FB MultiPier v.4

Pile Group Example

Professor's Driven Pile Institute

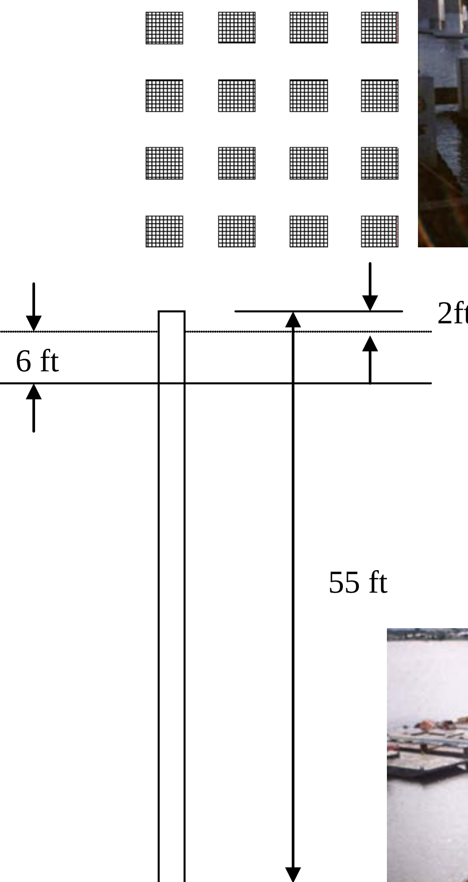
Utah State University – June 24, 2015

J. Brian Anderson (jbanders@auburn.edu)

Department of Civil Engineering Auburn University

Deep Foundations – FB-MultiPier

Example 2: 4x4 Pile Group - Roosevelt Bridge Example

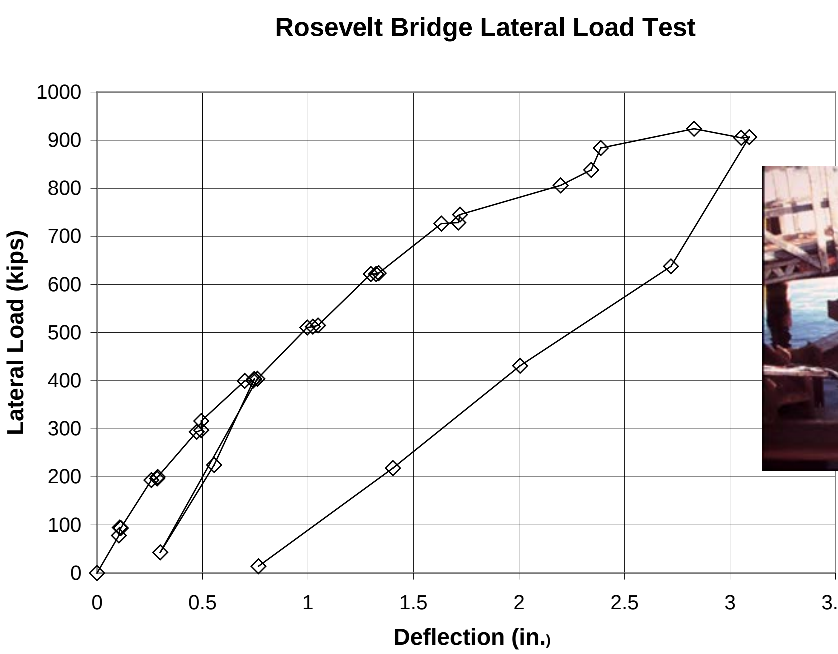


Soil:
$\phi = 32^o$
k = 120 pci
γ = 120 pcf
G = 7.5 ksi
ν = 0.33
τ_f = 570 psf
Q_{tip} = 384 kips

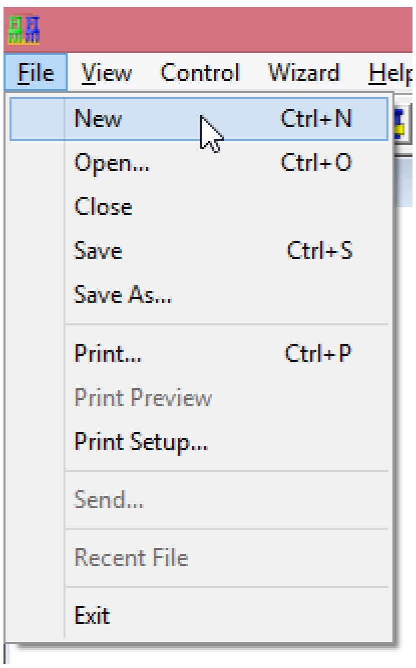
Pile Group
FDOT 30”
Prestressed Square
4x4
3D C/C spacing
Piles Pinned
Cap 24” Thick Steel
E_s = 29000ksi

Predict Deflections for Lateral Loads:

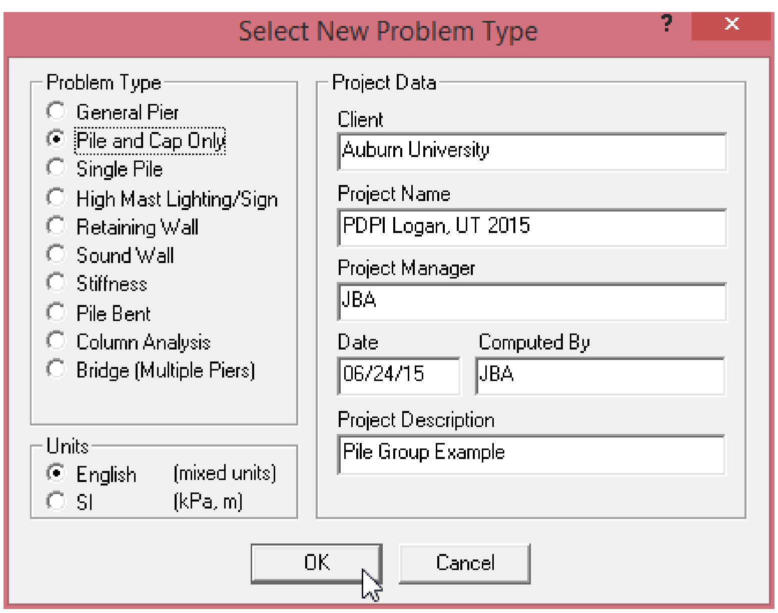
Lateral Load (kips)
0
125
250
375
500
625
750
875
1000



