Sog_tutorial_cover10
041505
MNL422

ADAPT

STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-BUILDER

SOG TUTORIAL

POST-TENSIONED FOUNDATION SLABS ON EXPANSIVE OR COMPRESSIBLE SOIL

Dr Bijan O Aalami
Professor Emeritus, San Francisco State University
Structural Engineer, California

Affiliate Member

POST-TENSIONING INSTITUTE

E-Mail support@adaptsoft.com **www.adaptsoft.com**
1733 Woodside Road, Suite 220, Redwood City, California, 94061, USA, Tel: (650) 306-2400 Fax (650) 364-4678

Sog_tutorial_toc10
070103
MNL422

LIST OF CONTENTS

1 OVERVIEW

The following tutorial illustrates how to create, analyze and design the example in Appendix A.7 of reference [PTI, 1996]. The tutorial is organized as follows:

- Model Generation
    - Geometry
    - Material
    - Post-tensioning
    - Loading
    - Mesh Generation
- Center Lift Condition
    - Soil Foundation
    - Analysis
    - Validation of Solution
    - Design
- Edge Lift Condition
    - Soil Foundation
    - Applied Displacement
    - Analysis
    - Validation of Solution
    - Design

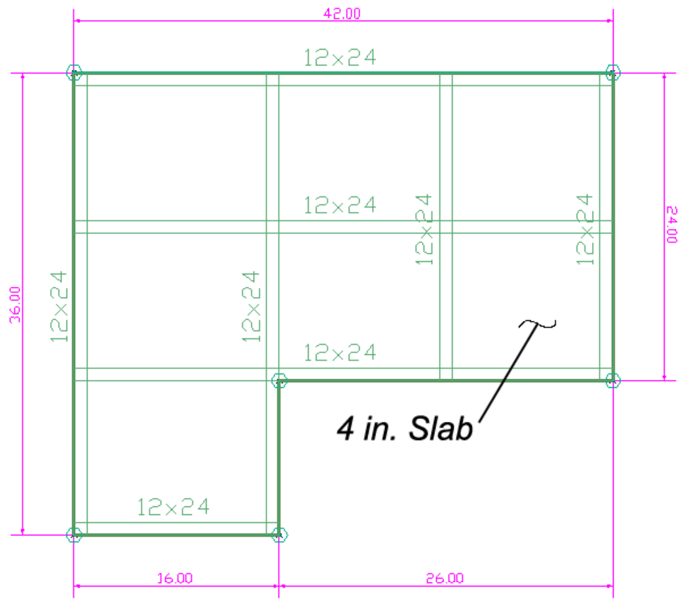
For a more comprehensive explanation of the solution for each swell mode, refer to *SOG User Manual*.

2 SOG Model Generation

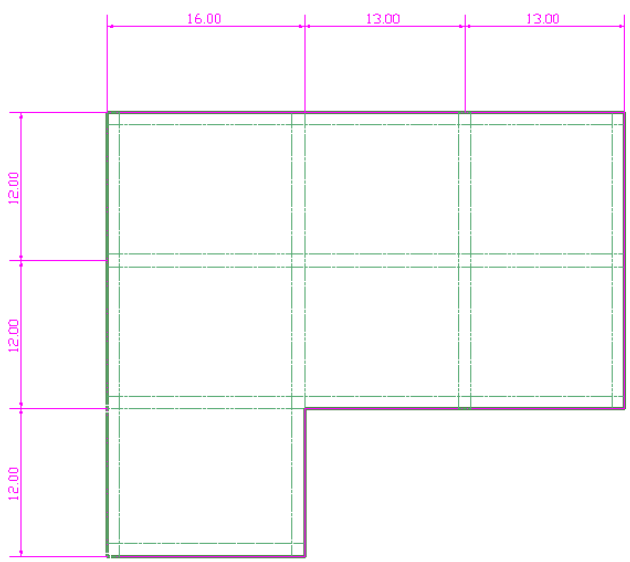
2.1 Create and Edit Structural Model

2.1.1 Geometry

The PTI Example A.7 consists of a Slab Region with a 4-inch thickness. Stiffening Beams of dimensions 12”x 24” are placed in both directions with the spacing shown in **Figs.2.1-1(a)** and **(b)**.



(a) Slab and Beam Dimensions



(b) Beam Spacing

FIGURE 2.1-1 FOUNDATION GEOMETRY

2.1.2 Generate the Slab Region

To manually create the Slab Region, a three-foot grid can be used as a guide to snap to. To setup a grid, use the following procedure.

1. From the *Snap* toolbar, click on the *Grid Settings* button. The Grid Settings dialog box will appear as shown below.

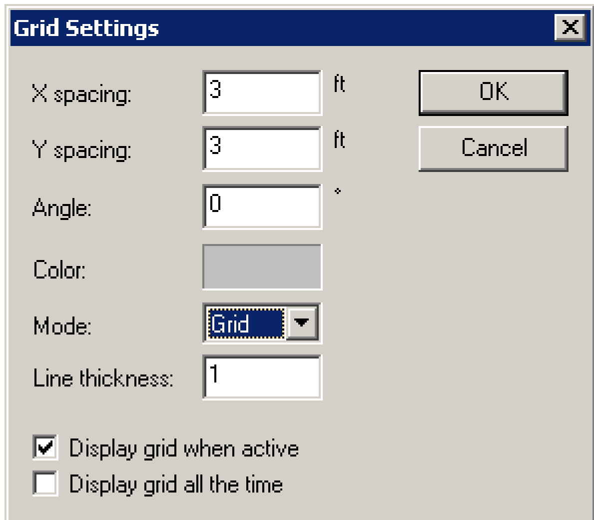


Figure 2.1.2-1 *Grid Settings* Dialog Box

2. Change the X-spacing and Y-spacing to 3 ft, check mark the "Display grid when active" setting and click *OK*.
3. From the Snap toolbar, click on the Snap to Grid button. A 3'x 3' grid will appear.
4. In the *View* menu, click on the *Display WCS*. When manually creating a model, you will want to make the project origin and the WCS (World Coordinate System) coincide.
5. From the *Camera and Viewports* toolbar, click on the *Top View* button.
6. From the Build menu, select *Display Modeling* toolbar, click on the *Create Slab Region* button. Although two of the Slab vertices (16,0 ft and 16,12 ft) do not snap directly to the preset grid, you can snap these two vertices to near the intended location (18,0 ft and 18,12 ft) during the Slab creation mode. After the Slab has been created, the X-coordinate can be manually input (16 ft) in the Slab Properties dialog box.
7. The Slab Region is generated by snapping to the coordinates shown in the following figure. Press "C" key to close the Slab Region.

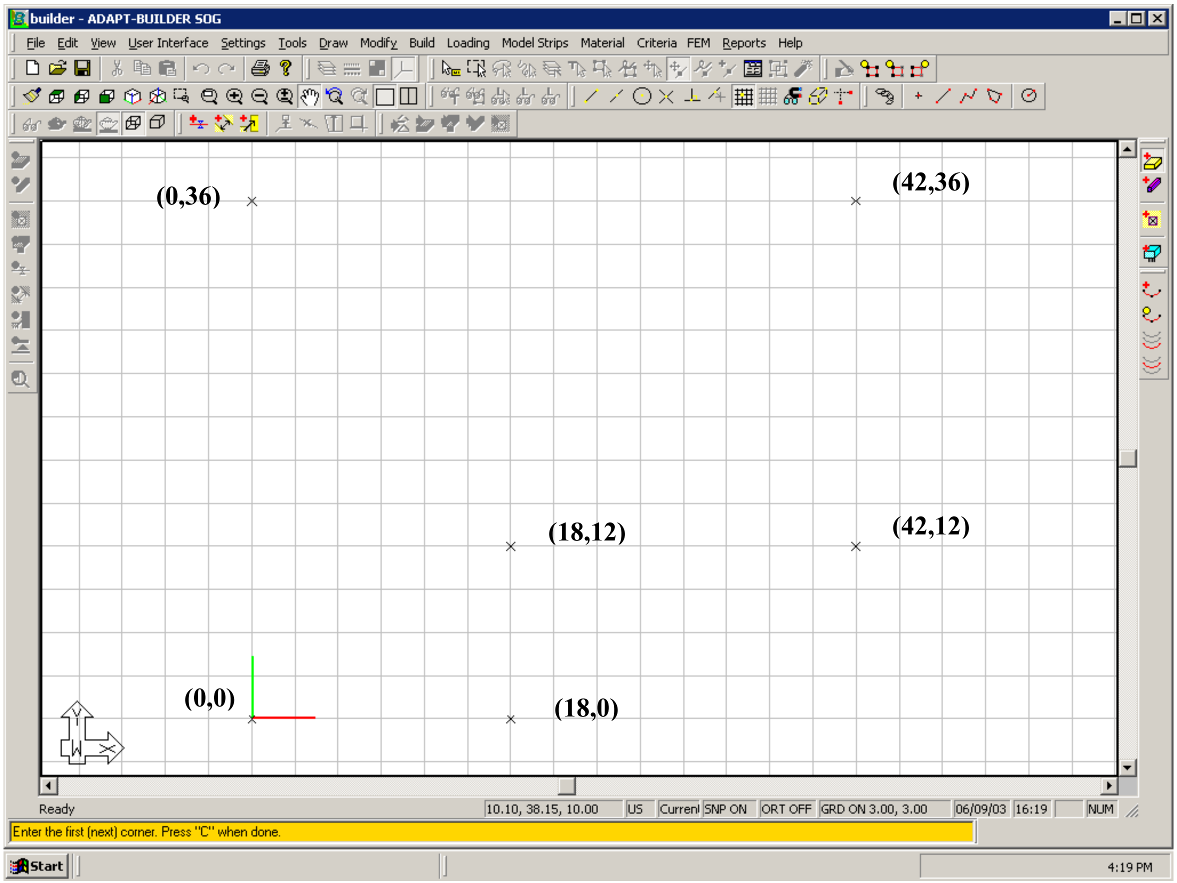


Figure 2.1.2-2 Grid Coordinates For Snapping

Double click on the Slab Region to open its properties box or select the Slab Region and open the properties box by clicking on the *Item's Properties* button.

8. Change the following parameters (**Fig. 2.1.2-3**), then click on the button to accept the changes:

- Thickness 4 inch
- Coordinates

#	X
2	16
3	16

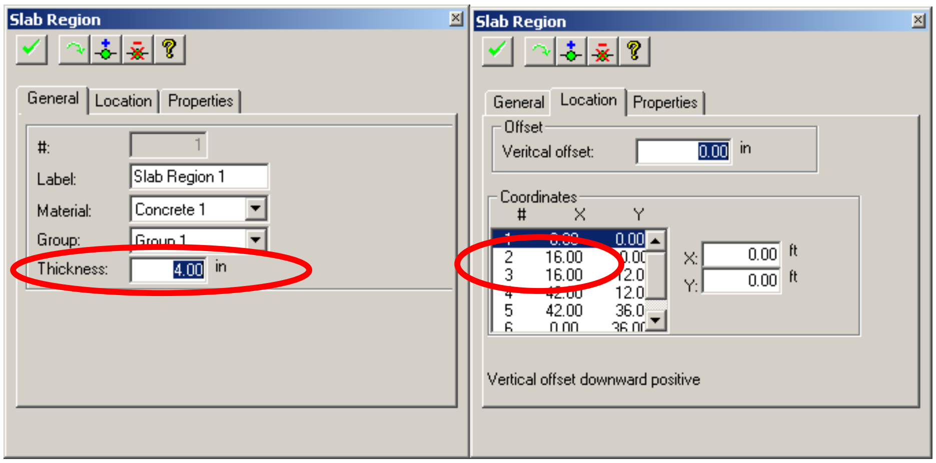


Figure 2.1.2-3 Modifications to Slab Region

2.1.3 Generate the Stiffening Beams

Stiffening Beams of dimensions 12"x20" will be used and offset 4" below the Slab soffit, giving a total height of 24".

