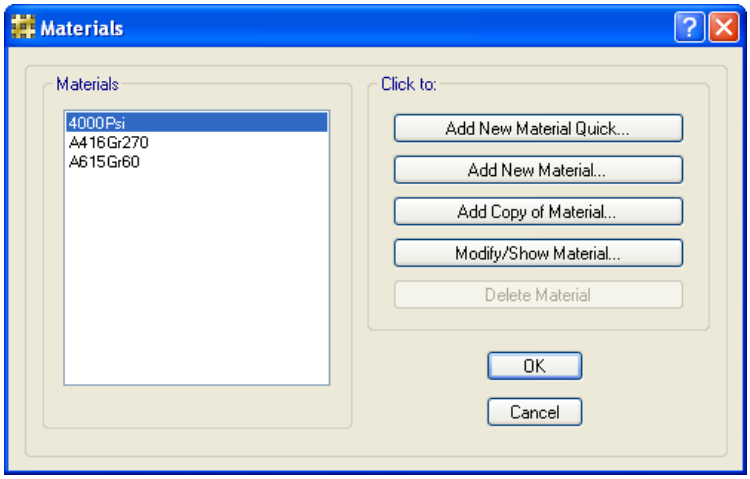


Figure 58 Materials form

B. Click the **Add New Material Quick** button to display the Quick Material Definition form shown in Figure 59. This form will be used to define the 5000psi concrete material for the model.

1. Select *Concrete* from the Material Type drop-down list.
2. Select *f'c 5000 psi* from the Specification drop-down list.
3. Click the **OK** button to exit the Quick Material Definition form.

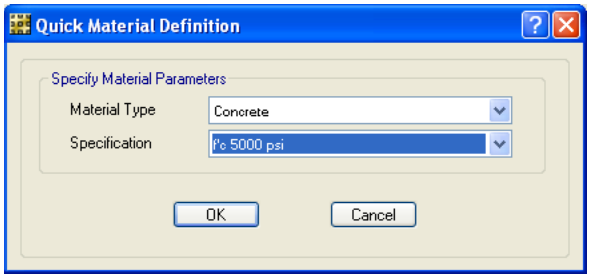


Figure 59 Quick Material Definition form

C. In the Materials area of the Materials form, highlight *A416Gr270*.

D. Click the **Modify/Show Material** button to display the Material Property Data form shown in Figure 60. This form lists the properties associated with Grade 270 prestressing strands; this is the tendon material property that will be used in our model.

1. Click the **OK** button on the Materials form to accept all of the defined materials.

E. Click the **File menu > Save** command to save your model.

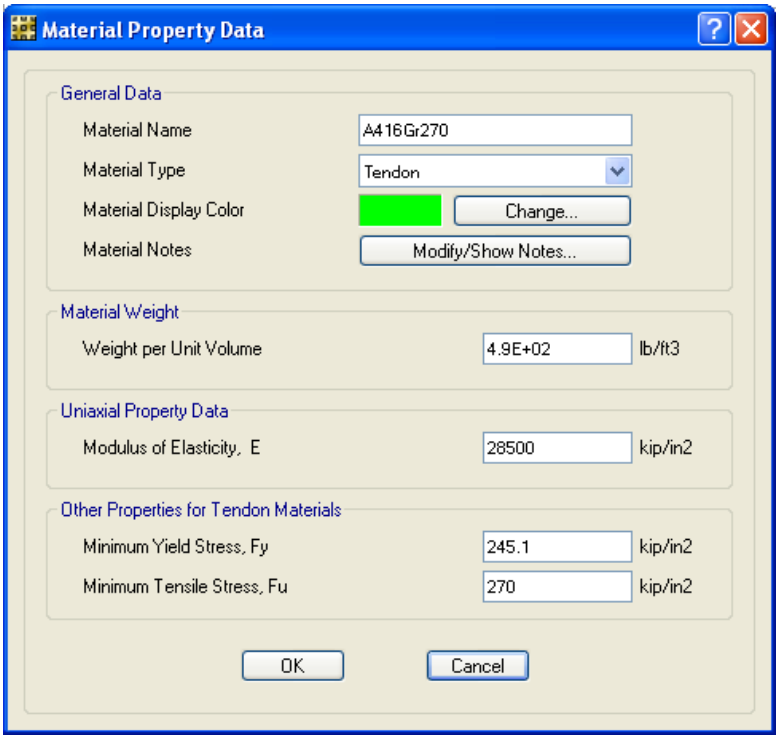


Figure 60 Material Property Data form

Define Slab Properties

A. Click the **Define menu > Slab Properties** command to access the Slab Properties form shown in Figure 61.

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

C. Recall that for this tutorial project, the slab thickness is 8 inches. To adjust the default dimensions of *SLAB1*, click the **Modify/Show Property** button to access the Slab Property Data form shown in Figure 62.

   1. In the General Data area, select *5000Psi* from the Slab Material drop-down list.

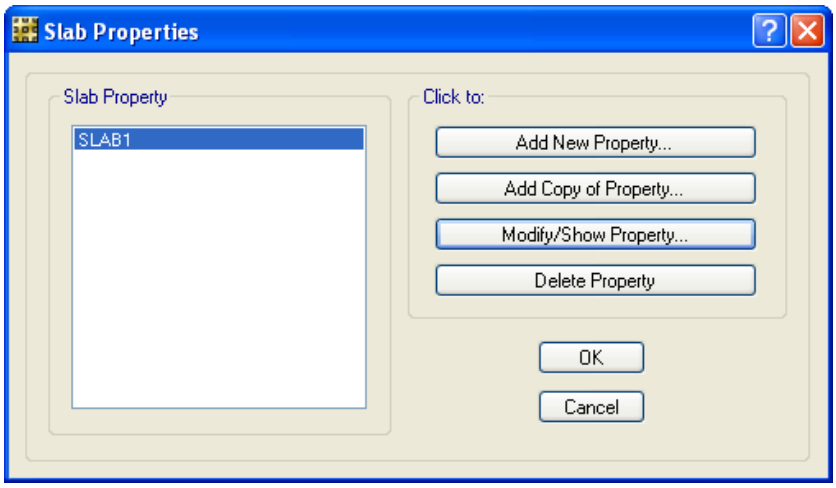


Figure 61 Slab Properties form

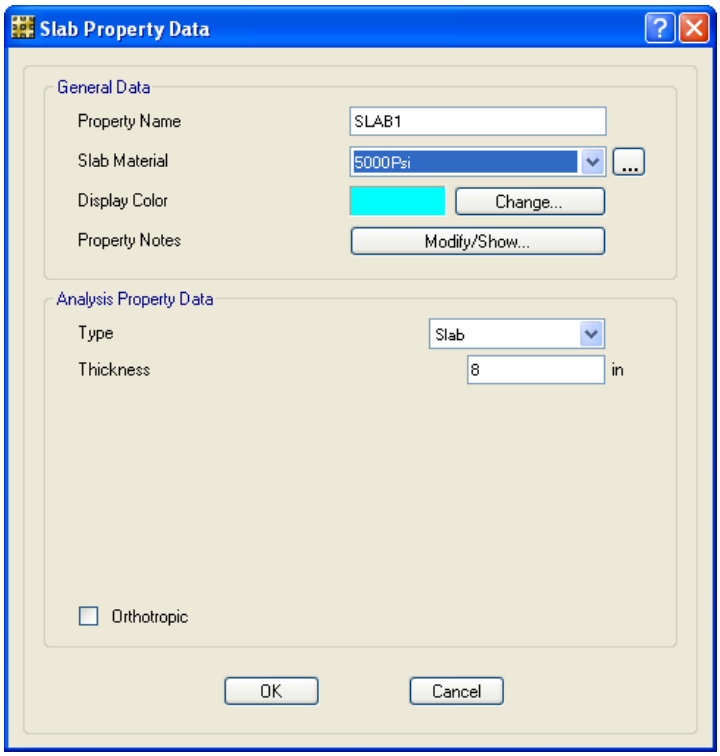


Figure 62 Slab Property Data form

Note: Additional material properties may be defined by clicking on the "**…**" button next to the load pattern name. A "**…**" button returns you to the form used to define the item in the adjacent drop-down list, which in this case is the Materials form.

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

3. If not already displayed, type **8** in the Thickness edit box in the Analysis Property Data area.

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

D. Click the **OK** button to end the slab property definition.

E. Click the **File menu > Save** command to save the model.

Define Tendon Properties

A. Click the **Define menu > Tendon Properties** command to access the Tendon Properties form.

B. Click the **Add New Property** button to access the Tendon Property Data form shown in Figure 63. Recall that the strands for the tendons have an area of 0.153 square inches.

1. Verify that *TENDON1* is showing in the Property Name edit box in the General Data area.

2. In the General Data area, make sure that *A416Gr270* is displayed in the Material Type drop-down list.

3. If not already entered, type **0.153** in the Strand Area edit box in the Property Data area.

4. Click the **OK** button to end the tendon property definition.

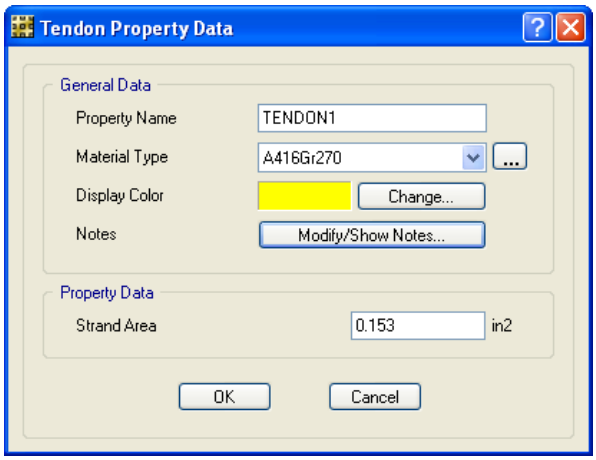


Figure 63 Tendon Property Data form

C. Click the **OK** button to leave the Tendon Properties form.

D. Click the **File menu** > **Save** command to save your model.

Define Column Properties

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

B. In the Column Property area, highlight *COL1*.

C. Click the **Modify/Show Property** button to access the Column Property Data form shown in Figure 64. Recall that the columns for the project are 12-inch square.

1. In the General Data area, select *5000Psi* from the Material drop-down list.

2. In the Column Section Dimensions area, select *Rectangular* from the Column Shape drop-down list.

3. In the Column Section Dimensions area, type **12** into the Parallel to 2-Axis edit box and type **12** into the Parallel to 3-Axis edit box.