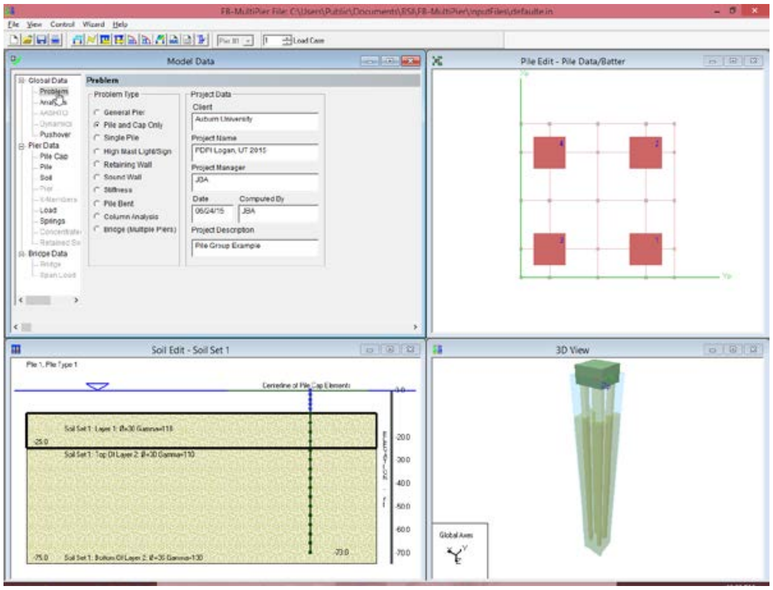
Start “FB-MultiPier” by selecting the shortcut on your desktop, or find “BSI > FB –MultiPier> FB – MultiPier” under the Start Menu.



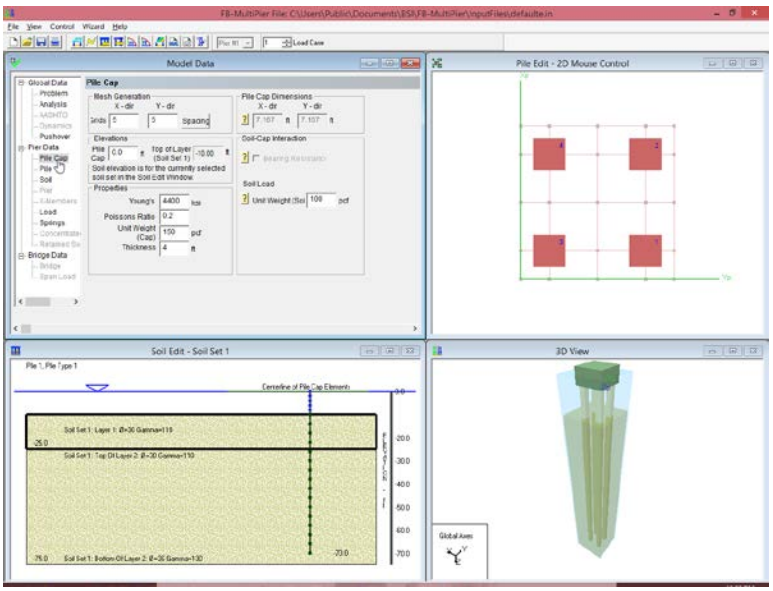
Select File>New from the menu at the top to create a new file.



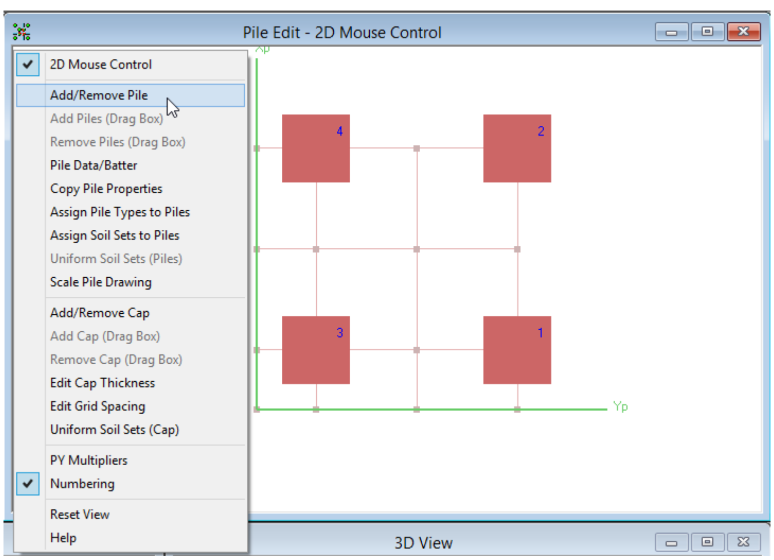
Fill in the appropriate project information. Select, for this case, choose Problem Type “Pile and Cap Only” Finally set Units to “English (mixed units)”. Click OK to continue.