1. From the Build toolbar, click on the *Create Beam* tool. Next click on the Item's Properties button. Change the following parameters (**Fig. 2.1.3-1**), then click on the button to accept the changes:

   - Cross-section
     Width: 12 in
     Depth: 20 in
   - Vertical Offset 4 in

Note: By changing the parameters before the Beam is created, the modifications become the default values. Therefore, all Beams created hereafter will have these specified dimensions and offset.

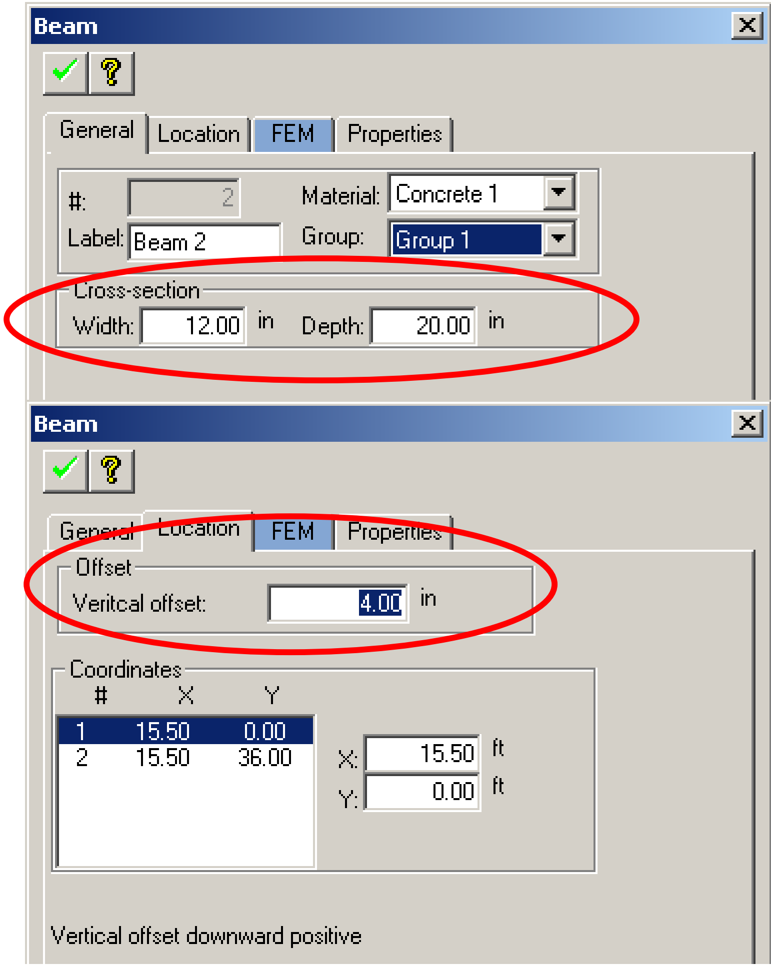


Figure 2.1.3-1 Modifications to Slab Region

2. Set Modeler's object snapping properties so that Snap *to Intersection* button is active (the icon will be highlighted after clicking the mouse). Turn off all other snapping tools.
3. Create the perimeter Beams by snapping to the Slab edge as shown in **Fig. 2.1.3-2.** Only create the Beams shown; the rest of the Beams will be generated using the *All Transformations* tool in the *Copy/Move* Toolbar.

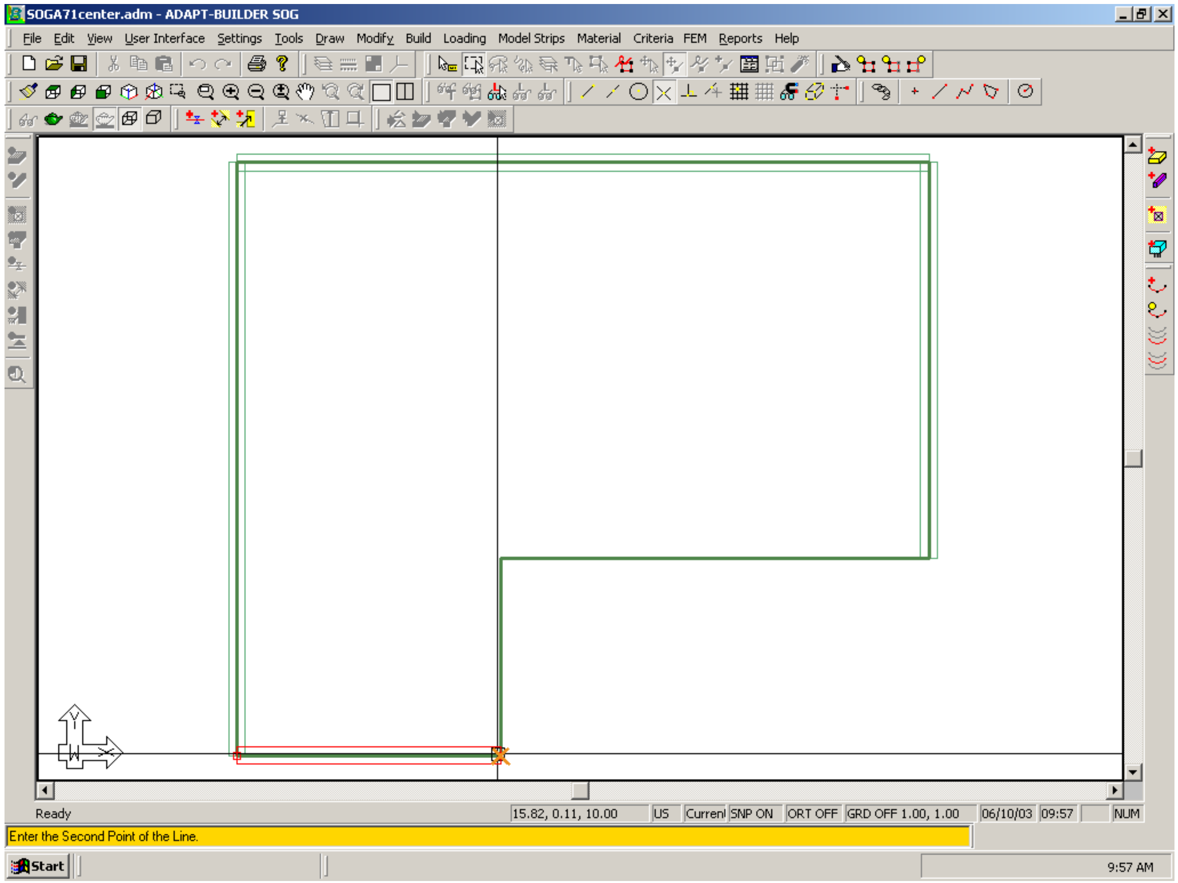


Figure 2.1.3-1 Create Perimeter Beams by Snapping to Edge of Slab

4. To copy a Beam, select a Beam of similar length and click the *All Transformations* tool. The *Copy-Move* dialog box will appear. Enter the appropriate X and Y offset and press copy. Repeat this step for the remaining Beams.

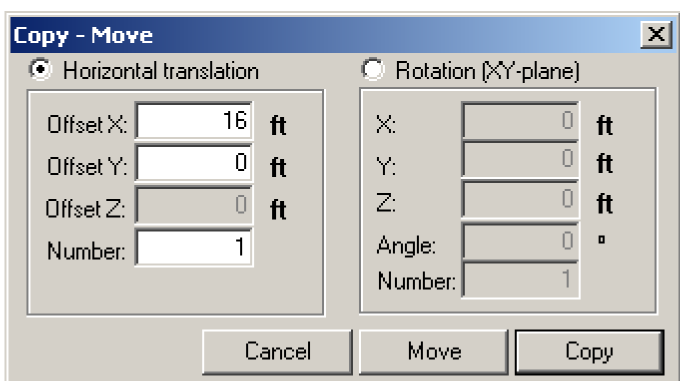


Figure 2.1.3-2 Copy-Move Dialog Box

5. To align the Beams flush with the Slab edge, click on the *Align Structural Components* tool located in the *Modeling* toolbar. Select

the Beam to be aligned by clicking on it. The following dialog box will appear (**Fig. 2.1.3-3**).

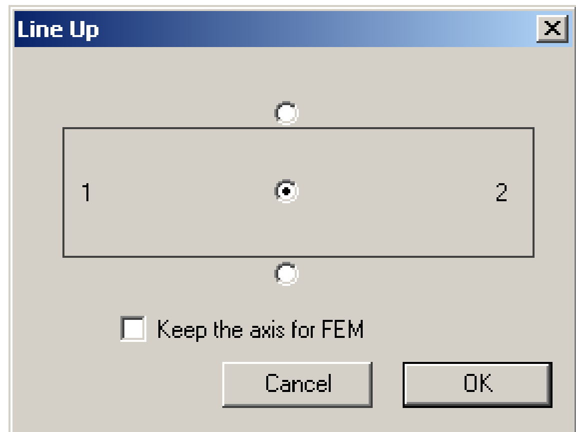


Figure 2.1.3-3 LineUp Dialog Box

Choose the side that you believe is the correct shifting direction according to the direction that the Beam was created and click *OK*. The program will shift the Beam in that direction and then ask you if the direction is correct. If you click yes, the Beam is placed at the shown location. If you click no, the program automatically shifts the Beam to the opposite side as shown below.