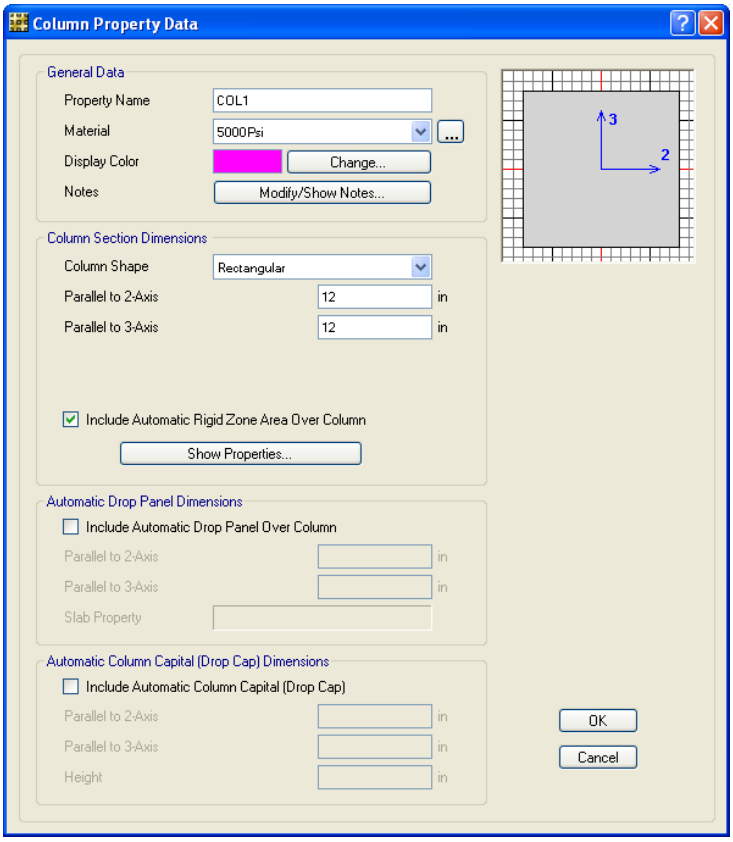


Figure 64 Column Property Data form

4. Make sure that the *Include Automatic Rigid Zone Area Over Column* option is checked. This option restricts deformation of the slab at the column location, which prevents unrealistic peaks in moment distribution from occurring.

5. Click the **OK** button to leave the Column Property Data form.

D. Click the **OK** button to accept the Column Property definition.

Define Wall Properties

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

B. In the Wall Property area, highlight *Wall1*.

C. Click the **Modify/Show Property** button to access the Wall Property Data form shown in Figure 65. Recall that the walls for the project are 8-inch thick.

1. In the General Data area, select *5000Psi* from the Wall Material drop-down list.

2. In the Wall Dimensions area, type **8** into the Thickness edit box.

3. Check the *Include Automatic Rigid Zone Area Over Wall* checkbox. This option restricts deformation of the slab at the wall location, which prevents unrealistic peaks in moment distribution from occurring.

4. Click the **OK** button to leave the Wall Property Data form.

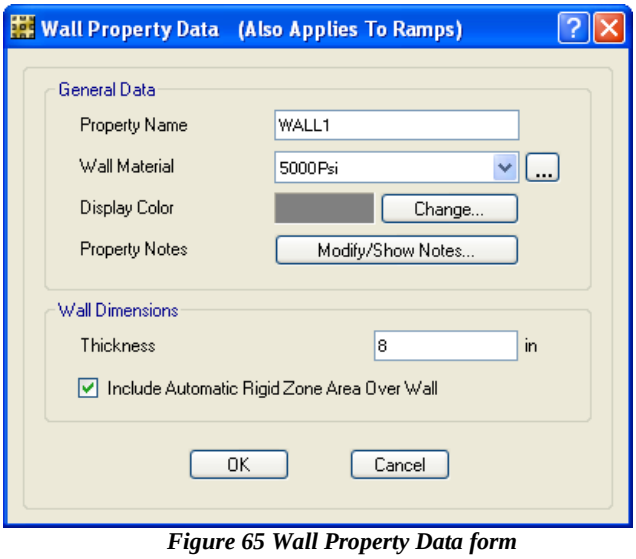


Figure 65 Wall Property Data form

D. Click the **OK** button to accept the Wall Property definition.

This completes the material and section property definition phase of the model creation. The slab and tendon properties will be assigned in the model datum plane, while the columns and walls will be assigned as supports. Supports also can be assigned as point restraints, point springs or line springs. Support stiffnesses are calculated by SAFE based on the cross-sectional properties, material properties and lengths specified when the columns and walls are drawn.

Step 3 Define Static Load Patterns

In this Step, the dead, live, and post-tensioning static load patterns are defined. That is, we will name the various types of loads (dead, live, pre-stress-final, and so on), and specify the self-weight multipliers. The loads will be assigned to objects, and the values for the loads specified (uniform dead load of 35 psf and live load of 50 psf), in Step 8.

A. Click the **Define menu > Load Patterns** command to access the Load Patterns form shown in Figure 66.

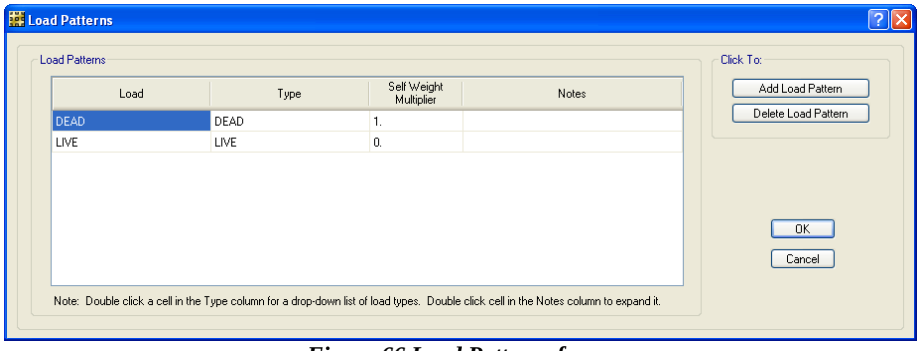


Figure 66 Load Patterns form

B. Note that load patterns *DEAD* and *LIVE* are defined by default.

C. 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) for the *DEAD* load. Only the *DEAD* load pattern should have a non-zero Self Weight Multiplier.

D. Click the **Add Load Pattern** button.

E. Highlight *LPAT1* in the Load column and type **PT-FINAL**.

F. Select *PRESTRESS-FINAL* from the drop-down list under Type for the PT-FINAL load pattern.

G. Click the **Add Load Pattern** button.

H. Highlight *LPAT1* in the Load column and type **PT-TRANSFER.**

I. Select *PRESTRESS-TRANSFER* from the drop-down list under Type for the PT-TRANSFER load pattern. The Load Patterns form should appear as shown in Figure 67.

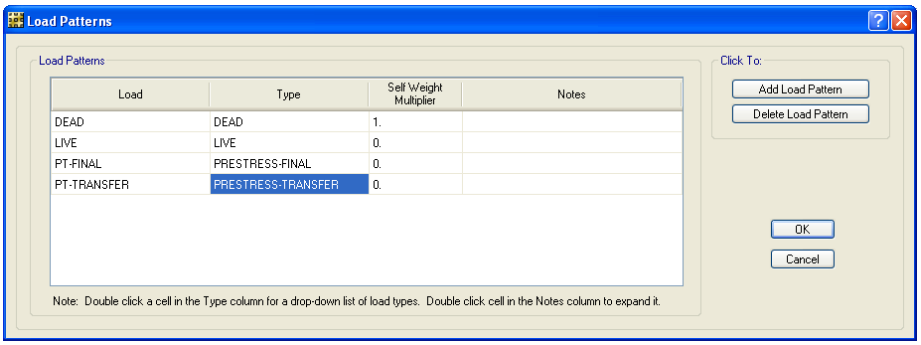


Figure 67 Load Patterns form after all load patterns are entered

J. Click the **OK** button to accept all of the defined static load patterns.

K. Click the **File menu > Save** command.

Step 4 Define Load Cases

In this Step, the Load Cases are defined. This is where the type of analysis is specified.

A. Click the **Define menu > Load Cases** command to access the Load Cases form shown in Figure 68.

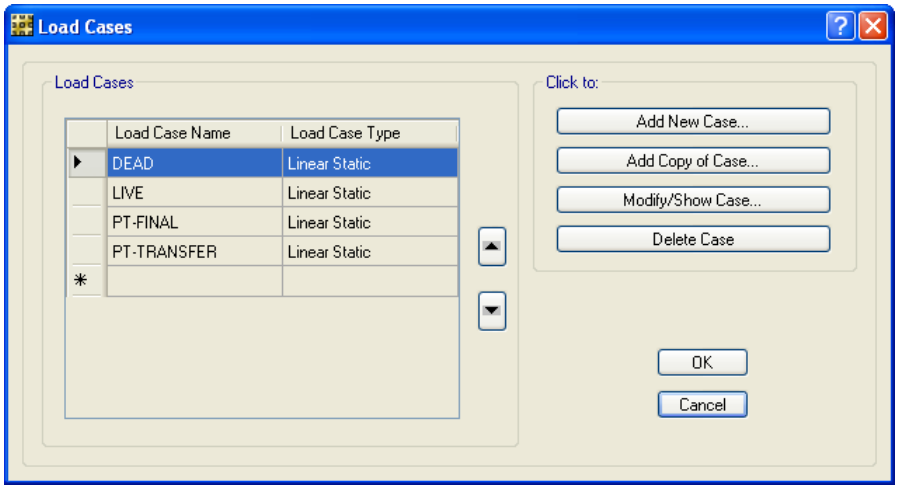


Figure 68 Load Cases form

B. With the *DEAD* Load Case Name highlighted, click the **Modify/Show Case** button to display the Load Case Data form shown in Figure 69. This data form changes based on the type of load case specified.

1. In the Load Case Type area, select *Static* from the drop-down list. Modal and Hyperstatic are also available as load case types.

2. In the Analysis Type area, select the *Linear* option. When working with a Static Load Case Type, the program offers the option to do Linear, Nonlinear (Allow Uplift), Nonlinear (Cracked), or Nonlinear (Long Term Cracked) analysis. For our tutorial example a Static, Linear analysis will be performed for DEAD, LIVE, PT-FINAL and PT-TRANSFER.