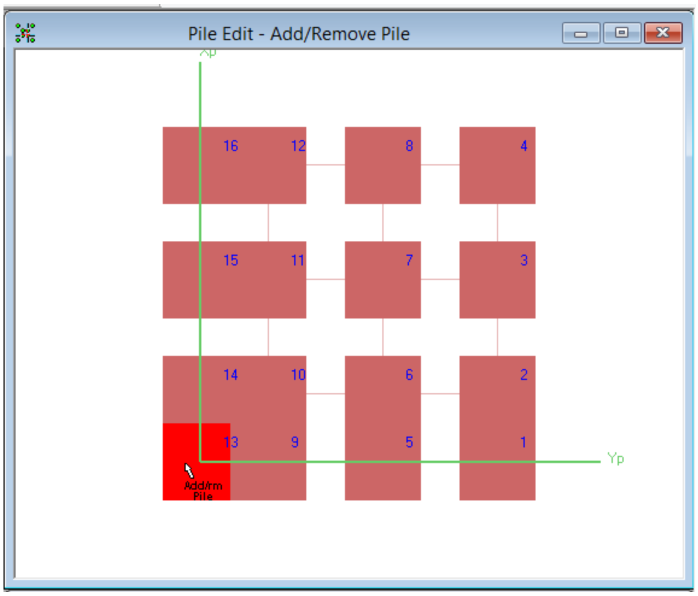
he editor windows will now appear. The top left window is the Model Data window where most info is entered. The top right window is the Pile Edit window shows the pile group in plan. The bottom left window is the Soil Edit window where the soil stratigraphy is shown. A 3-D view of the pile group is shown in the bottom right pane.
A default problem is automatically loaded which you will need to change



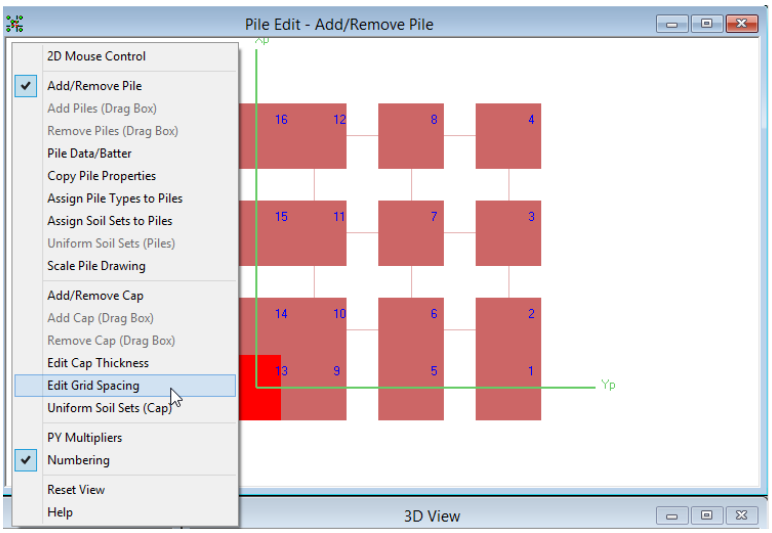
Select "Pile Cap" from the Menu. Change the number of grid points to 4 x 4.



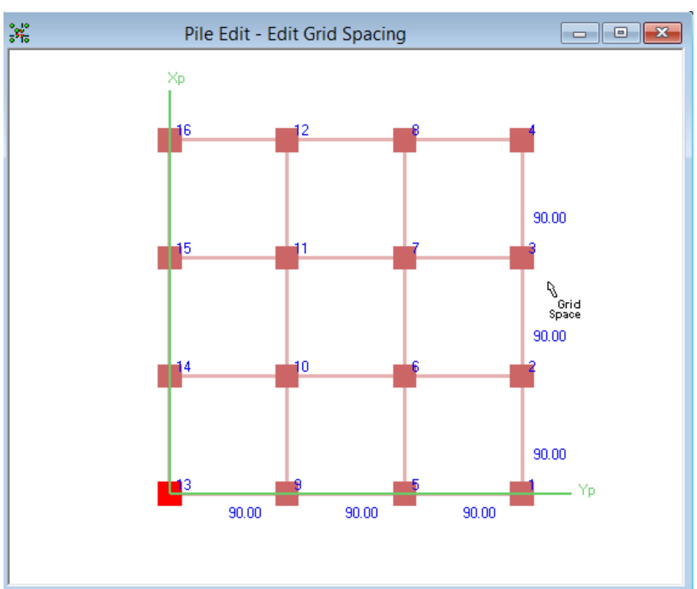
The resulting pile cap will need adjustment for spacing and placement of piles. Right click in the "Pile Edit" window and select "Add/Remove Pile," click on the cap node points to add piles to all the nodes.



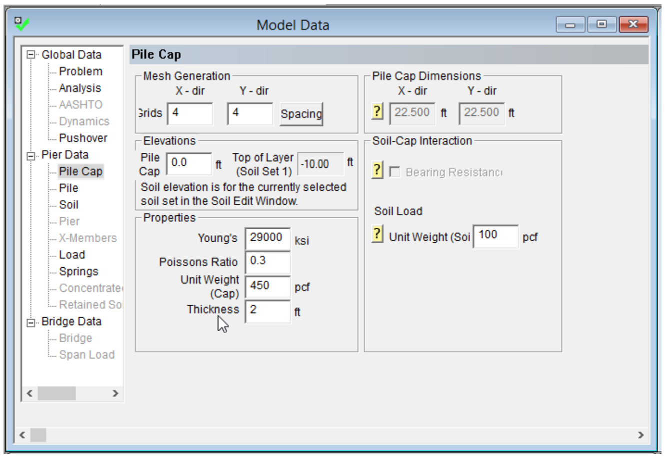
The spacing will now need to be adjusted to 3d or 3 X 30" center-to-center between the piles in both directions.