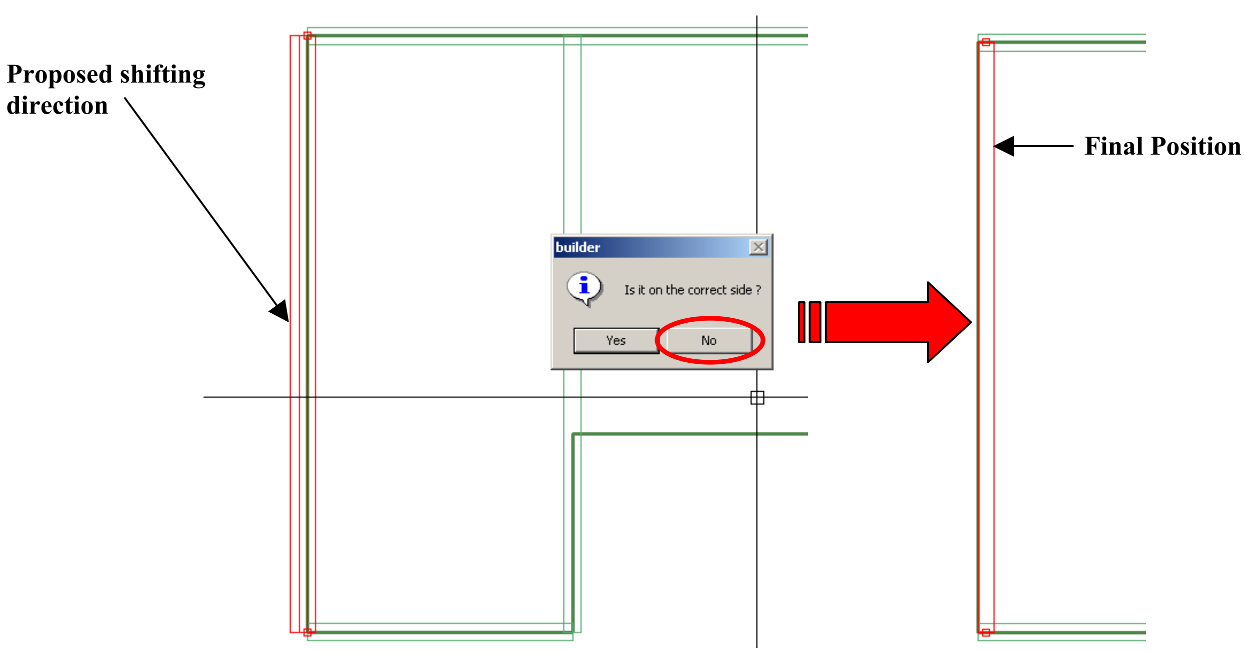
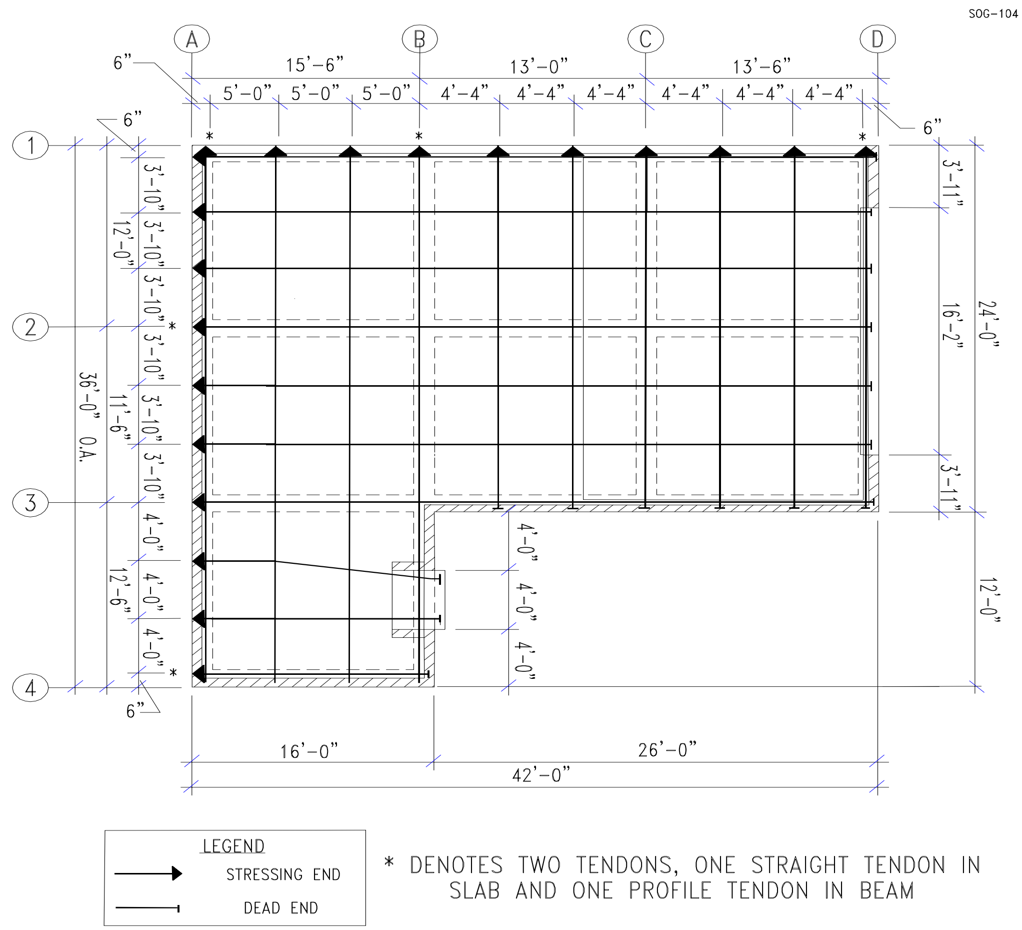


Figure 2.1.3-4 Beam Shift

6. Align the remaining Beams, according to the previous step.

2.2 Generate Beam and Slab Tendons

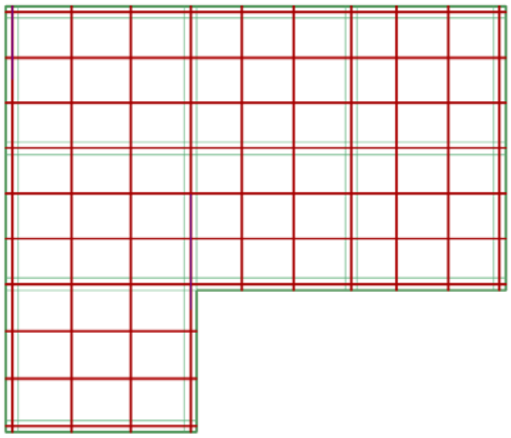
The Slab is reinforced with unbonded single strand (monostrand) post-tensioning Tendons. There are eight Tendons in the longitudinal direction and thirteen Tendons in the transverse direction (**Fig. 2.2-1 and 2.2-2**). The Tendons are straight (no profile) and are located at the mid-depth of Slab, 2 in. down from the top of Slab. The beam Tendons are draped and are located 3.25 in. from the bottom of the Beams; both the beam Tendons and the slab Tendons are eccentric with respect to the centroid of the ribbed Slab. The average precompression is 114 psi. The profile of beam Tendons is shown in **Fig. 2.2-3**.



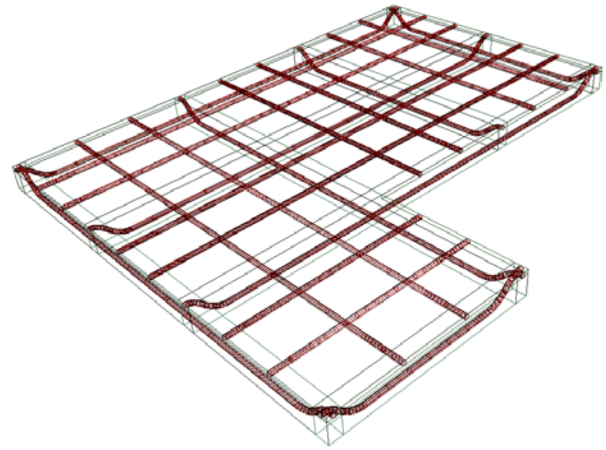
PLAN

PTI's SOG DESIGN EXAMPLE A.7

FIGURE 2.2-1



(a) Layout of Tendons in plan



(b) 3D-view of Tendon layout

FIGURE 2.2-2 TENDON LAYOUT

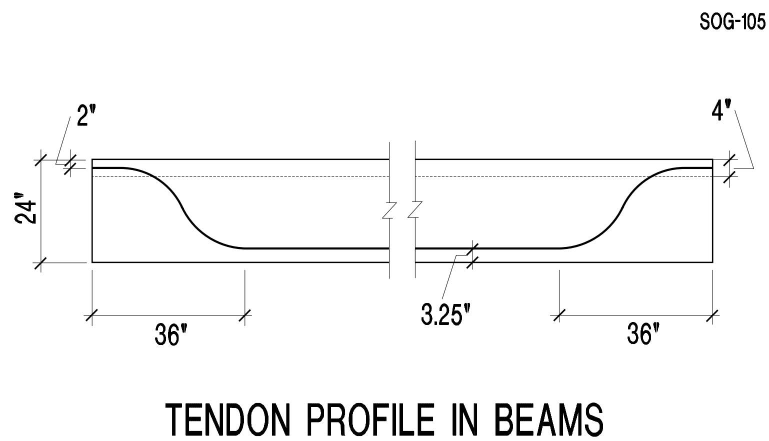


FIGURE 2.2-3

2.2.1 Beam Tendons

1. The beam Tendon profile is generated by creating three spans. The first and last spans are reversed curves 3ft from the Slab edge. The second span is straight. To create snapping guides for the first and last span, draw lines along the Slab edge and offset them inward (using the Manual Transformation 123) from the Slab edge by 3ft as shown in the diagram below (**Fig. 2.2.1-1**).

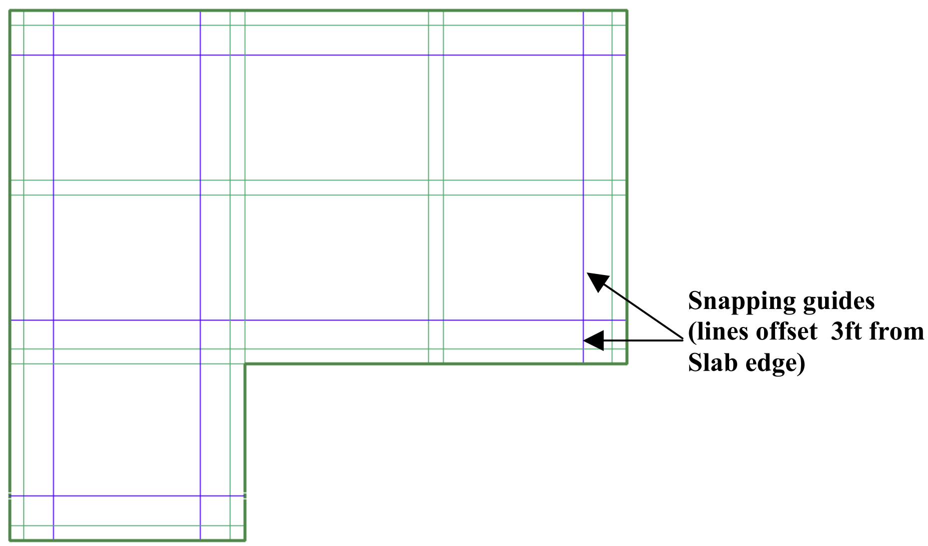


FIGURE 2.2.1-1 SNAPPING GUIDES FOR BEAM TENDONS

2. Set Modeler's object snapping properties so that *Snap to Endpoint* is active. Turn off all other snapping tools.
3. To create a beam Tendon, click on the *Create Tendon* tool.
4. Snap to the endpoint of the Beam as shown in **Fig 2.2.1-2**.
5. Next activate the *Snap to Perpendicular* tool and snap to the guideline located 3ft to the right of the Beam endpoint, as well as the guideline 3ft left of the Beam endpoint.
6. Reactivate the *Snap to Endpoint* tool and snap to the Beam endpoint. Press the "C" key to terminate the Tendon.