3. In the Loads Applied area, verify that the load pattern is *DEAD* with a scale factor of *1*.

4. Click the **OK** button to close the Load Case Data form.

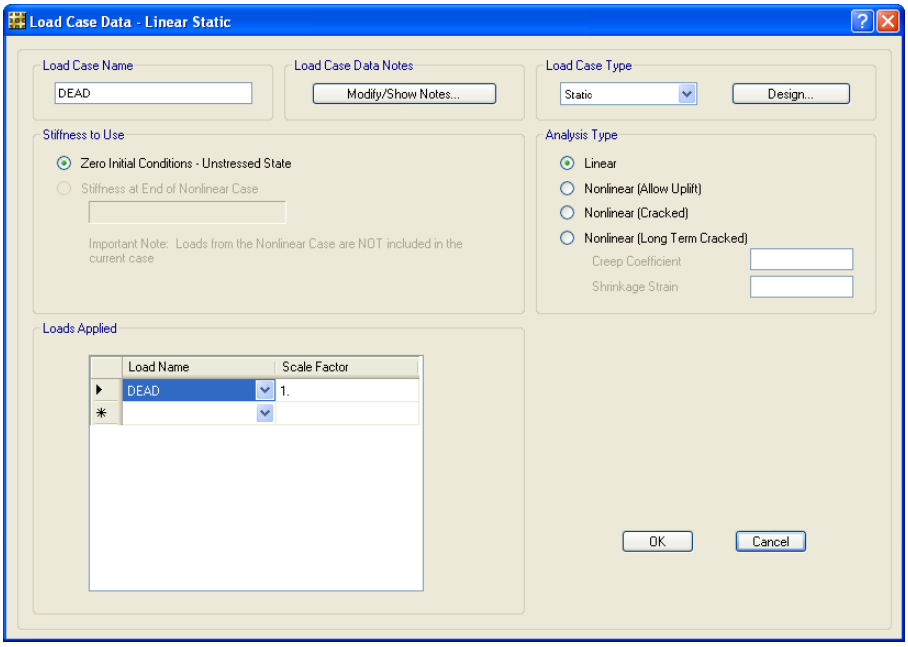


Figure 69 Load Case Data form

C. Review the other load cases, if so desired, by selecting a case and using the **Modify/Show Case** button as described for the DEAD load case.

D. Click the **OK** button to close the Load Cases form.

E. Click the **File menu** > **Save** command.

Step 5 Draw Objects

In this Step, slabs, columns, walls, openings, and design strips will be drawn.

Draw Slabs

Ensure that the Plan View is active (click anywhere in the display window; a window is active when the Display Title Bar, just below the hori-

zontal toolbar, is highlighted). Now draw area objects to model the slab using the following Action Items.

A. Click the **Draw menu > Snap Options** command to display the Snap Options form shown in Figure 70.

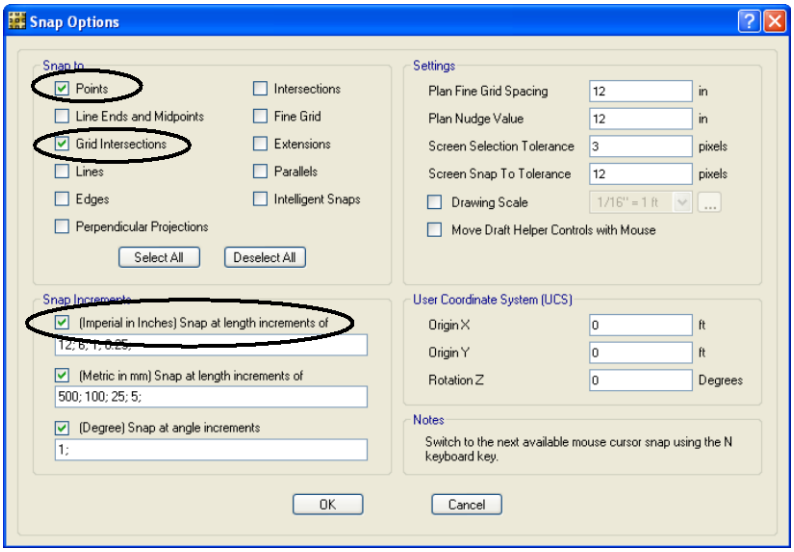


Figure 70 Snap Options form

B. In the Snap to area, make sure that the *Points* and *Grid Intersections* options are checked. These snap options will assist in accurately positioning objects when drawing.

C. In the Snap Increments area, make sure that the *(Imperial in Inches) Snap at length increments of* option is checked. When drawing slab edges, beams and walls, a dimension line will appear and the object will snap to the values specified in this edit box.

D. Click the **OK** button to close the Snap Options form.

E. Click the **Draw menu > Draw Slabs/Areas** command to access the Draw Slabs/Areas form shown in Figure 71. If the Draw Slabs/Areas form covers part of the model, click on the blue title bar, hold down the mouse button, and drag it out of the way.

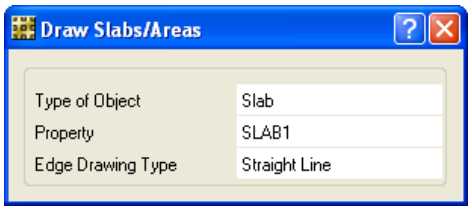


Figure 71 Draw Slabs/Areas form

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

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

H. Click in the Edge Drawing Type drop-down list and select *Straight Line*. Although not used in this project, slab edges also may be drawn with arcs and curves.

I. To draw the first corner of the slab, click once in the Plan View at the intersection of grid lines 1 and A (the cursor should display *Grid Point 1 A* at the correct location). Then moving clockwise around the grid (note how the slab edge is dimensioned as you draw), click once at these grid intersections in this order to draw the outline of the slab: 8A, 8D, 4D, 4F, and 1F. If you mistakenly click at the wrong point, use the ←**Backspace** key on the keyboard to "undo" the point and try again. After clicking at grid 1F, press the **Enter** key on the keyboard. The shaded slab object should now display.

   If you made a mistake and wish to try re-drawing the entire object, click the **Select menu > Select > Pointer/Window** command to leave the Draw mode and go to the Select mode. Then click the **Edit menu > Undo Area Add** command, and repeat Items E through I.

J. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Slabs/Areas command.

K. Select the slab by clicking on it anywhere. The status bar in the lower left-hand corner should show "1 Areas, 6 Edges selected." If you make a mistake in selecting, press the **Select menu > Clear Selection** command and try again.

L. Click the **Edit menu > Edit Areas > Expand/Shrink Areas** command to display the Expand/Shrink Areas form shown in Figure 72.

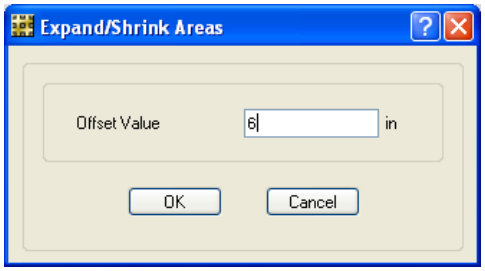


Figure 72 Expand/Shrink Areas form

M. Type **6** into the Offset Value edit box. We will use this form to expand (a positive value expands) the slab by 6-inches at each corner to create the perimeter overhang that is needed to accommodate the width of the columns (1/2 of 12 inches).

N. Click the **OK** button to finish the slab.

O. Click the **File menu > Save** command to save your model.

Draw Columns

With the active window set as described in the preceding *Draw Slabs* section (i.e., Plan View window active and the snap to points and grid intersections enabled), use the following Action Items to draw columns.

A. Click the **Draw menu > Draw Columns** command to access the Draw Columns form shown in Figure 73.

B. Click in the Property Below edit box and select *COL1* from the drop-down list. This is the property for the 12-inch by 12-inch column defined in Step 2.

C. Click in the Property Above edit box and select *COL1* from the drop-down list.

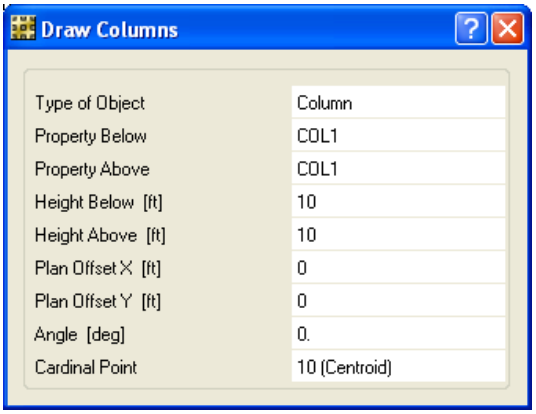


Figure 73 Draw Columns form

D. Click in the Height Below edit box and type **10**. Note that the units are feet.

E. Click in the Height Above edit box and type **10**.

F. Click in the Cardinal Point edit box and select *10 (Centroid)* from the drop-down list.

G. Left click once at each of the following grid intersections to draw the columns: 1A, 1B, 1F, 2A, 2B, 2F, 3F, 4F, 5A, 5B, 5D, 8A, 8B, 8C, and 8D.

If you have made a mistake while drawing the columns, click the **Select menu > Select > Pointer/Window** command to leave the Draw mode and go to the Select mode. Then click the **Edit menu > Undo Column Object Add** command as many times as needed to remove the incorrectly drawn columns, and repeat Items A through G.

H. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Columns command.

I. Click the **File menu > Save** command to save your model.

J. Click the **View menu > Set Default 3D View** command to display the model in 3D. Note how the columns extend above and below the slab.

K. Click the **View menu > Set Plan View** command to return to the Plan View before continuing with the project.

Draw Walls

Similar to the preceding two sections, ensure that the Plan View is active and that the snap to points, grid intersections and the snap increments options are enabled. Now use the following Action Items to draw walls.

A. Click the **Draw menu > Snap Options** command to access the Snap Options form.

B. Check the *Line Ends and Midpoints* checkbox in the Snap to area.

C. Click the **OK** button to close the Snap Options form.

D. Click the **Draw menu > Draw Walls** command to access the Draw Walls form shown in Figure 74.

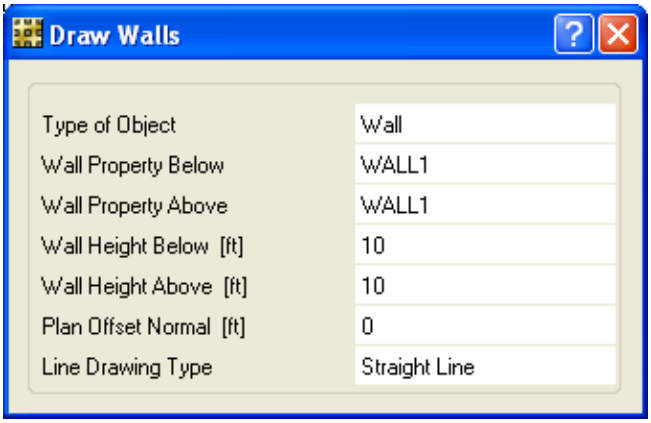


Figure 74 Draw Walls form

E. Click in the drop-down list opposite the Wall Property Below item and select *WALL1*. This is the wall property for the 8-inch-thick wall defined in Step 2.

F. Click in the Wall Property Above drop-down list and select *WALL1*.

G. Click in the Wall Height Below edit box and type **10**. Note that the units are feet.

H. Click in the Wall Height Above edit box and type **10**.

I. Click in the drop-down list opposite the Line Drawing Type item and select *Straight Line*. Although not used in this project, walls may also be drawn with arcs and curves.

J. Left click at grid intersection 1C to begin drawing the first wall. Draw the wall along grid line C to grid intersection 2C and then click again. The snap increments dimension line should show *16 ft*. Hit the **Enter** key on the keyboard to complete the wall. This places a wall 16 feet long along grid line C.