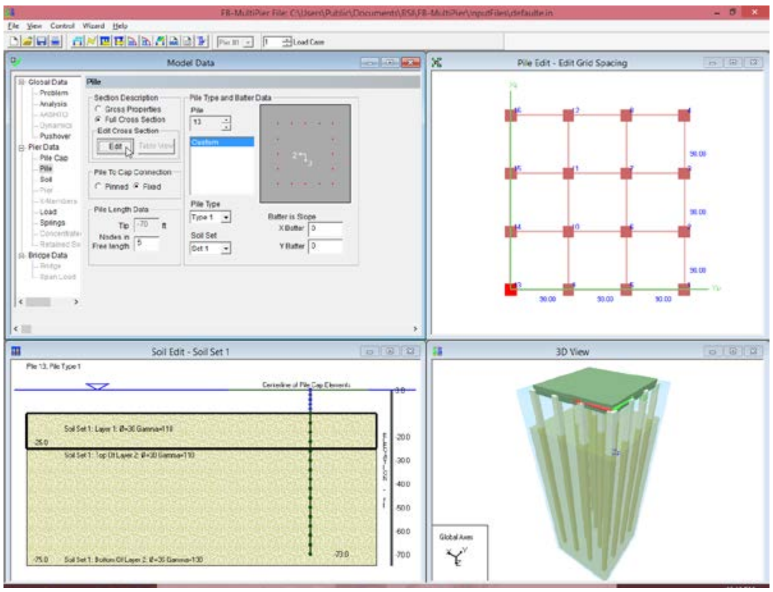
Right click in the "Pile Edit window and select "Edit Grid Spacing." Click on the grid spacing numbers to change them. There is also a table entry option.



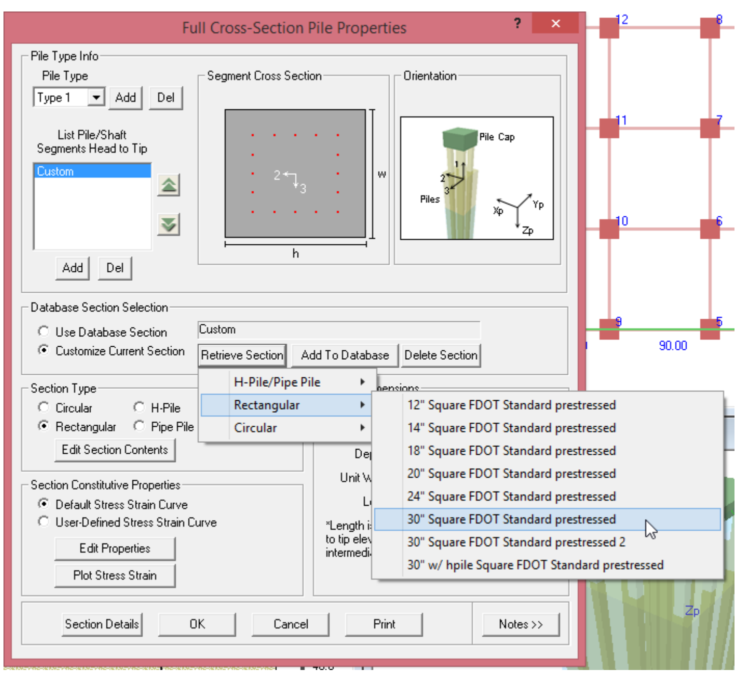
The resulting pile group should look symmetrical and the correct spacing (90 inches center-to-center) can be seen in the blue numbering.



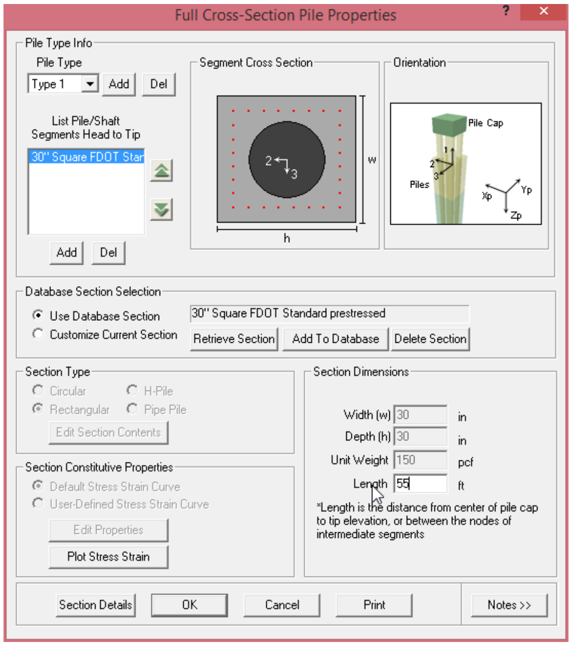
The cap is steel. Enter 29000ksi for the Young's Modulus, 0.3 for Poisson's Ratio, 2 ft for thickness and unit weight of 450pcf.



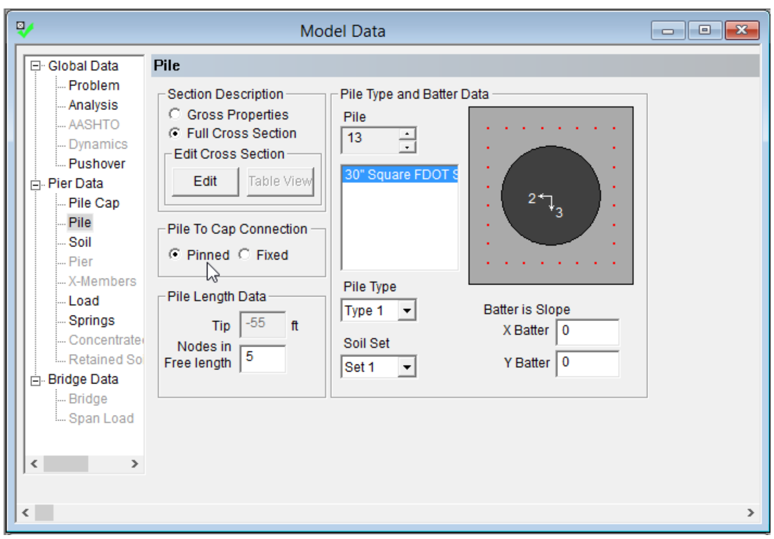
Select "Pile" from the Menu. Click "Edit" under "Edit Cross Section."