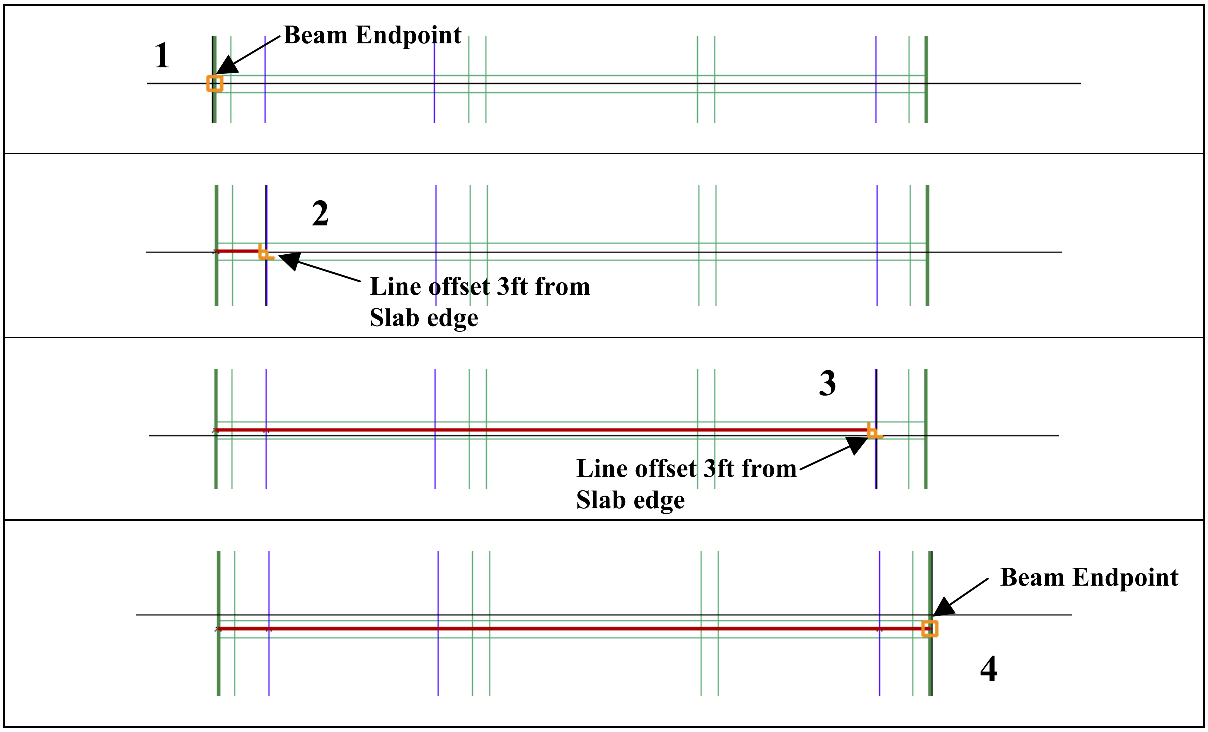


FIGURE 2.2.1-2 SNAPPING SEQUENCE FOR BEAM TENDONS

7. To generate the Tendon profile shown in **Fig. 2.2-3**, open the Tendon properties box by either double clicking on the Tendon or selecting the Tendon and then clicking the Item's Properties tool.

8. Select the *Shape/System/Friction* tab as shown in **Fig. 2.2.1-3**. Enter the following parameters for each span, then click on the button to accept the changes:

Span #	X1, X2, X3	Shape	CGS	Friction and System
Span1	0.49,0.50,0.49	Reversed Parabola	2, 11.37, 20.75	Unbonded
Span2	0.10,0.50,0.10	Reversed Parabola	20.75, 3.25, 20.75	Unbonded
Span3	0.49,0.50,0.49	Reversed Parabola	20.75, 11.37, 2	Unbonded

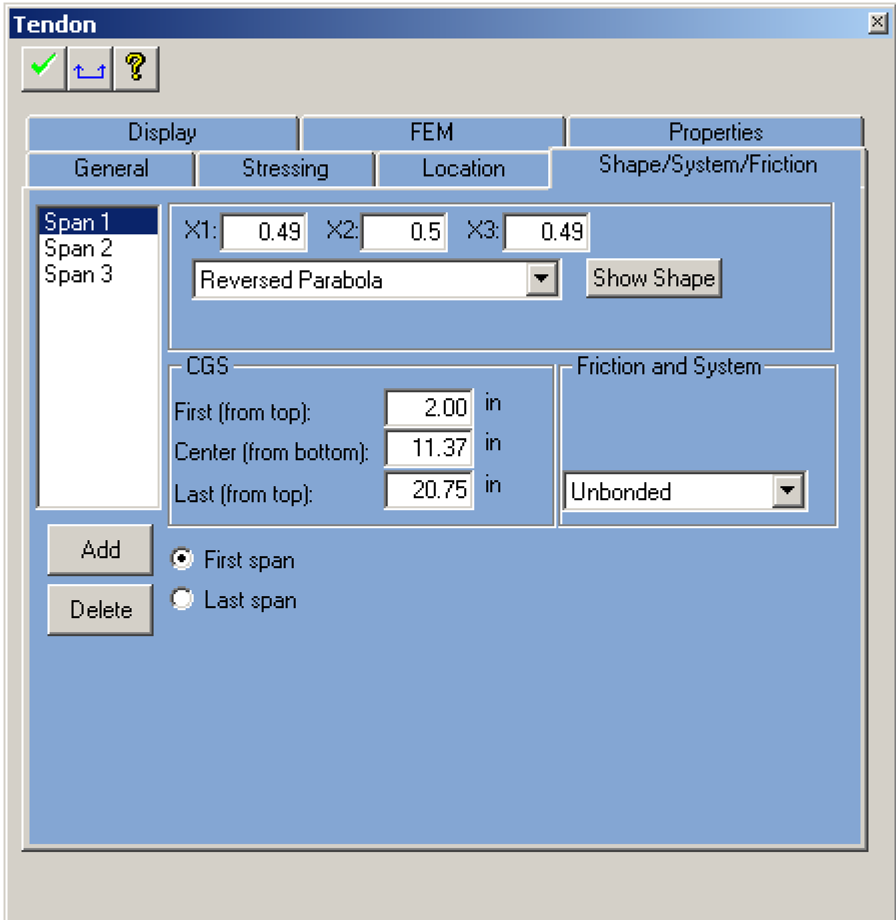


FIGURE 2.2.1-3 SNAPPING SEQUENCE FOR BEAM TENDONS

2.2.2 Slab Tendons

Slab Tendons are single span Tendons that have a CGS of 2" from top and bottom.

1. To quickly generate the slab Tendons, a master Tendon is created first with the correct profile.
2. Now Tendons can replicated from the master using the *All Transformation* tool as demonstrated in **Section 2.1.3** for Beam generation.

2.3 Set Material Properties for Concrete and Prestressing

2.3.1 Concrete Material Properties

1. To set the material properties for concrete, go to the *Material* menu and select *Concrete*. The following dialog box will appear.

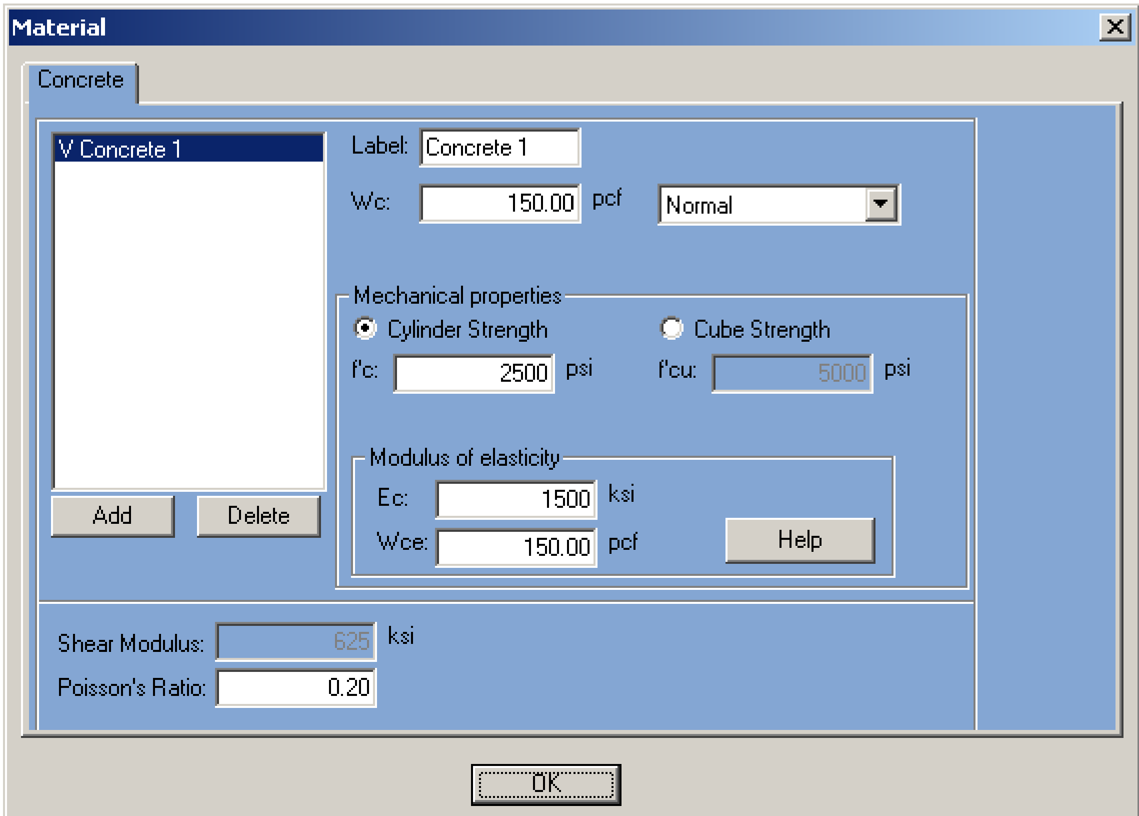


FIGURE 2.3.1-1 CONCRETE MATERIAL DIALOG BOX

2. Change the following parameters, then click *OK*.
   - f'c: 2500 psi
   - Ec: 1500 ksi

2.4 Apply Loading

2.4.1 Uniform Live Load

1. From the menu bar, click on *Loading* and select *Display Loading Toolbars* from the pull down menu.
2. Click on the Slab. A change in color indicates that this Slab was selected by the program.
3. Select *Patch Load Wizard* .
4. The dialog box shown in **Fig. 2.4.1-1** opens. Change the “Load case” to live load and the value to 0.04ksf. Note that the selfweight of the structure is calculated automatically by the program, using the geometry of the structural model and the unit weight defined by the user. The program has the conventional unit weight value of concrete as its default value.