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

K. For the next wall, also located along grid line C, left click at grid intersection 3C to begin drawing the wall and at intersection 4C (the snap increment dimension line should show *16 ft*) to designate the end of the wall. Hit the **Enter** key on the keyboard to complete the wall.

L. Click at grid intersection 1E and then at intersection 2E to draw the next wall along grid E. Hit the **Enter** key on the keyboard to complete the wall.

M. Click at grid intersection 3E and then at intersection 4E to draw the last x-direction wall along grid E. Hit the **Enter** key on the keyboard to complete the wall.

N. Click at grid intersection 3A and then at intersection 3B to draw the first y-direction wall along grid 3. Hit the **Enter** key on the keyboard to complete the wall.

O. Snap the mouse to the *Mid Point* of the wall (a triangular cursor should display) along grid line C between grids 3 and 4, and left click. Move the mouse to the *Mid Point* of the wall along grid line E between grids 3 and 4 and click again – the snap increment dimension line should show *28 ft*. Hit the **Enter** key on the keyboard to complete the wall.

P. Click at grid intersection 6C and then at intersection 6D to draw the next y-direction wall along grid 6. Hit the **Enter** key on the keyboard to complete the wall.

Q. Click at grid intersection 7A and then at intersection 7B to draw the last y-direction wall. Hit the **Enter** key on the keyboard to complete the wall.

R. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Walls command. Your model should look similar to Figure 75.

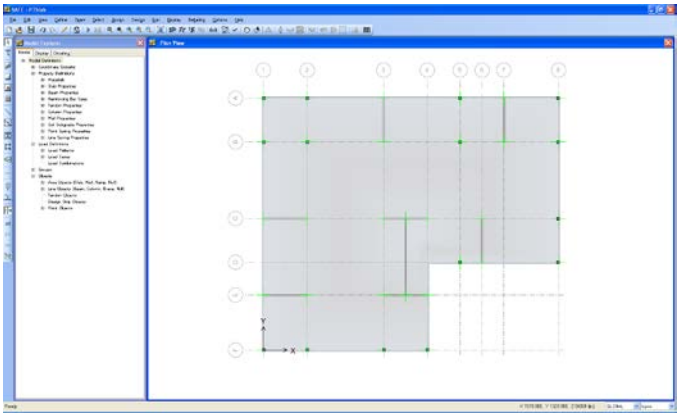


Figure 75 The model after drawing columns and walls

Draw Openings (Area Objects)

Similar to the previous sections, ensure that the Plan View is active. Draw an area object to model the opening as follows:

A. Click the **Draw menu > Draw Rectangular Slabs/Areas** command to display the Draw Rectangular Slabs/Areas form shown in Figure 76.

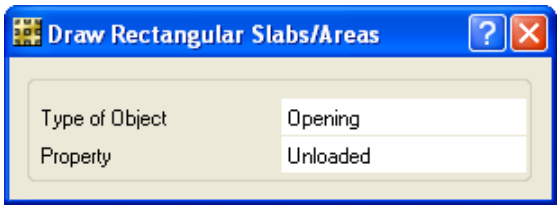


Figure 76 Draw Rectangular Slabs/Areas form

B. Click once in the drop-down list opposite the Type of Object item and select *Opening*.

C. Left click at the intersection of grid lines 3 and C, and while holding the left mouse button down, move diagonally down and to the right to the point where grid line D intersects the adjacent y-direction wall, and release the button. An area object (8 ft x 28 ft) with different shading should appear, indicating that an opening for the stairs has been drawn. An opening takes priority over an object with assigned slab properties when the program determines the stiffness formulation.

D. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Rectangular Slabs/Areas command.

E. Click the **Draw menu > Snap Options** command to access the Snap Options form.

F. Check the *Intersections* checkbox in the Snap to area.

G. Click the **OK** button to close the Snap Options form.

H. Click the **View menu > Rubber Band Zoom** command. Left click just above and to the left of grid intersection 6A, and while holding the mouse button down, drag a window down and to the right of grid intersection 7B.

I. Click the **Draw menu > Draw Rectangular Slabs/Areas** command to display the Draw Rectangular Slabs/Areas form.

J. To draw the second opening, snap the mouse to the intersection of grid line 6 and the edge of the slab (the cursor should say *Intersection*), and while holding the left mouse button down, move diagonally down and to the right to the *Mid Point* of the wall that lies along grid line 7, and release the button.

K. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Rectangular Slabs/Areas command.

L. Click the View menu > Restore Full View command.

M. Click the **File menu > Save** command to save your model.

Draw Design Strips

Similar to the previous sections, ensure that the Plan View is active. Design strips determine how tendons and reinforcing will be calculated and positioned in the slab. Typically design strips are positioned in two principal directions: Layer A and Layer B. Additionally, for this model only column strips will be defined, which follows the generally accepted practice for post-tensioned slabs. Use the following Action Items to draw design strips:

A. Click the **Draw menu > Draw Design Strips** command to display the Draw Design Strips form shown in Figure 77.

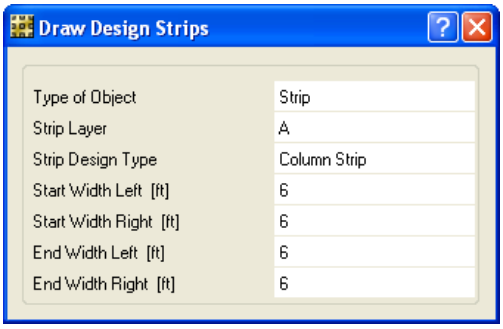


Figure 77 Draw Design Strips form

B. Select *A* from the Strip Layer drop-down list. Design strips subsequently drawn will be assigned to Layer A.

C. Select *Column Strip* from the Strip Design Type drop-down list.

D. The values showing in the width edit boxes may be ignored, as we will edit the design strips in a later step to automatically adjust their width based on the location of adjacent strips.

E. Left click at grid intersection 1A to begin drawing the first design strip. Click again at grid intersection 6A and hit the **Enter** key on the keyboard to complete the design strip.

F. Repeat Item E for the following pairs of grid intersections: 7A to 8A; 1B to 8B; 1C to 8C; 1E to 4E; and 1F to 4F.

G. Snap the mouse to the *Point* at the approximate intersection of grids D and 3.5 (between 3 and 4) along the wall and left click to begin the design strip, and then click again at grid intersection 8D. Hit the **Enter** key to complete the design strip. This completes the Layer A design strips.

   Note that although the majority of the design strips were drawn using grid intersections rather than the edge of the slab, the program assumes that they should go to the slab edge and will automatically extend the design strips when the tolerance is small.

H. On the Draw Design Strips form, select *B* from the Strip Layer drop-down list. Design strips subsequently drawn will be assigned to Layer B.

I. Left click at grid intersection 1A to begin drawing the first Layer B design strip. Click again at grid intersection 1F and hit the **Enter** key on the keyboard to complete the design strip.

J. Repeat Item I for the following pairs of grid intersections: 2A to 2F; 3A to 3F; 4A to 4F; 5A to 5D; 6A to 6D; 7A to 7D; and 8A to 8D.

K. Click on the **Select menu > Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Design Strips command and return to the select mode.

L. Click the **Select menu > Select > Properties > Design Strip Layers** command to display the Select Design Strip Layers form shown in Figure 78.

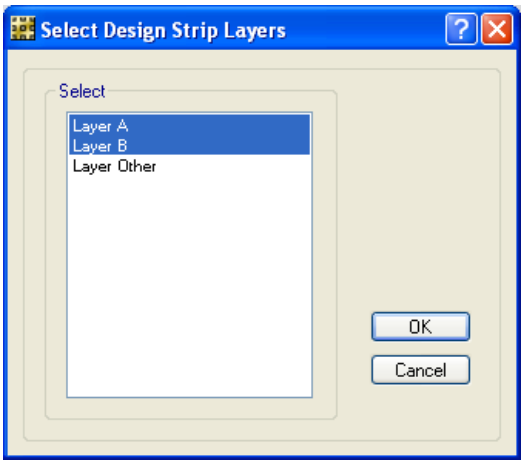


Figure 78 Select Design Strip Layers form

M. Highlight *Layer A* and *Layer B* in the Select area by holding down the Shift key on the keyboard while clicking on the items.

N. Click the **OK** button to leave the form. The status bar in the lower left-hand corner should now show "15 Design Strips selected."

O. Click the **Edit menu > Add/Edit Design Strips > Edit Strip Widths** command to display the Edit Strip Widths form.

P. In the Strip Width Options area, select the *Auto Widen Entire Strip* option. Selecting this option allows the program to automatically set the width of each design strip based on the location of adjacent strips.

Q. Click the **OK** button to accept the selections and leave the Edit Strip Widths form.

R. Click the **File menu > Save** command to save your model.

Step 6 Add Tendons

In this step, tendons will be added to the model. Although tendons may be drawn in the model using the **Draw menu > Draw Tendons** command, it is often more efficient to add them to the design strips, which is how tendons will be added to this project. The number of tendons (and strands) will be determined by the program based on the following defaults: tendon jacking stress = 216 ksi; stressing losses = 27 ksi; long-term losses = 13.5 ksi; tendons are jacked from the J-end (ending point) of the tendon.

Similar to the previous sections, ensure that the Plan View is active. Add tendons to the design strips as follows:

A. Click the **Select menu > Select > Properties > Design Strip Layers** command to display the Select Design Strip Layers form.

B. Highlight *Layer A* in the Select area.

C. Click the **OK** button to accept the selection. The status bar in the lower left-hand corner should now show "7 Design Strips selected."

D. Click the **Edit menu > Add/Edit Tendons > Add Tendons in Strips** command to display the Quick Tendon Layout form shown in Figure 79.

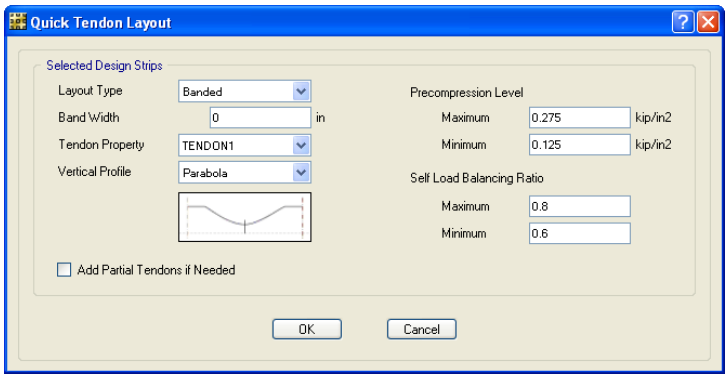


Figure 79 Quick Tendon Layout form

E. Select *Banded* from the Layout Type drop-down list. Tendons will be banded in the Layer A direction, and distributed in the Layer B direction.

F. Type **0** into the Band Width edit box. By using a value of 0, the program will calculate how many total strands are needed.