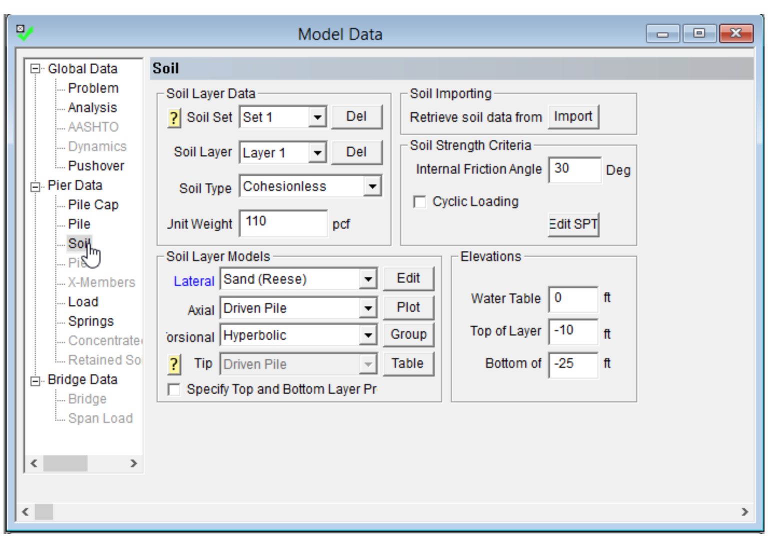
The pile section used in this analysis is the FDOT 30" Standard. Under "Database Section Selection," click on "Retrieve Section" and select "30" square FDOT Standard Prestressed."



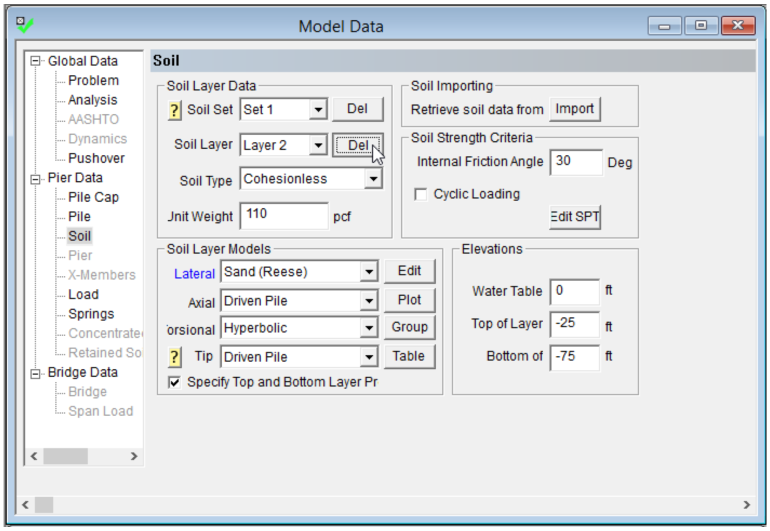
Change the pile length to 55ft.
Click OK to continue.



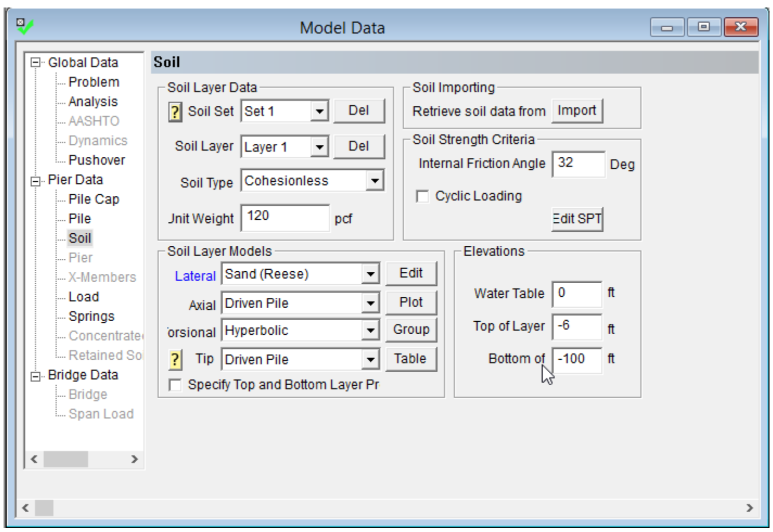
Under "Pile to Cap Connection," select Pinned (not typical but is the case for modeling the load test). This concludes the structural input.



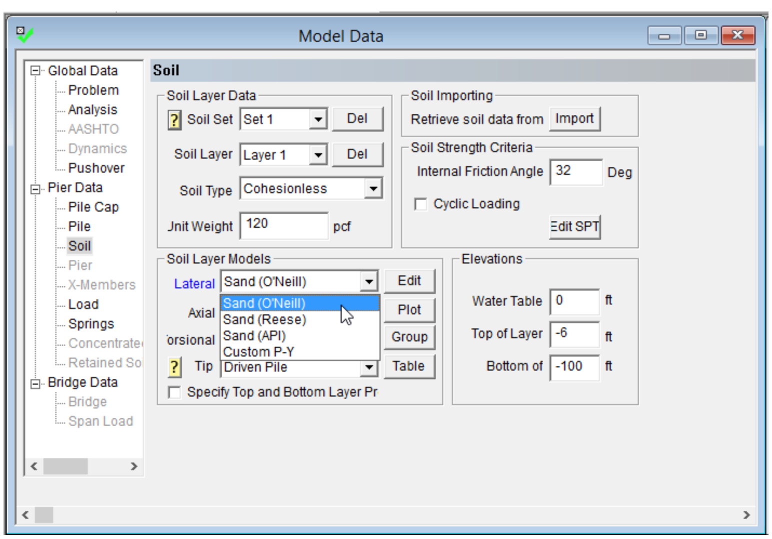
Select "Soil" from the menu in the Model Data pane. Note that the "Soil Layer" drop down box contains the layers shown in the "Soil Edit" pane below.