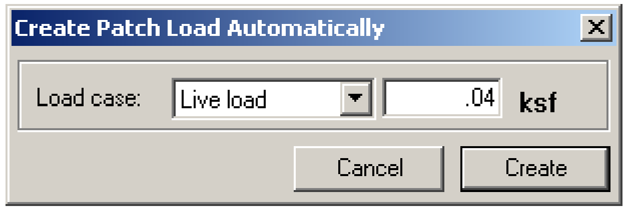


FIGURE 2.4.1-1 PATCH LOAD WIZARD DIALOG BOX

2.4.2 Perimeter Load

1. To apply the perimeter loading, click on the Slab. A change in color indicates that this Slab was selected by the program.
2. Select the *Line Load Wizard* tool.
3. The dialog box shown in **Fig. 2.4.2-1** opens. Change the value to 1.04 k/ft and click on *Create*.

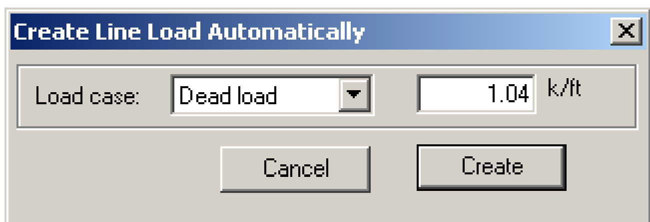


FIGURE 2.4.2-1 LINE LOAD WIZARD DIALOG BOX

2.4.3 Load Combination

1. From the menu bar, click on *Loading* and select *Load Combination* from the pull down menu. The *Combinations* dialog box appears.
2. By default the program automatically creates a load combination called *Basic Case,* which includes *Selfweight*. To add the dead load case to the combination, choose *Dead load* from the *Load cases* combo box. Leave the *Load factor* value as 1. Click *Add* under the *Combination parts*. Dead load will appear as part of the combination. Repeat this step for live load and prestressing (**Fig. 2.4.3-1**).
3. After all the load cases have been added in the *Combination list*, click *Save,* then *OK* to close the dialog box.

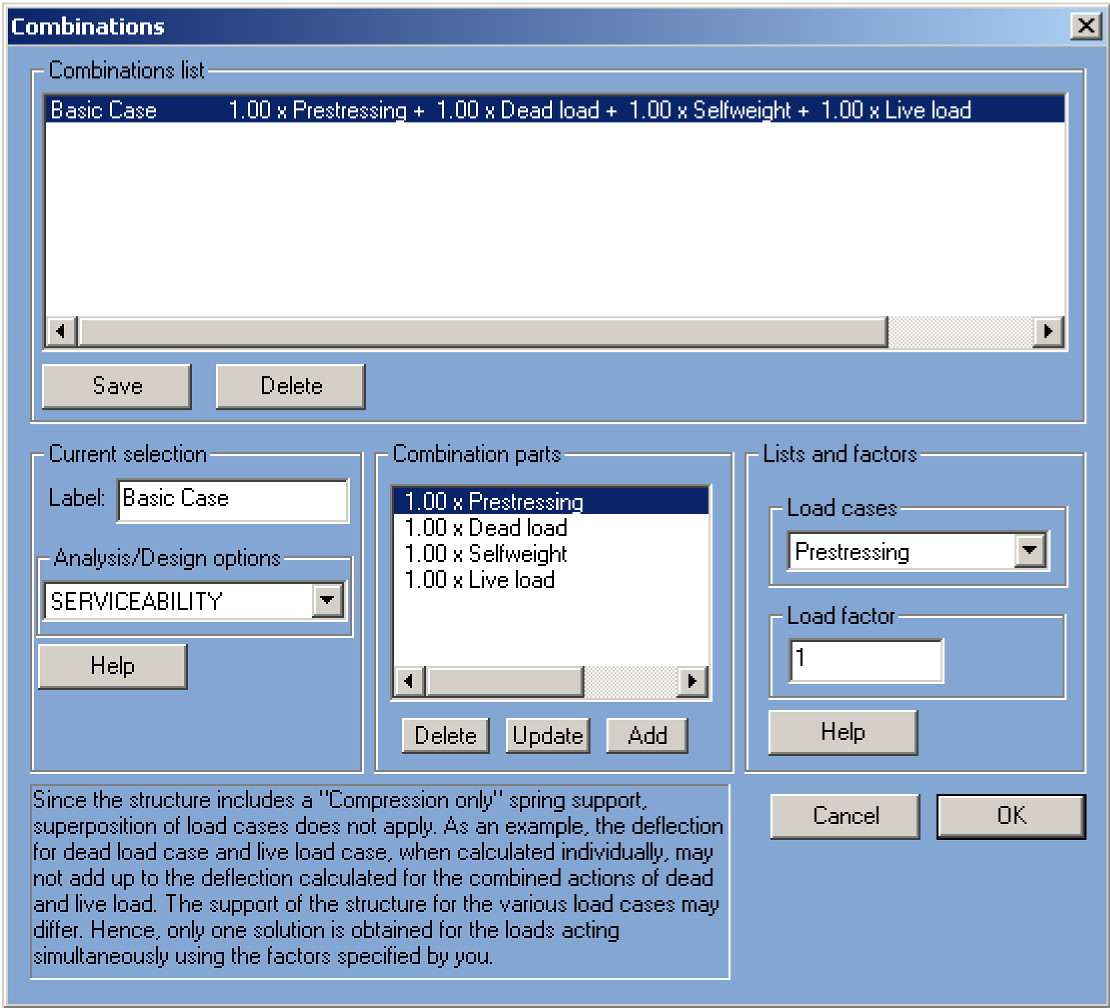


FIGURE 2.4.3-1 LINE LOAD WIZARD DIALOG BOX

2.5 Generate Mesh

From the menu bar, select *FEM* and click on *Display FEM Toolbars*, a pull down menu item. From the toolbars displayed, click on the button for Automatic Mesh Generation  to open the dialog box shown in **Fig. 2.5-1**, and accept the default values. After the completion of meshing, a message box shown in **Fig. 2.5-2** will open. Click *Yes* to view the meshing. This will show the automatic mesh generated by the program, **Fig. 2.5-3**.

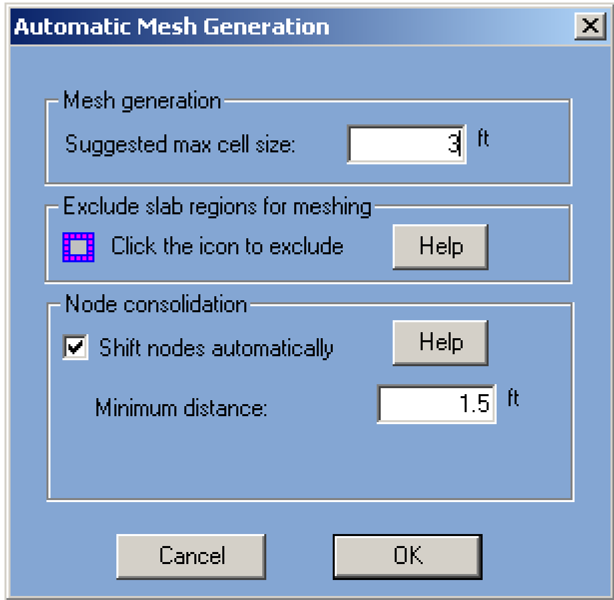


FIGURE 2.5-1 AUTOMATIC MESH GENERATION

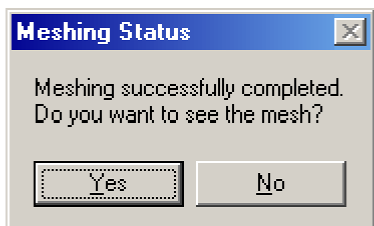


FIGURE 2.5-2 MESHING VIEW MESSAGE BOX

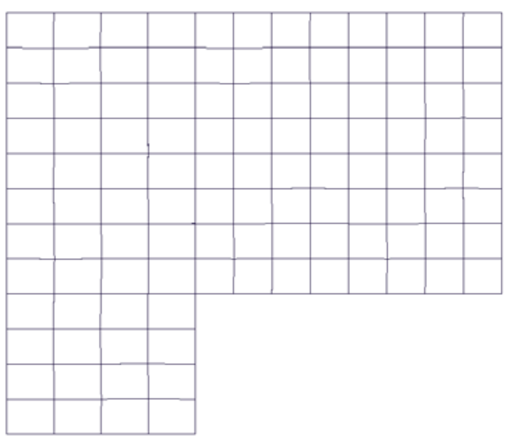


FIGURE 2.5-3 PLAN VIEW OF THE SLAB MESHING

2.6 Save Model as a Template for Both Soil Conditions

This general model will be used as a template for both center lift and edge lift condition. The model should be saved two times as:

- PTI_Example_Center_Lift_EX7.adm

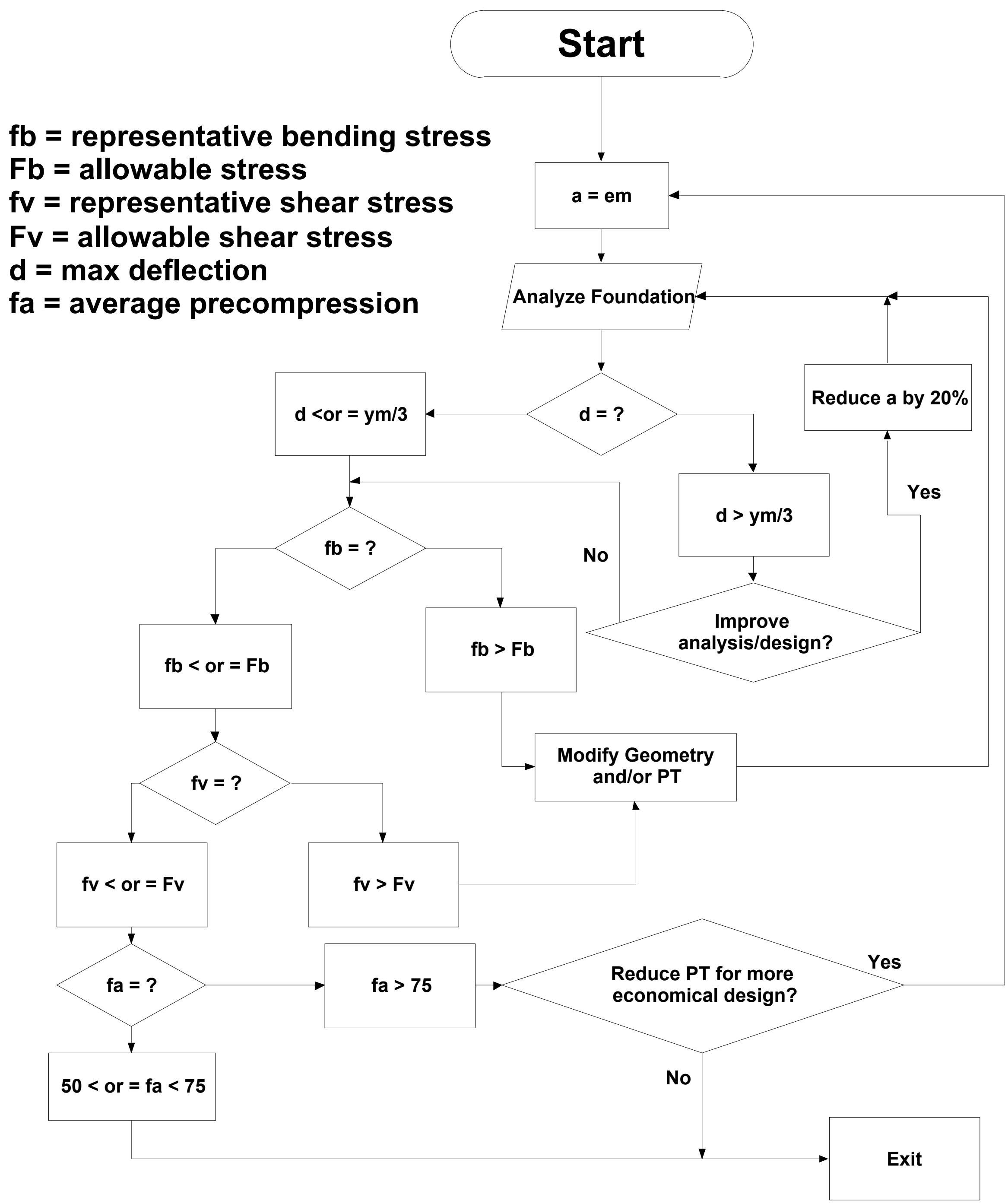
- PTI_Example_Edge_Lift_EX7.adm.