G. Select *TENDON1* from the Tendon Property drop-down list. This is the tendon property defined in Step 2.

H. Select *Parabola* from the Vertical Profile drop-down list.

I. Verify that the Self Load Balancing Ratios are set to *0.8* for Maximum and *0.6* for Minimum. This controls how much of the structure's self weight the program will try to balance with the post-tensioning.

J. Click the **OK** button to leave the Quick Tendon Layout form. A single tendon in each Layer A design strip should now appear.

K. Click the **Select menu > Select > Properties > Design Strip Layers** command to display the Select Design Strip Layers form.

L. Highlight *Layer B* in the Select area.

M. Click the **OK** button to accept the selection. The status bar in the lower left-hand corner should now show "8 Design Strips selected."

N. Click the **Edit menu > Add/Edit Tendons > Add Tendons in Strips** command to display the Quick Tendon Layout form.

O. Select *Distributed* from the Layout Type drop-down list.

P. Type **38** into the Tendon Spacing edit box. The program will determine how many strands are needed in each tendon for this spacing.

Q. Select *Reverse Parabola* from the Vertical Profile drop-down list.

R. Verify that the Self Load Balancing Ratios are set to *0.8* for Maximum and *0.6* for Minimum.

S. Click the **OK** button to leave the Quick Tendon Layout form. Tendons in the Layer B design strips should now display.

T. Hold down the **Shift** key on your keyboard and **right** click once on any of the tendons just drawn. A selection list similar to the one shown in Figure 80 displays because multiple objects exist at that location. In this case, one tendon and one area object exist at the same location. Note that the selection list will display only when the **Shift** key is used with the right click.

U. Highlight the tendon object and click the **OK** button. Because a right click action initiated the selection process, a Tendon Object Information form similar to the one shown in Figure 81 will display.

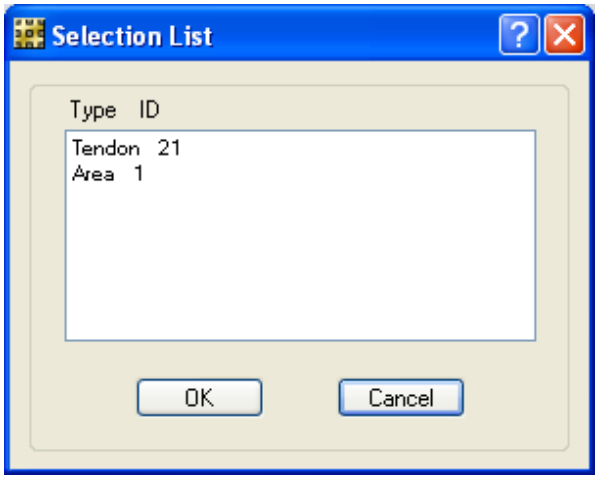


Figure 80 Selection List form

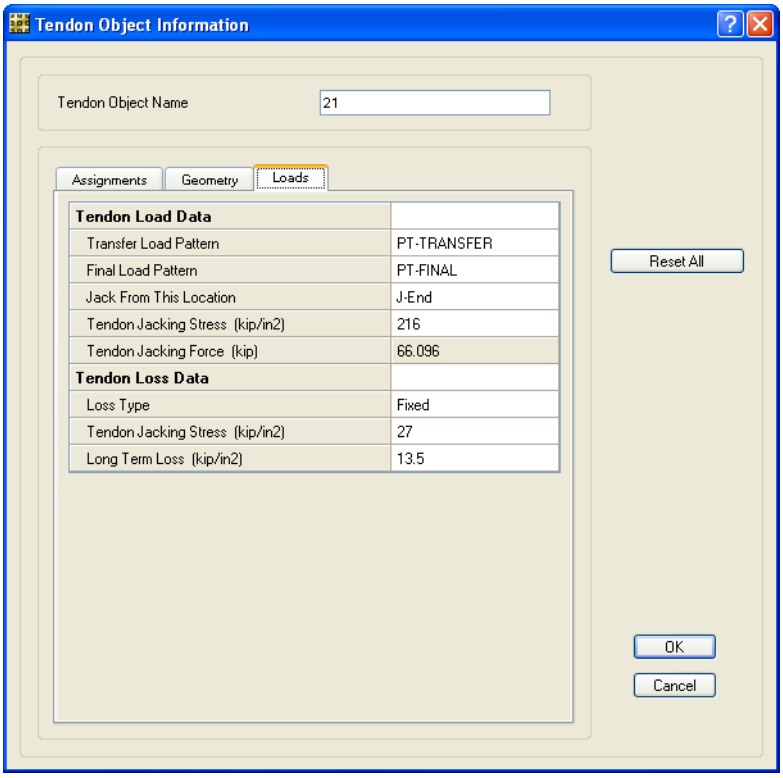


Figure 81 Tendon Object Information form

1. Click on the *Loads* tab. Note that the Tendon Jacking Stress is 216 ksi, and that the losses are 27 ksi for Stressing and 13.5 ksi for Long Term.

2. Click on the *Geometry* tab. This tab describes the geometry of the tendon in both plan and elevation.

3. Left click in any of the edit boxes associated with a vertical span, and the Tendon Vertical Profile form appears, similar to that shown in Figure 82. This form displays the vertical profile of the tendon in elevation, and allows the tendon profile to be altered either graphically, by grabbing a control point on the tendon and moving it to a new position, or numerically, by selecting the desired span and entering new values into the table. Click the **OK** button, to accept the default profiles.

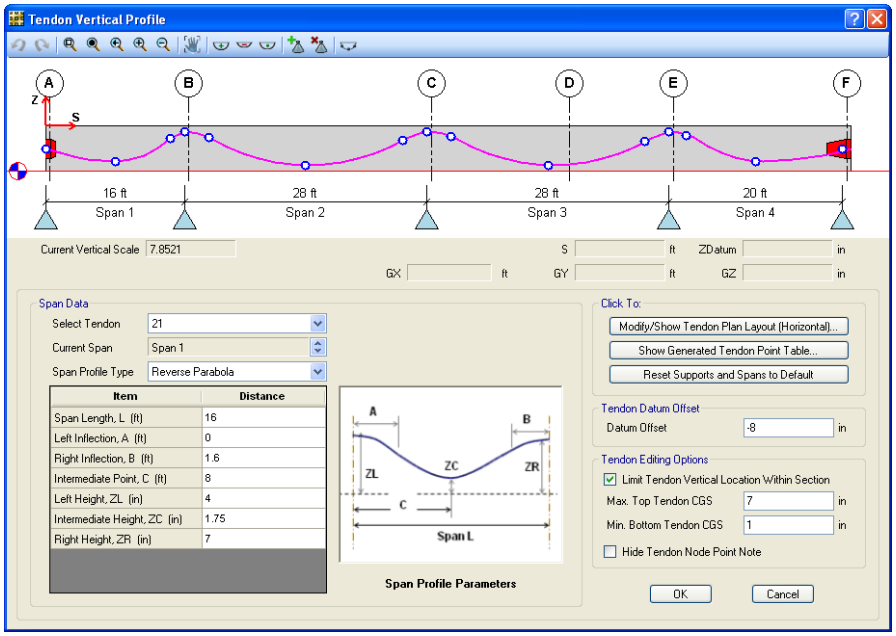


Figure 82 Tendon Vertical Profile form

4. Click the *Assignments* tab. This tab lists the number of strands associated with the selected tendon.

5. Click the **OK** button to exit the Tendon Object Information form.

V. Click the **File menu > Save** command to save your model.

Step 7 Set Display Options

In this Step, the set display options will be used to alter the objects displayed.

A. Click the **View menu > Set Display Options** command. When the Set Display Options form displays, uncheck the *Tendon*, *Design Strip Layer A* and *Design Strip Layer B* check boxes in the Items Present in View area as shown in Figure 83. This action will turn off the display of the tendons and design strips.

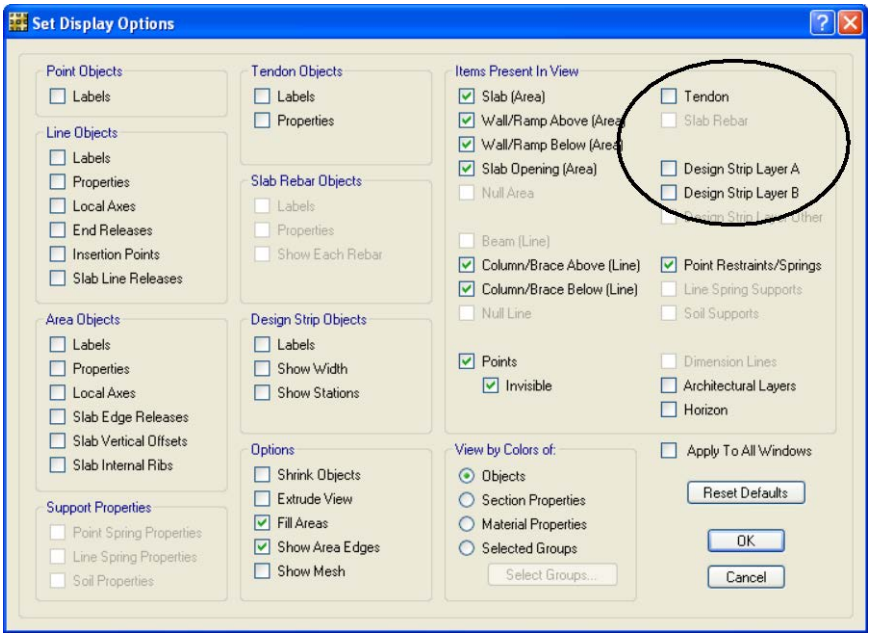


Figure 83 Set Display Options form

B. Click the **OK** button to accept the changes, and the model now appears as shown in Figure 84.

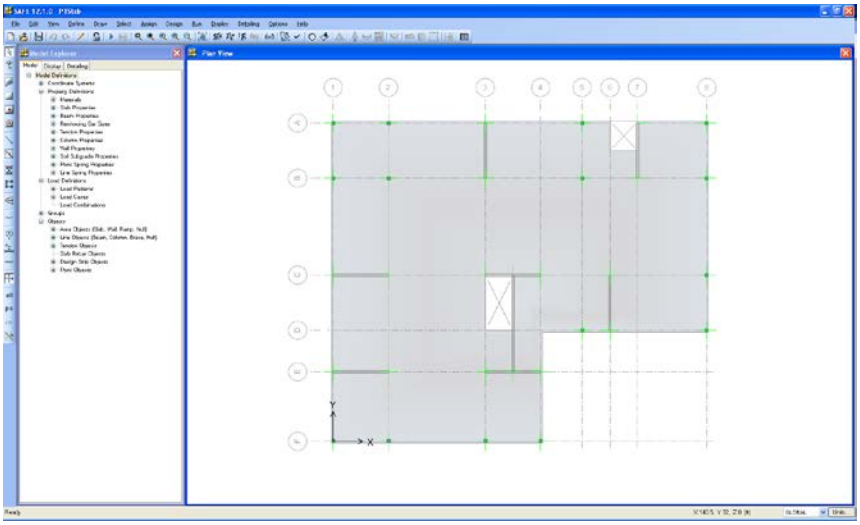


Figure 84 Model after all objects have been drawn

Step 8 Assign Loads

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

A. Select the slab by clicking on it anywhere that is ***not*** a wall, column or opening. The status bar in the lower left-hand corner should show "1 Areas, 6 Edges selected." If you make a mistake in selecting, click the **Select menu > Clear Selection** command, and try again.