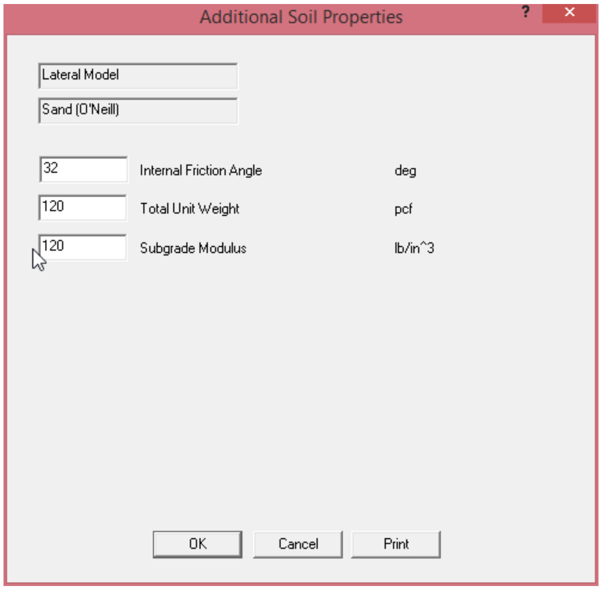
Since we will be using a single soil layer for this example, select layer 2 then hit the delete key to remove the layer.



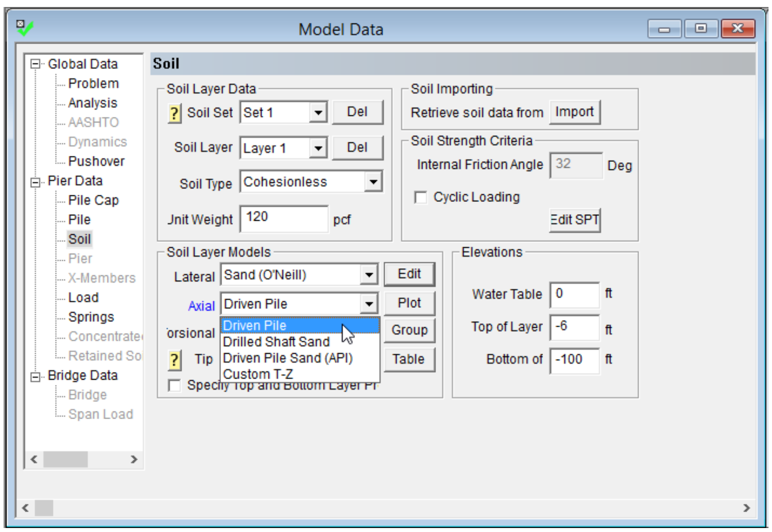
Set the Soil Type, for Layer 1, to “cohesionless.” Enter Total Unit Weight equal to 120 pcf. Enter a friction angle of 32^{o}. Set the top of the layer at -6ft and the bottom at –100ft (for proper embedment). The water surface is at 0ft.



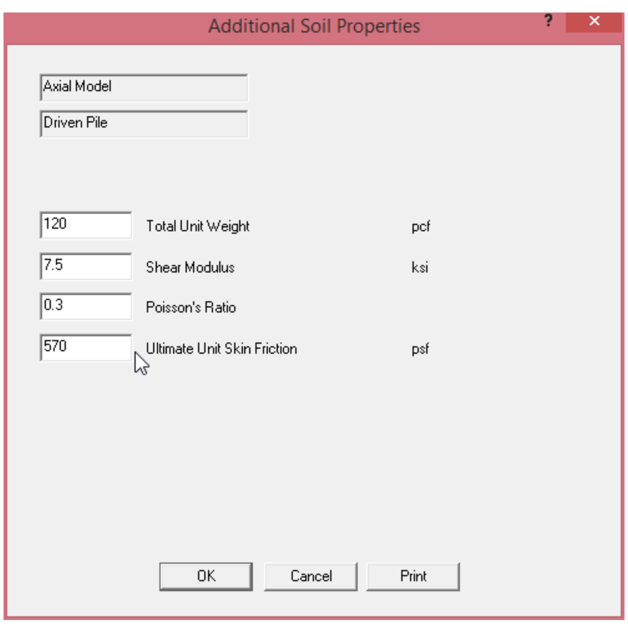
Under Soil Layer Models, use the drop down box by “Lateral” to choose the lateral soil type, in this case, Sand (O’Neill).



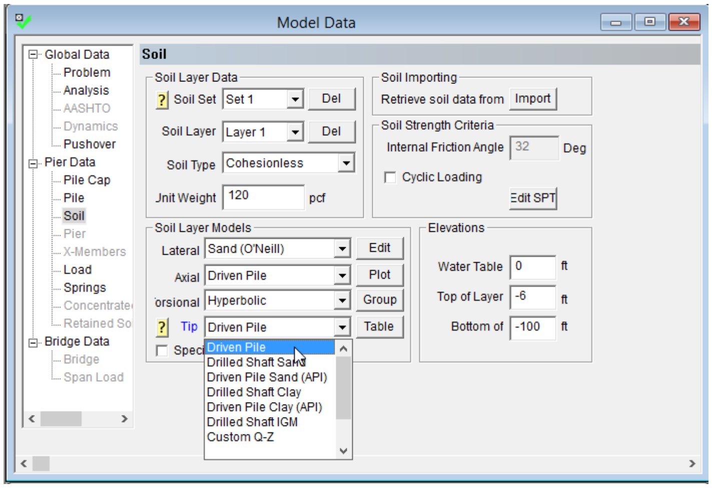
Click “Edit.” The unit weight and friction angle will be copied from before. Enter the value for subgrade modulus, 120 pci. Click “OK” to continue.