Note: The program does not allow spaces in file names.

3 Center Lift Condition

3.1 Flow Chart of Design of SOG for Center Lift Condition

Flow Chart of Center Lift Design



3.2 Create Soil Foundation

The soil foundation for the first iteration of center lift condition is placed at a distance equal to e_m from edge of Slab.

1. To generate the soil foundation, click the *Area Spring* tool. Using the *Snap to Intersection* tool, snap the Area Spring to the Slab vertices and press the "C" key to close the spring.
2. Double click on the Area Spring to open its properties box or select the Area Spring and open the properties box by clicking on the *Item's Properties* button (**Fig. 3.2-1**). Change the following parameters, then click on the button to accept the changes:

- kza (Bulk modulus of soil) 71 pci
- Coordinates

#	X	Y
1	4.5	4.5
2	4.5	31.5
3	37.5	31.5
4	37.5	16.5
5	11.5	16.5
6	11.5	4.5

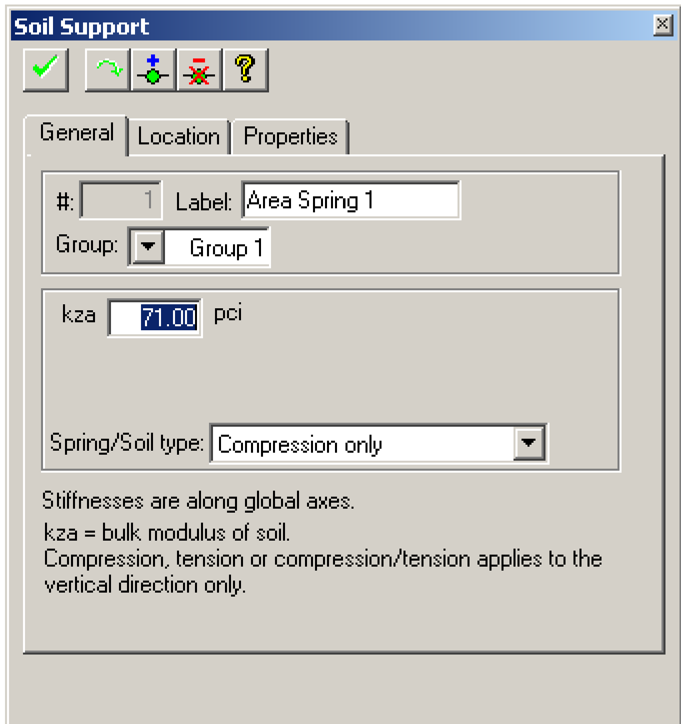


FIGURE 3.2-1 SOIL SUPPORT PROPERTIES BOX

3.3 Analyze and Verify Results

3.3.1 Convert Mesh into Elements and then Solve the Structure

From the pull down menu of *FEM*, select *Analyze Structure* icon. This will perform the finite element analysis of the structure and report its completion on the computer screen.

3.3.2 View the Analysis Results

From the *FEM* toolbar, click on the *View Analysis Result* icon. This will bring up the viewer screen, as shown in **Fig. 3.3.2-1**. Next, we will view the deflection contour of the Slab.

- Click on the "Load Cases/Combinations" tab on the bottom left of the screen.
- From the menu that opens, select "Basic Case".
- Click on the "Results" tab in the top left region of the screen.

- Check that d ≤ ym/3. Use the flow chart in Section 3.1 to help facilitate any modifications if d > ym/3.

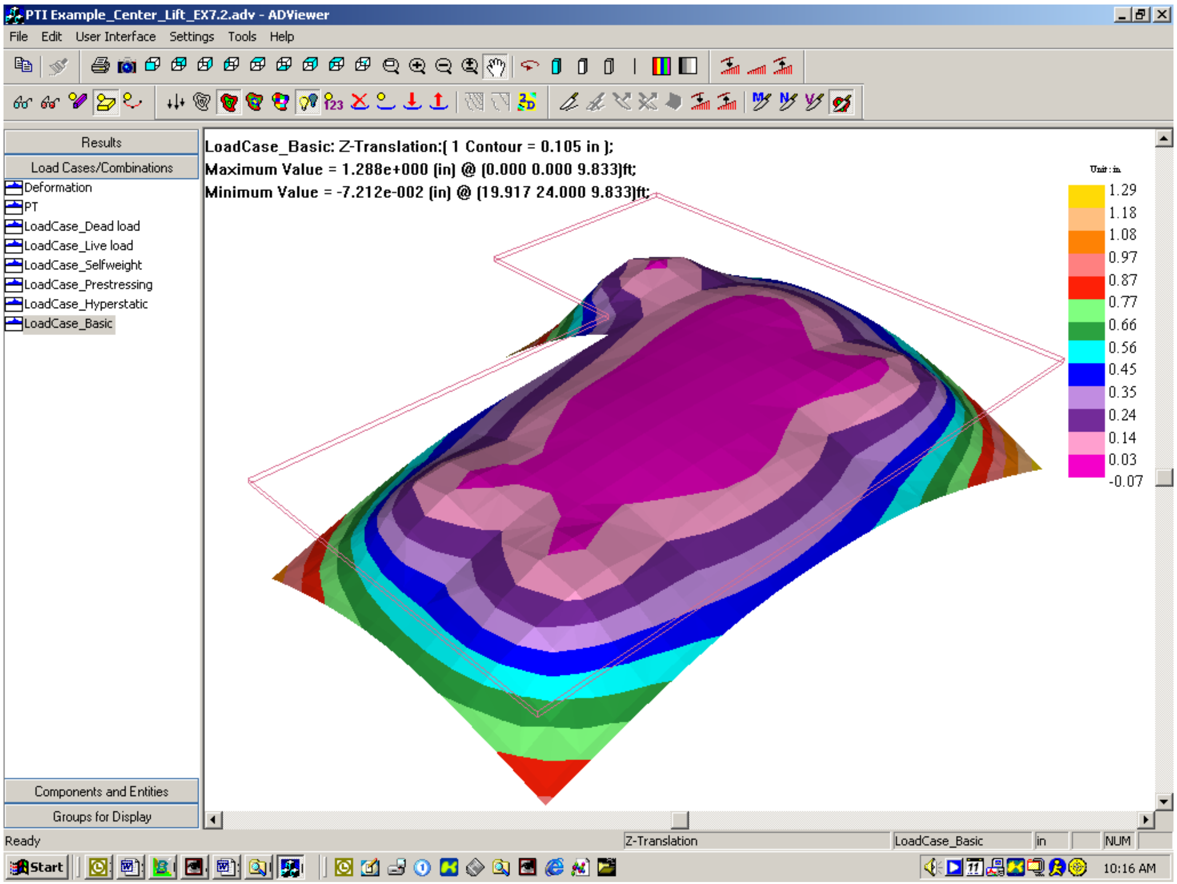


FIGURE 3.3.2-1 RESULTS VIEWER

3.4 Check Design for Stress, Shear and Deflection

Before checking the design stresses, shear and deflection, the analysis should be validated as laid out in the flowchart of **Section 3.1**. When a valid solution is obtained, the Slab is broken into design strips and design sections. The program then performs a stress check on each design section. Shears and deflection for each design section are also calculated. The following steps show you how to obtain the results of the design.

1. In order to obtain design strips, you must first create Support Lines and design strips in two orthogonal directions. **Section 1.2** of the *Modeler Tutorial* can be used as a guide in Support Line and design strip generation.
2. Click on the *Generate Design Sections Automatically* tool from the FEM pull down menu.

3. Design sections will be created automatically. **Fig. 3.4-1** shows an example of the Support Lines and the associated design strips for both X- and Y-direction. If the image does not appear, click on *Display Design Sections* button.

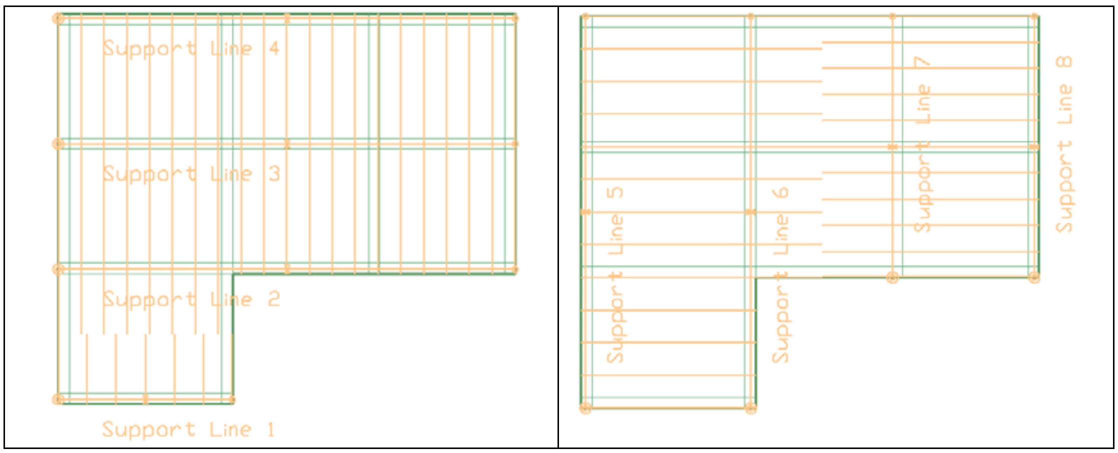
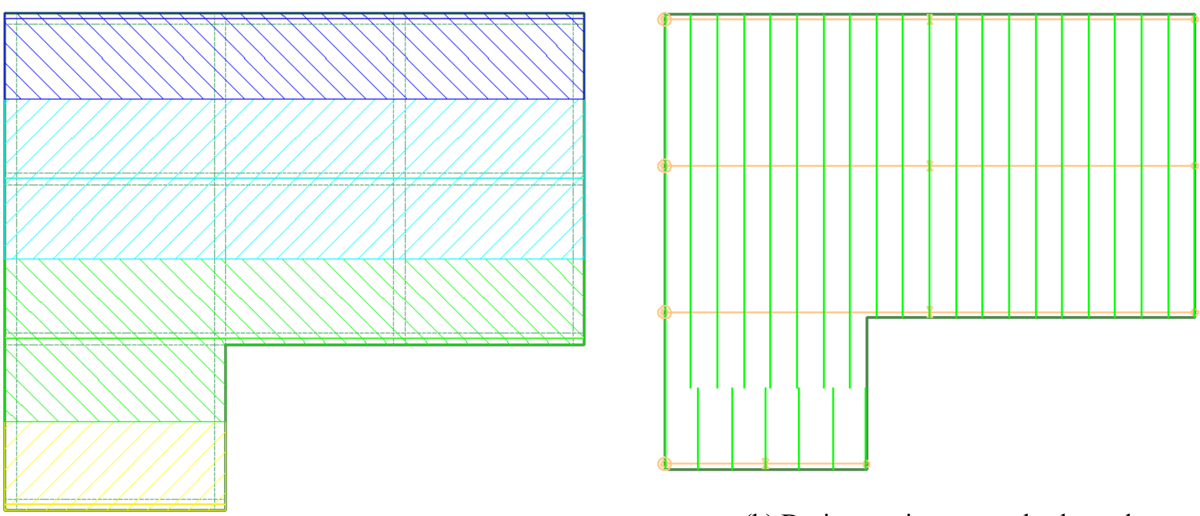