B. Click the **Assign menu > Load Data > Surface Loads** command to access the Surface Loads form shown in Figure 85.

C. If it is not already displayed, select DEAD from the Load Pattern Name drop-down list.

D. Select *Gravity* from the Load Direction area drop-down list.

E. In the Uniform Loads, type **35** in the Uniform Load edit box. Note that the input units are set to psf.

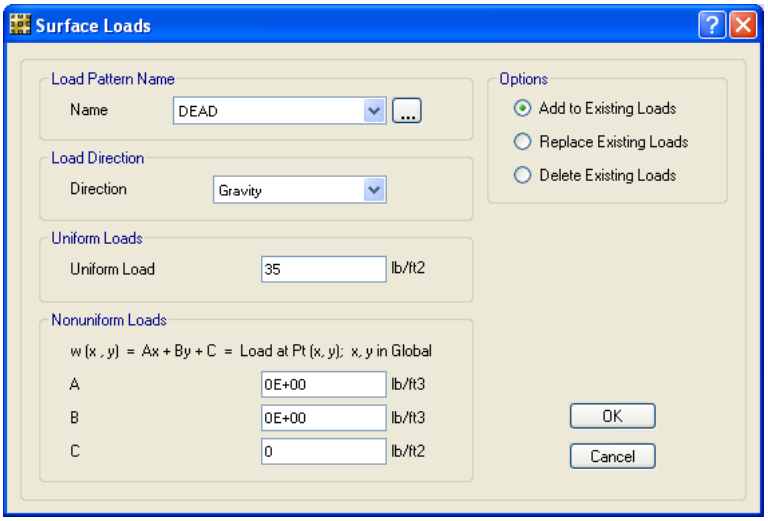


Figure 85 Surface Loads form

Note: Additional load patterns may be defined by clicking on the "**…**" button next to the load pattern name. A "**…**" button returns you to the form used to define the item in the adjacent drop-down list, which in this case is the Load Patterns form.

F. 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.

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

H. Click the **Assign menu > Load Data > Surface Loads** command to again access the Surface Loads form.

I. Select *LIVE* from the Load Pattern Name drop-down list.

J. Type **50** in the Uniform Load edit box in the Uniform Loads area.

K. 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 wall, column or opening to access the Slab-Type Area Object Information form shown in Figure 86.

Select the *Loads* tab and note that the DEAD Load Pattern has a Load Value of 35psf, and that the LIVE Load Pattern has a Load Value of 50psf. Click the **OK** button to close the Slab-Type Area Object Information form. Click the **File menu > Save** command to save your model.

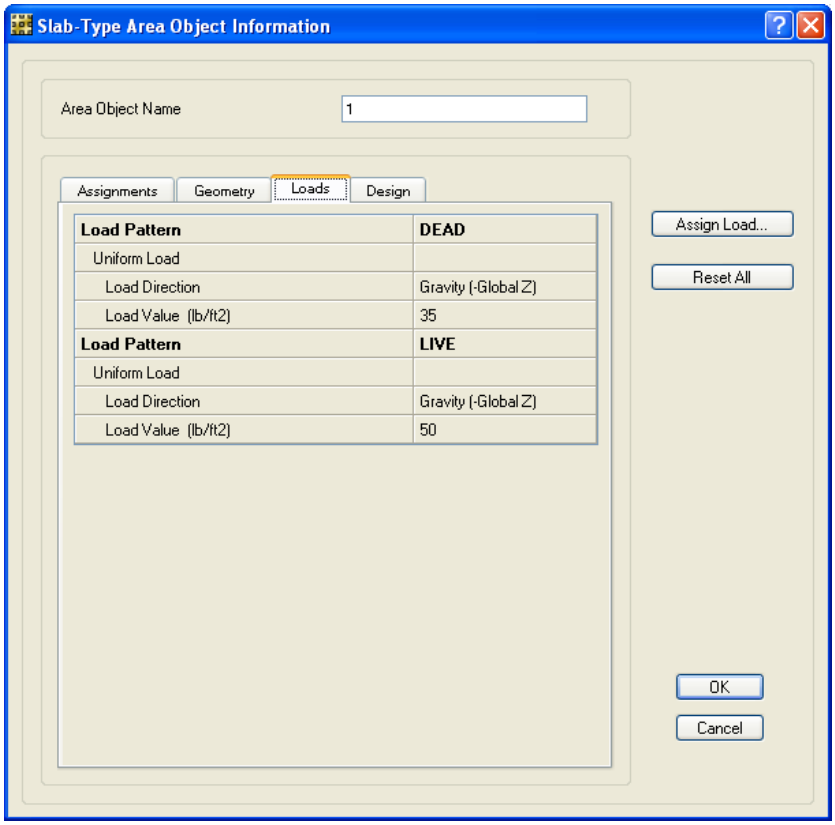


Figure 86 Slab-Type Area Object Information form

Step 9 Run the Analysis and Design

In this Step, the analysis and design will be run.

A. Click the **Run menu > Run Analysis & Design** command to start the analysis. The program will create the analysis model from your object-based SAFE model, and will display information in the status bar in the lower left-hand corner as the analysis and design proceeds. Additional information about the run may be accessed at a later time using the **File menu > Show Input/Output Text Files** command and selecting the filename with a *.LOG* extension.

B. When the analysis and design are finished, the program automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the **Options menu > Lock/Unlock Model** icon  appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results.

Step 10 Graphically Review the Analysis Results

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

A. Click the **View menu > Set Default 3D View** command to display the deformed shape for the DEAD load case in 3D.

B. Click the **Start Animation** button in the lower right-hand corner of the display to animate the deformed shape. Speed of the animation may be adjusted by using the slider control adjacent to the button. Click the **Stop Animation** button to end the animation.

C. Click the **Display menu > Show Deformed Shape** command to access the Deformed Shape form shown in Figure 87.

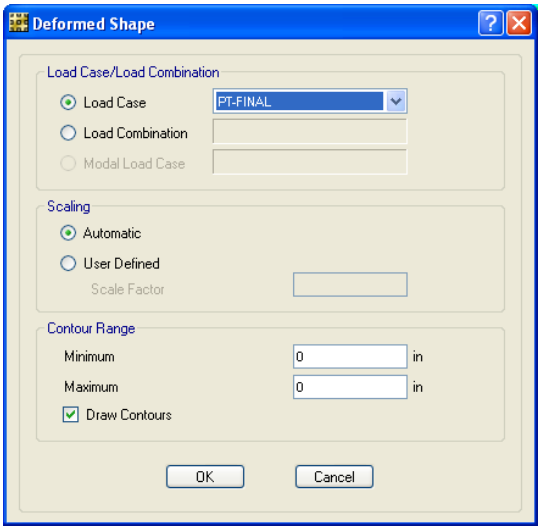


Figure 87 Deformed Shape form

D. In the Load Case/Load Combination area, select the *Load Case* option.

E. Select *PT-FINAL* from the Load Case drop-down list. Note that in addition to the DEAD, LIVE, PT-FINAL and PT-TRANSFER load cases that were previously defined, the program has automatically created a PT-FINAL-HP load case for hyperstatic analysis.

F. Select the *Automatic* option in the Scaling area.

G. Check the *Draw Contours* checkbox in the Contour Range area.

H. Click the **OK** button to generate a 3-D deformed shape with contours for the PT-FINAL load case.

I. Click the **Display menu > Show Slab Forces/Stresses** command to bring up the Slab Forces/Stresses form shown in Figure 88.

J. Select *PT-TRANSFER* from the Load Case drop-down list.

K. Select the *Stresses Top Face* option in the Component Type area.

L. Select the *S11* option in the Component area.

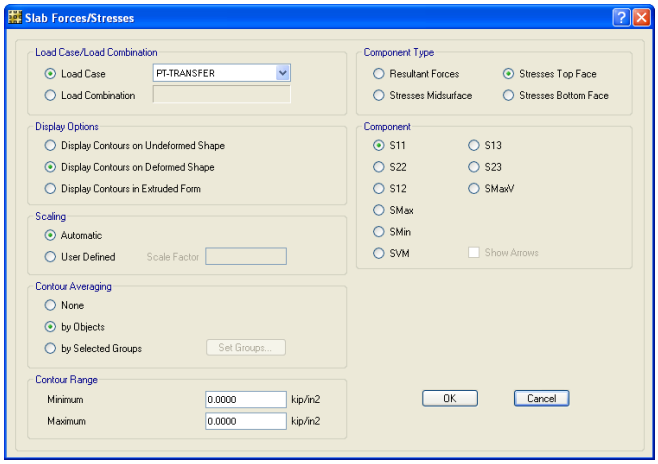


Figure 88 Slab Forces/Stresses form

M. Select the *Display Contours on Deformed Shape* option in the Display Options area.

N. Click the **OK** button to generate the stress diagram shown in Figure 89.

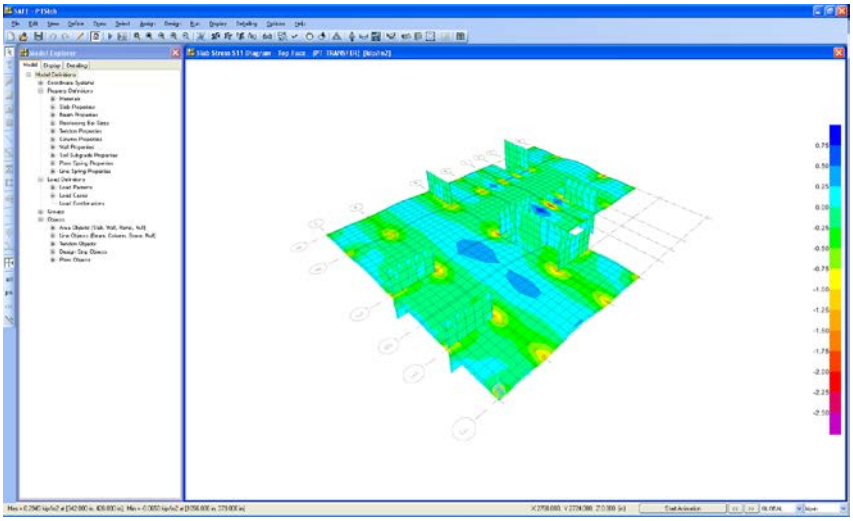


Figure 89 S11 Stress Diagram

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

O. Click the **Display menu > Show Undeformed Shape** command to clear the display of the stress diagram.

P. Click the **View menu > Set Plan View** command to return to the Plan View.

Step 11 Design Display

In this Step, design results for the slab will be displayed. Note that the design was run along with the analysis in Step 9. Design results are for the ACI 318-08 code, which was selected in Step 1. Design preferences may be reviewed or changed by going to the **Design menu > Design Preferences** command (some design preferences are also set on the section property data forms); be sure to re-run the analysis and design (Step 9) if changes to the preferences are made.