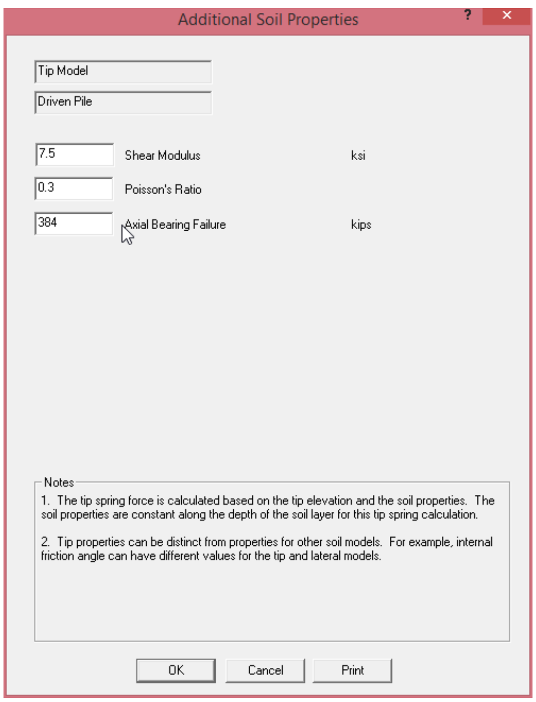
Under Soil Layer Models, use the drop down box by “Axial” to choose the axial soil type, in this case, Driven Pile.



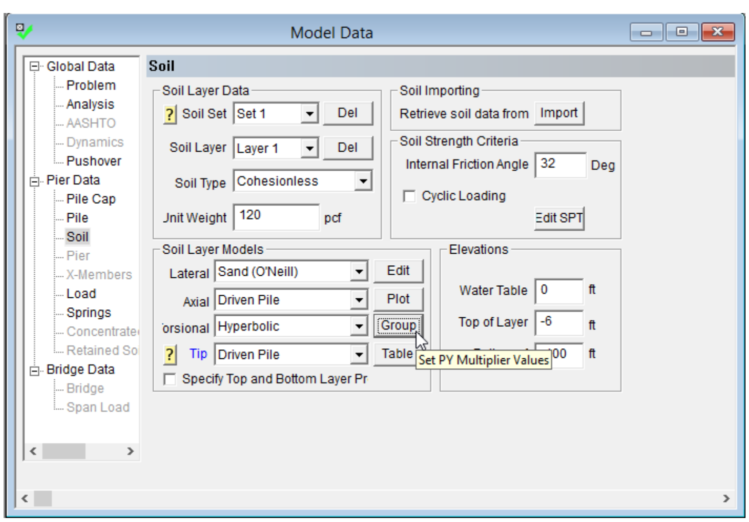
Click “Edit.” The unit weight and friction angle will be copied from before. Enter the value for shear modulus, 7.5ksi, Poisson’s Ratio, 0.3, Vertical Failure Shear, 570psf. Click “OK” to continue.



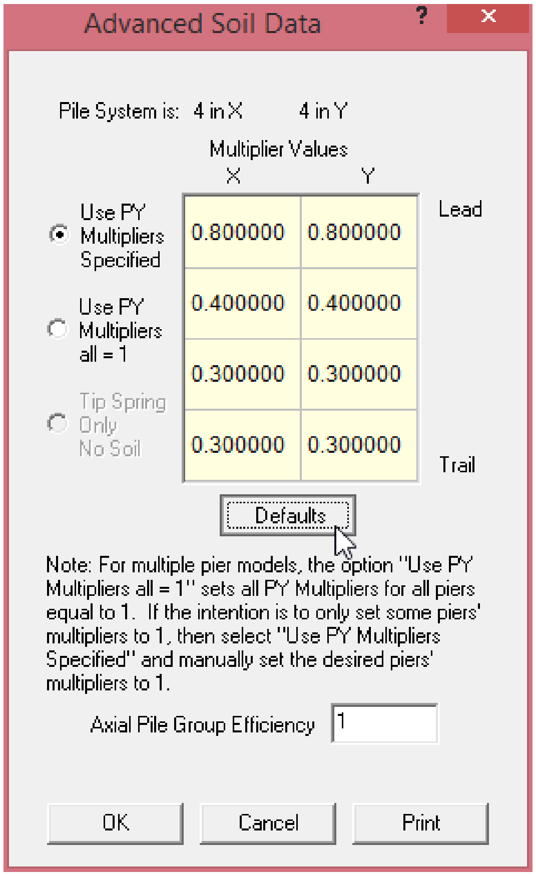
Under Soil Layer Models, use the drop down box by "Tip" to choose the lateral soil type, in this case, Driven Pile.