FIGURE 3.4-1 SUPPORT LINES AND DESIGN SECTIONS IN X-AND Y-DIRECTION

4. Click on the *Design the Design Section(s)* button to execute a stress check for each of the design sections shown, as well as to calculate the shears and deformations. **Figure 3.4-2(b)** shows all the design strips of the X-direction in green, indicating that the stresses do not exceed allowable values.



(a) Design strip designation in X-direction (b) Design section stress check results

FIGURE 3. 4-2 DESIGN STRIPS AND STRESS CHECK RESULTS IN THE X-DIRECTION FOR CENTER LIFT CONDITION

5. The actions of each Support Line are shown separately. To see the design actions of a Support Line:

   - Click on one of the design sections of the design strip. A change in color of the Support Line indicates that this design strip was selected. Go to the results summary screen by clicking on the *Show Design Summary* option from the *FEM* pull down menu. The result summary window will open.

   - From the combo box at the top of the screen select *Basic Case*. Then click on the *Stress Diagram* button on the left of it. A distribution, such as shown in **Fig. 3.4-3**, appears. This distribution shows the magnitude of the stress for the Support Line selected.

   - A similar graph for shear can be generated by clicking on the Shear Diagram button.

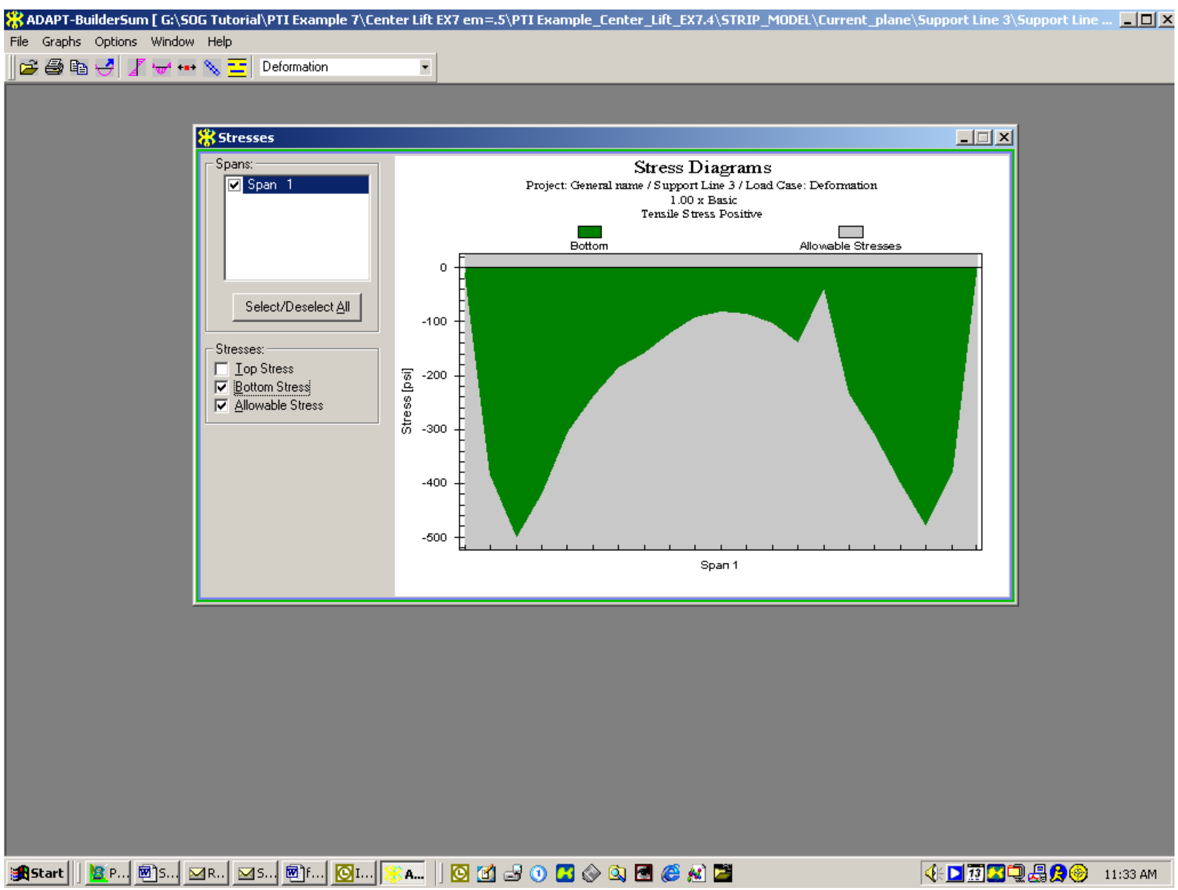
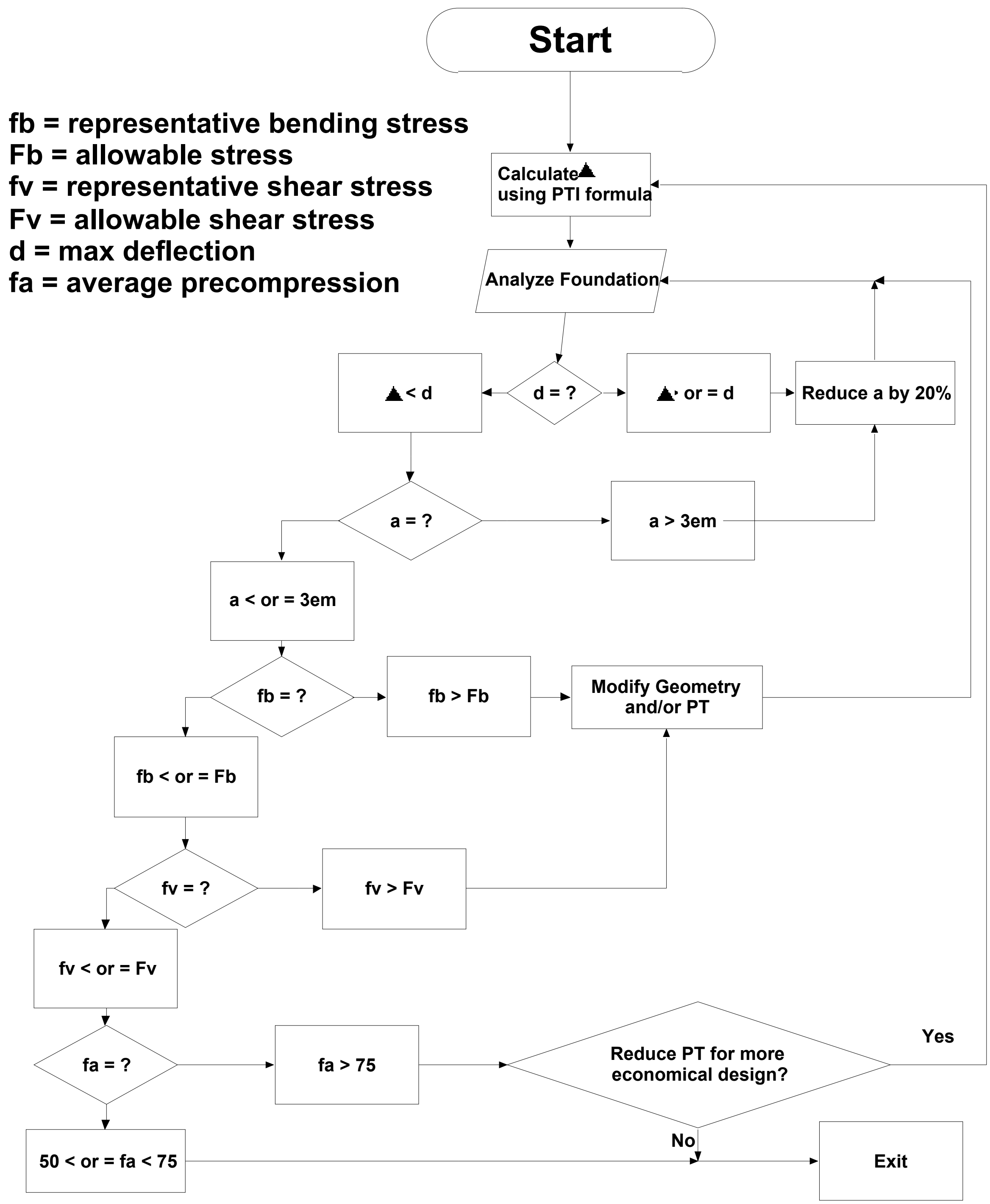


FIGURE 3.4-3 DISTRIBUTION OF DESIGN STRESS IN "RESULTS SUMMARY"

4 Edge Lift Condition

4.1 Flow Chart of Design of SOG for Edge Lift Condition

Flow Chart of Edge Lift Design



4.2 Create Soil Foundation

The soil foundation for edge lift is placed at the edge of Slab. Generate an *Area Spring* as shown in **Section 3.2**, leaving the spring at the Slab edge.

4.3 Apply a Displacement Along Perimeter

The applied displacement is a line displacement placed along the perimeter of the Slab. Before generating the line displacement, you must first calculate the magnitude of the average displacement for the entire Slab. For this example, the long direction and short direction Beams have different lengths and spacing; therefore producing two different applied displacements. Use the average of the displacements calculated.

1. To calculate the applied displacement, click on the *Criteria* menu. The *Criteria* dialog box will appear. Select the *Soil Parameters* tab and check mark the Edge Lift Condition. Input the following parameters:

    - Edge moisture variation em: 5.5 ft
    - Vertical differential movement ym: 0.71 in

2. Select the *Edge Displacements* tab as shown in **Fig. 4.3-1**. Enter the following parameters for each direction and click *Calculate*.