A. Click the **Display menu > Show Slab Design** command to access the Slab Design form shown in Figure 90.

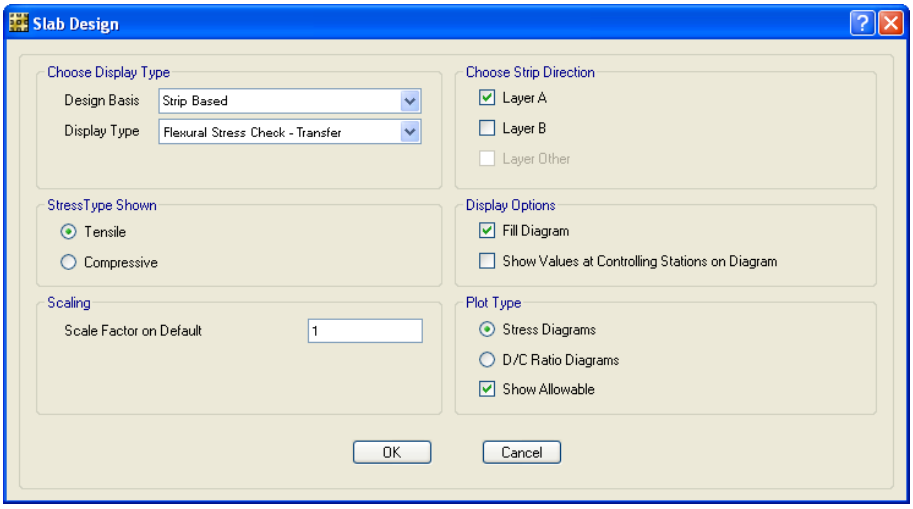


Figure 90 Slab Design form

B. In the Choose Display Type area, select *Strip Based* from the Design Basis drop-down list and *Flexural Stress Check - Transfer* from the Display Type drop-down list.

C. In the Choose Strip Direction area, check the *Layer A* checkbox and uncheck the *Layer B* checkbox. This will display the stress check results in the Layer A (X) direction only.

D. In the Stress Type Shown area, select the *Tensile* option.

E. Click the **OK** button to leave the Slab Design form and display the stress check results. The tensile stresses at transfer are displayed; if *failed* is displayed, the stress at that location exceeds the allowable P/T Stress Check ratio specified on the Design Preferences form accessed using the **Design menu > Design Preferences** command.

   Positioning the cursor at any location on a Layer A design strip causes the top and bottom stress values to be displayed in the lower left corner of the window and at the cursor.

F. To view the stress check results in the Layer B direction, click the **Display menu > Show Slab Design** command to display the Slab Design form.

G. In the Choose Display Type area, select *Flexural Stress Check – Long Term* from the Display Type drop-down list.

H. In the Choose Strip Direction area, uncheck the *Layer A* checkbox and check the *Layer B* checkbox.

I. In the Stress Type Shown area, select the *Compressive* option.

J. Click the **OK** button to leave the Slab Design form and display the long-term compressive stress check results for the Layer B direction. The view will be updated to that shown in Figure 91. Again, positioning the cursor anywhere on the strips will result in the display of the stress check values in the lower left-hand corner of the SAFE window.

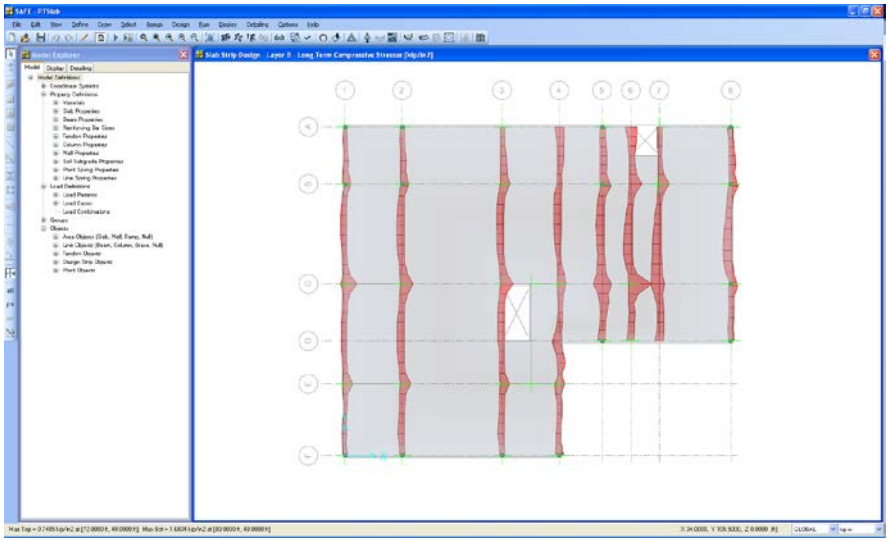


Figure 91 Layer B Long-Term Compressive Stresses

Step 12 Run Detailing

In this Step, detailing will be run and displayed. Detailing may be run only after analysis and design are complete.

A. Click the **Detailing menu > Detailing Preferences** command to display the Detailing Preferences form shown in Figure 92.

   Use this form to set the regional standards, to control how dimensioning is displayed, to manage reinforcing bar notation, and to select the units for material quantity takeoffs.

B. Review the settings on this form (we will accept the default selections), and then click the **OK** button to close the form.

C. Click the **Detailing menu > Slab/Mat Reinforcing Preferences** to display the Slab/Mat Detailing Preferences form shown in Figure 93.

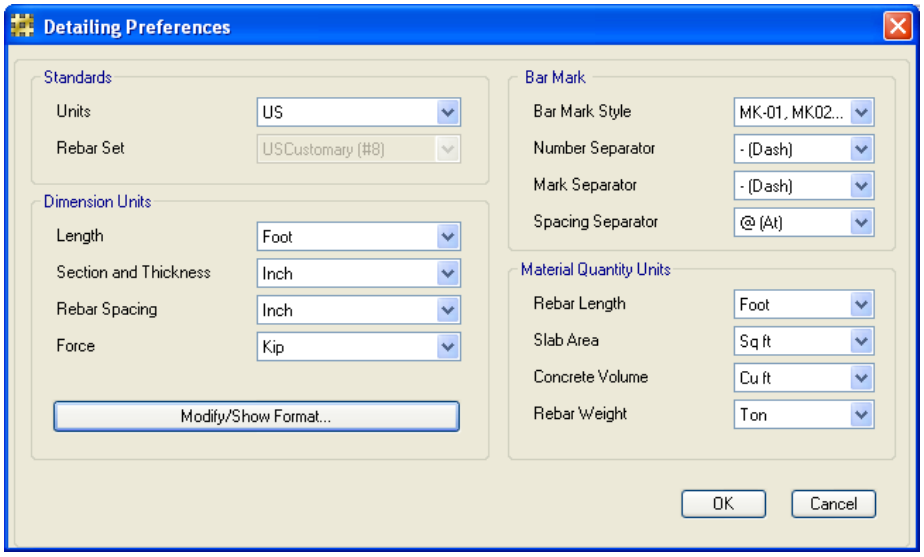


Figure 92 Detailing Preferences form

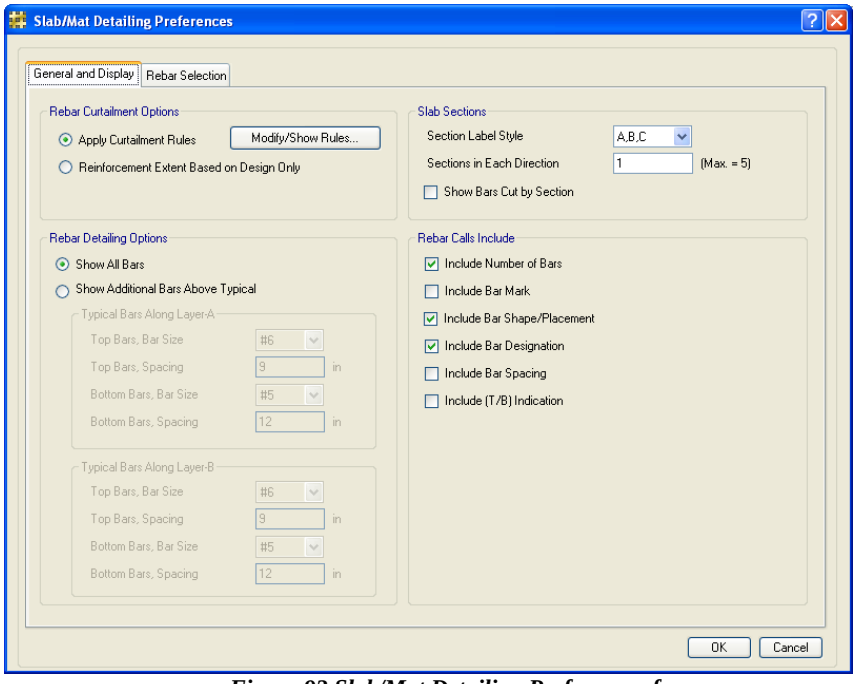


Figure 93 Slab/Mat Detailing Preferences form

D. Click the *General and Display* tab. On this tab review or alter the rebar curtailment, detailing and callout options, as well as set how sections should be cut. We will accept the default settings.

E. Click the *Rebar Selection* tab and review or change the rebar selection rules, preferred sizes, minimums and reinforcing around openings. We will accept the default settings.

F. Click the **OK** button to accept the selections and close the form.

G. Click the **Detailing menu > Drawing Sheet Setup** command to display the Drawing Sheet Setup form. The sheet size, scales, title block and text sizes can be reviewed and changed using this form. We will accept the default settings.

H. Click the **OK** button to close the form.

I. Review the line thicknesses and styles by clicking the **Detailing menu > Drawing Format Properties** command.

J. Click the **OK** button to accept the selections and close the form.

K. Now that the detailing preferences and drawing setup options have been reviewed, click the **Run menu > Run Detailing** command to generate the detailing drawings. A framing plan is displayed when detailing is complete.

L. Click the **Detailing menu > Show Detailing** command to access the Display Detailing Item form shown in Figure 94.

M. Select the *Drawing* option.

N. Select *Tendon Layout Plan* from the Drawing drop-down list.

O. Click the **OK** button to leave the Display Detailing Item form and display the selected drawing shown in Figure 95.

P. Clicking on the *Detailing* tab in the Model Explorer and expanding the Views and Drawing Sheets trees also provides access to detailing drawings and views.