    - Slab edge label: Long direction
    - Construction geometry: Ribbed Slab
    - Slab length normal to edge L: 42 ft
    - Average rib spacing normal to edge S: 12 ft
    - Rib's total depth (average h): 24 in
    - Average weight on Slab edge P: 1.04 k/ft
    - <u>Calculated Displacement</u> 0.35 in

    - Slab edge label: Short direction
    - Construction geometry: Ribbed Slab
    - Slab length normal to edge L: 24 ft
    - Average rib spacing normal to edge S: 14 ft
    - Rib's total depth (average h): 24 in
    - Average weight on Slab edge P: 1.04 k/ft
    - <u>Calculated Displacement</u> 0.33 in

3. Use 0.34 in as the average applied displacement.

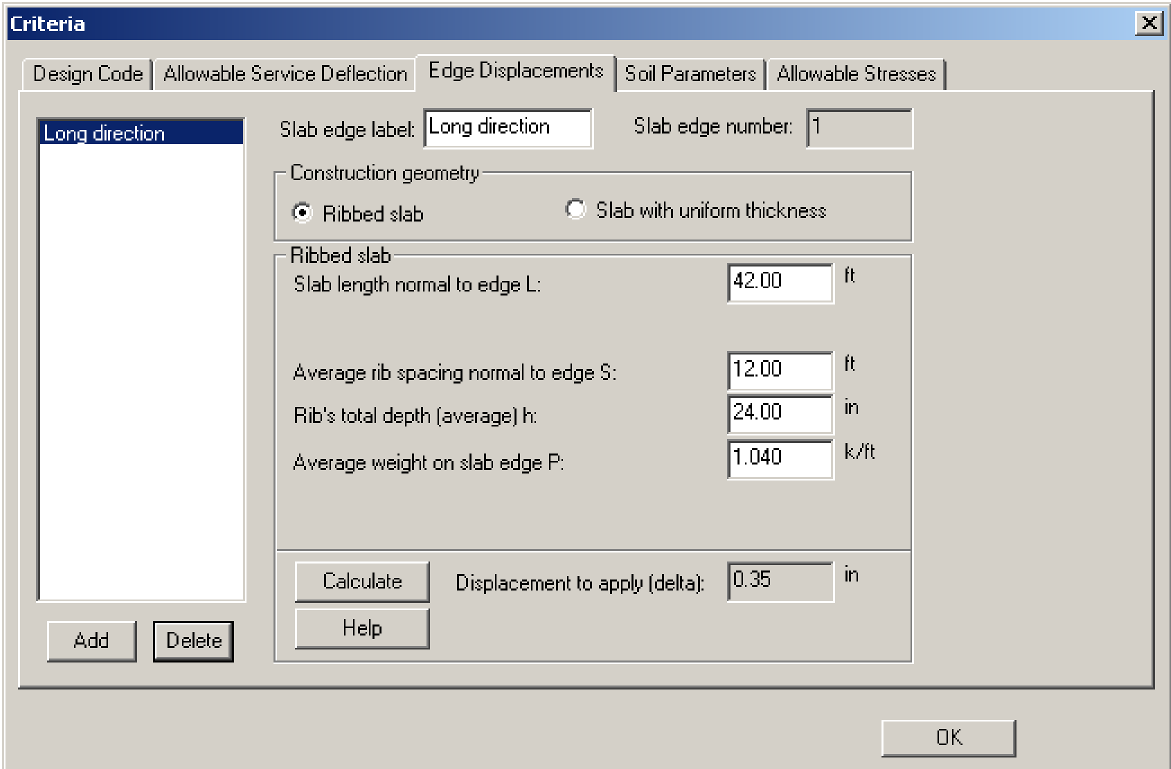


FIGURE 4.3-1 CRITERIA DIALOG BOX

4. In the *Loading* menu, select *Line Displacement* for the *Apply Displacement* drop down menu. Next click on the Item's Properties button. Change the following parameters (**Fig. 4.3-2**), then click on the button to accept the changes:

Note: By changing the parameters before the Line Displacement is created, these modifications become the default values. Therefore, all Line Displacements created hereafter will have this specified Z translation.

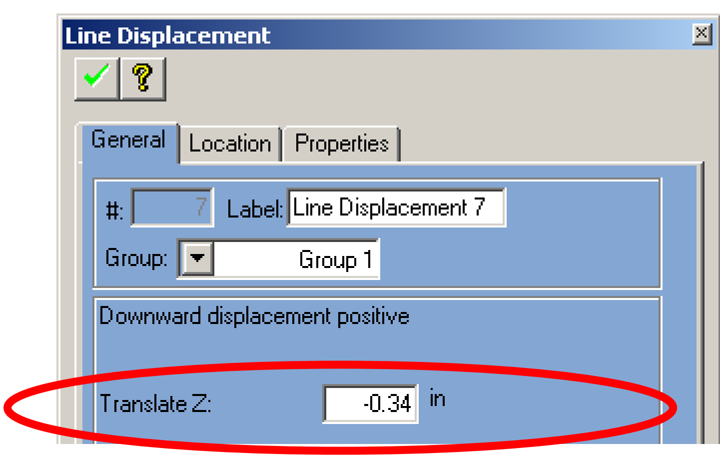


FIGURE 4.3-2 CRITERIA DIALOG BOX

5. Using the *Snap to Intersection*  tool, snap the line displacements along the perimeter of the Slab as shown in **Fig. 4.3-3.**

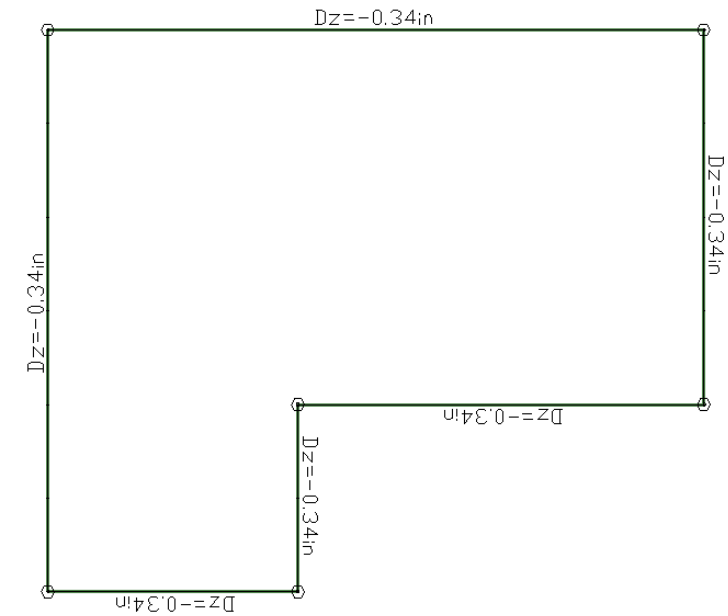


FIGURE 4.3-3 APPLIED DISPLACEMENT

4.4 Analyze and Verify Results

From the *FEM* pull down menu, select *Analyze Structure*  icon. This will perform the finite element analysis of the structure and report its completion on the computer screen.

4.4.1 View the Analysis Results

From the *FEM* toolbar, click on the *View Analysis Results*  icon. This will bring up the viewer screen, as shown in **Fig. 4.4.1-1**. Next, we will view the deflection contour of the Slab.

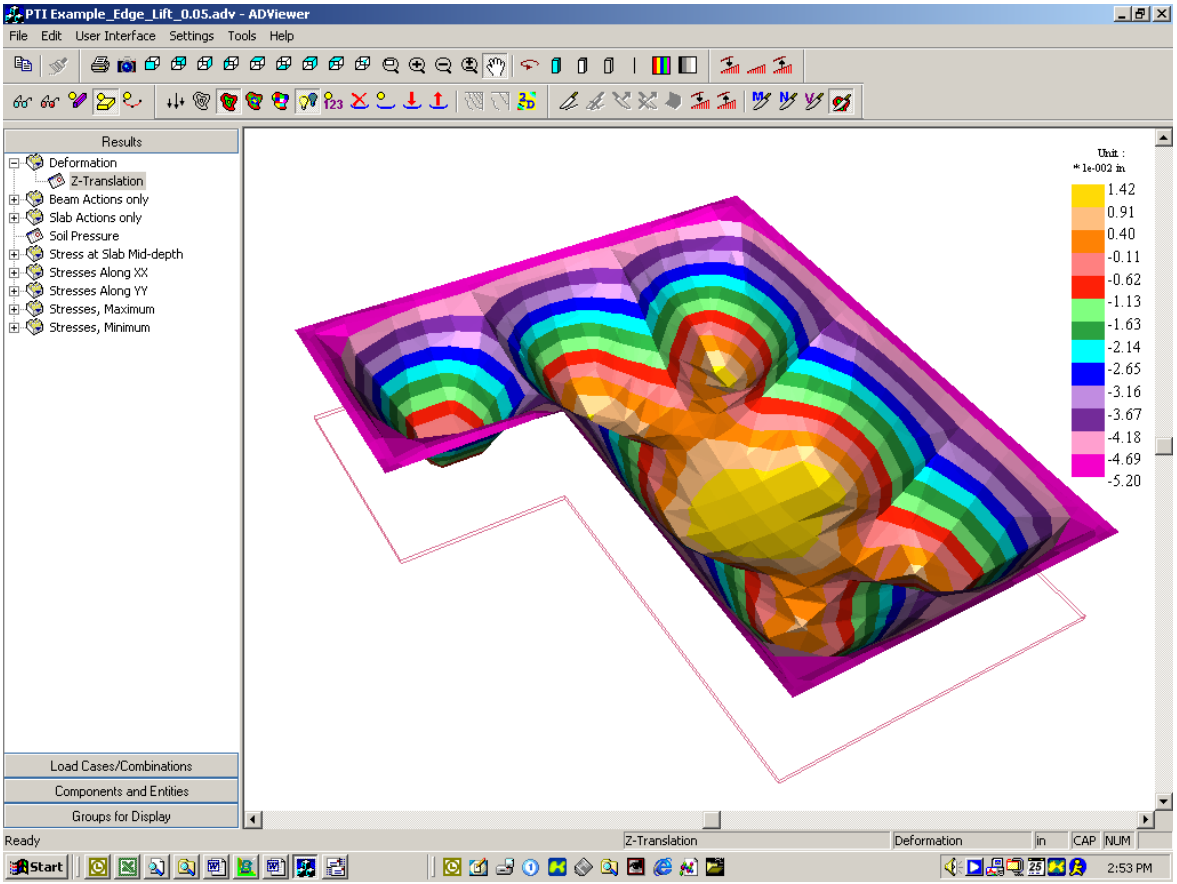


FIGURE 4.4.1-1 RESULTS VIEWER

- Click on the "Load Cases/Combinations" tab on the bottom left of the screen
- From the menu that opens, select "Basic_Combination."
- Click on the "Results" tab in the top left region of the screen.

4.5 Check Design for Stress, Shear and Deflection

Repeat the design step outlined in **Section 3.4**. Your final design is based on the worst-case scenario of the two soil conditions.