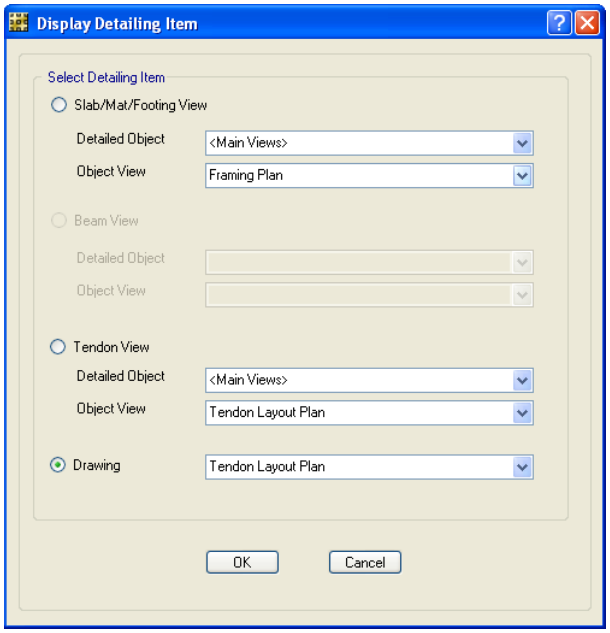


Figure 94 Display Detailing Item form

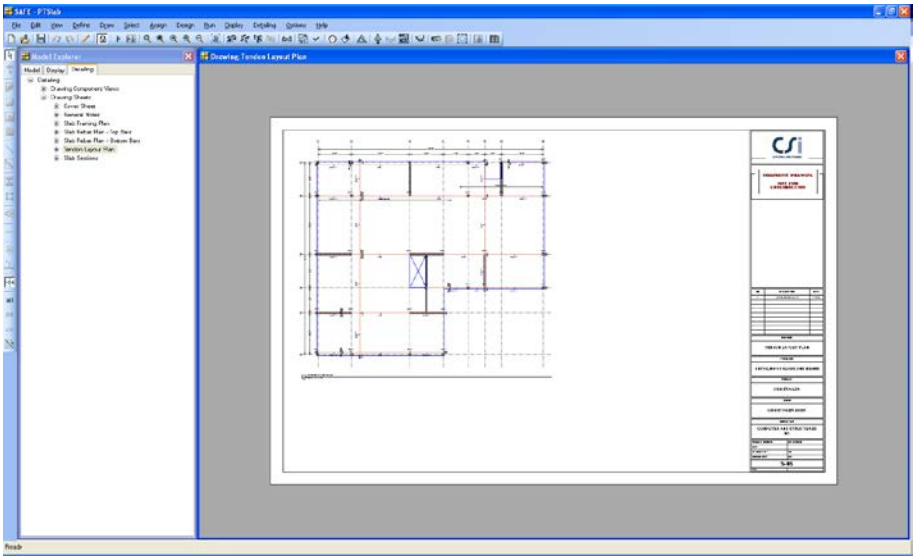


Figure 95 Tendon Layout Plan

Q. Click the **Display menu > Show Undeformed Shape** command to return to the model.

Step 13 Create Report

In this Step, a report describing model input and output results will be created.

A. Click the **File menu > Report Setup** command to display the Report Setup Data form.

B. In the Report Output Type area, make sure that the *RTF File* option is selected.

C. In the Report Items area, uncheck the *Include Hyperlinked Contents* checkbox.

D. Click the **OK** button to leave the Report Setup Data form.

E. Click the **File menu > Create Report** command to display the Microsoft Word Rich Text File Report form.

F. Type **PTSlab** in the File name edit box and click the **Save** button. A report, with a cover similar to that shown in Figure 96 should be displayed in your word processor, and will be saved to your hard disk.

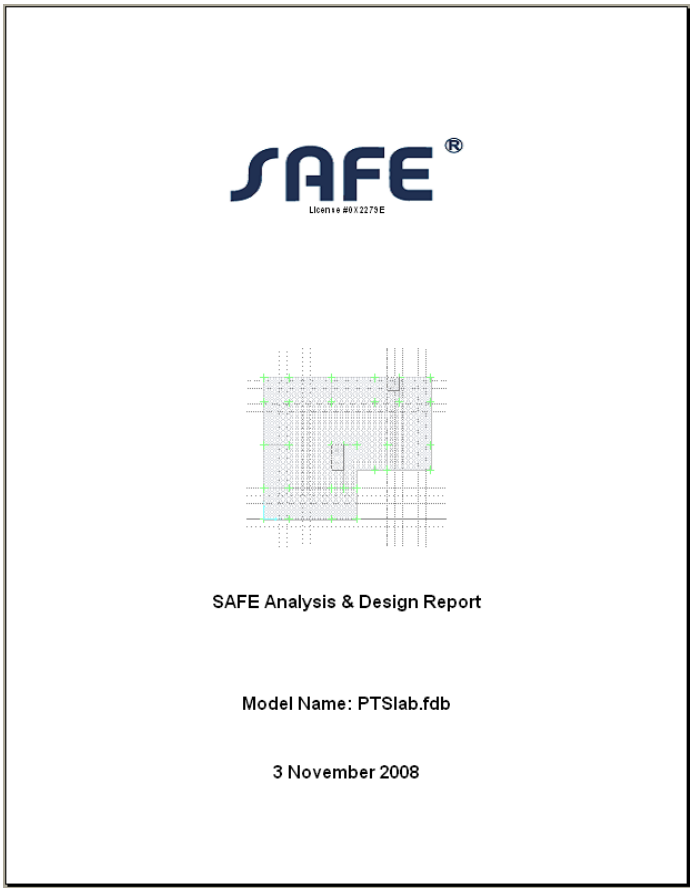
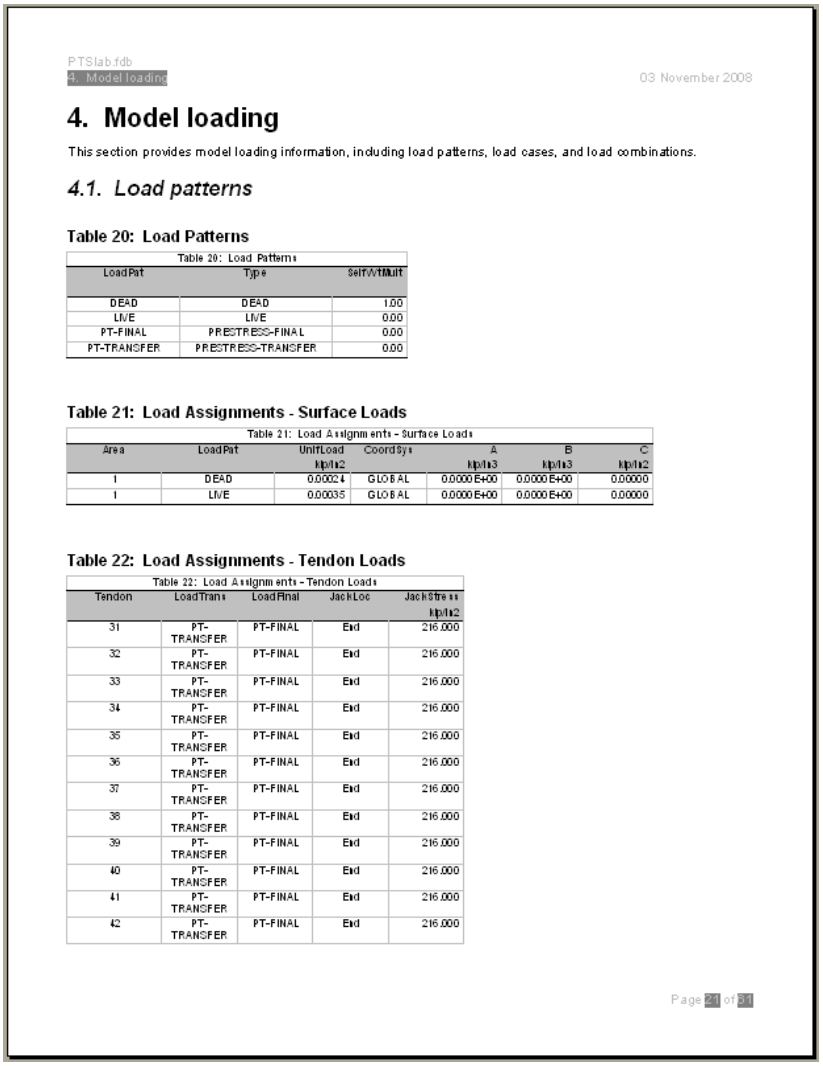


Figure 96 Cover Page for Report

G. Scroll through the report to find tables that list geometry and properties, analysis results and design information, such as that shown if Figure 97.

H. Close your word processor and return to the SAFE program.

I. Click the **File menu > Save** command to save your model one last time.



PTSlab.fdb
4. Model loading
03 November 2008

4. Model loading

This section provides model loading information, including load patterns, load cases, and load combinations.

4.1. Load patterns

Table 20: Load Patterns

Table 20: Load Patterns

LoadPat	Type	SelfWtMult
DEAD	DEAD	1.00
LIVE	LIVE	0.00
PT-FINAL	PRESTRESS-FINAL	0.00
PT-TRANSFER	PRESTRESS-TRANSFER	0.00

Table 21: Load Assignments - Surface Loads

Table 21: Load Assignments - Surface Loads

Area	LoadPat	UnifLoad	CoordSys	A	B	C
		kip/in2		kip/in3	kip/in3	kip/in2
1	DEAD	0.00024	GLOBAL	0.0000E+00	0.0000E+00	0.00000
1	LIVE	0.00035	GLOBAL	0.0000E+00	0.0000E+00	0.00000

Table 22: Load Assignments - Tendon Loads

Table 22: Load Assignments - Tendon Loads

Tendon	LoadTrans	LoadFinal	JackLoc	JackStress
				kip/in2
31	PT-TRANSFER	PT-FINAL	End	216.000
32	PT-TRANSFER	PT-FINAL	End	216.000
33	PT-TRANSFER	PT-FINAL	End	216.000
34	PT-TRANSFER	PT-FINAL	End	216.000
35	PT-TRANSFER	PT-FINAL	End	216.000
36	PT-TRANSFER	PT-FINAL	End	216.000
37	PT-TRANSFER	PT-FINAL	End	216.000
38	PT-TRANSFER	PT-FINAL	End	216.000
39	PT-TRANSFER	PT-FINAL	End	216.000
40	PT-TRANSFER	PT-FINAL	End	216.000
41	PT-TRANSFER	PT-FINAL	End	216.000
42	PT-TRANSFER	PT-FINAL	End	216.000

Page 21 of 31

Figure 97 Typical Report information

Congratulations! You've successfully created, analyzed, designed, detailed, and reviewed a SAFE post-tensioned concrete model.