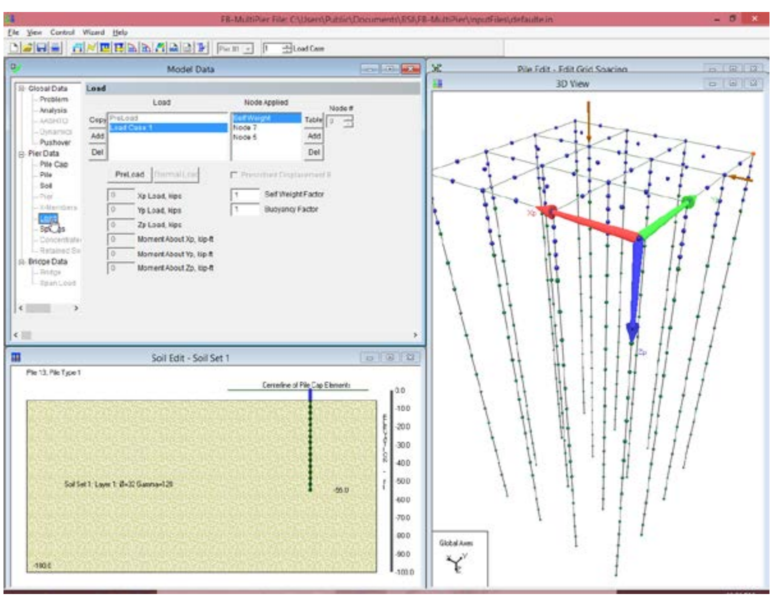
Click "Edit." Enter the value for shear modulus, 7.5ksi, Poisson's Ratio, 0.3, Axial Bearing Failure, 384kips. Click "OK" to continue.



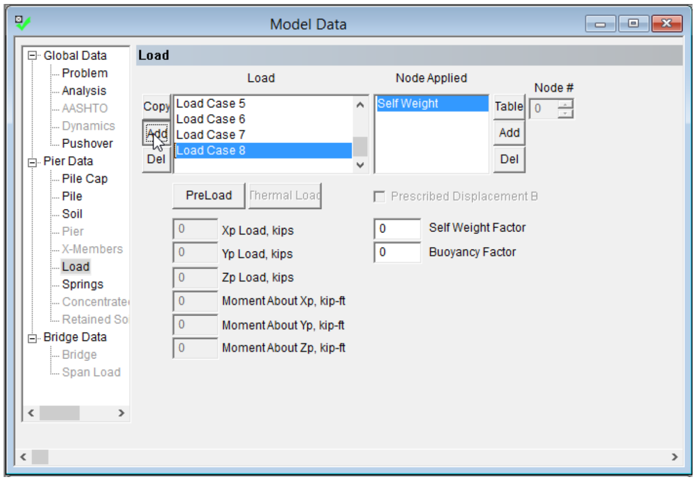
The far field or pile group interaction is modeled using p-y multipliers. Click on the Group button to edit the multipliers.



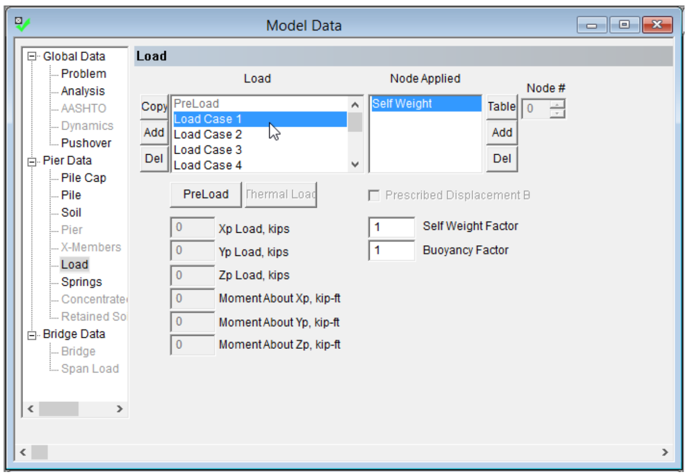
We would like to set the multipliers to the default value of 0.8, 0.4, 0.3, 0.2….0.2., 0.3. Click the Defaults button to set the multipliers



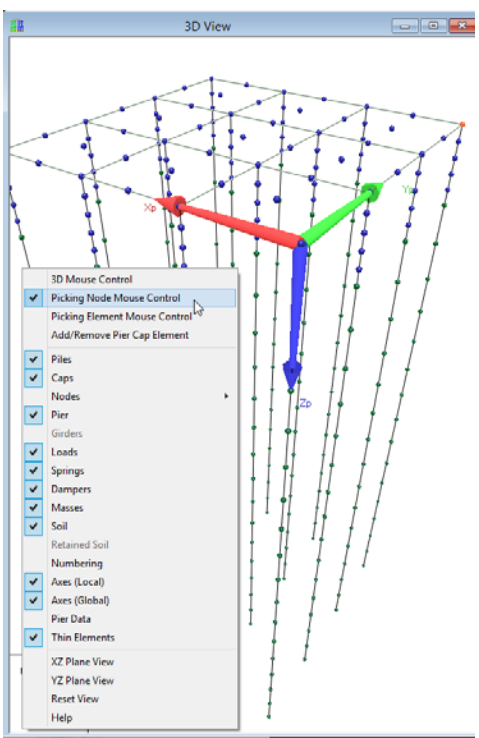
Next, we need to specify the loads on the pile. Now, choose “Load” tab in the model data window.



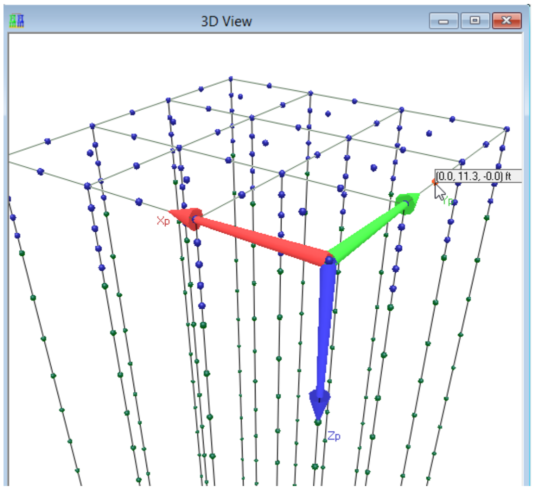
One load cases is specified by default. In this problem there are 8 load cases. Click the Add button to add additional load cases.



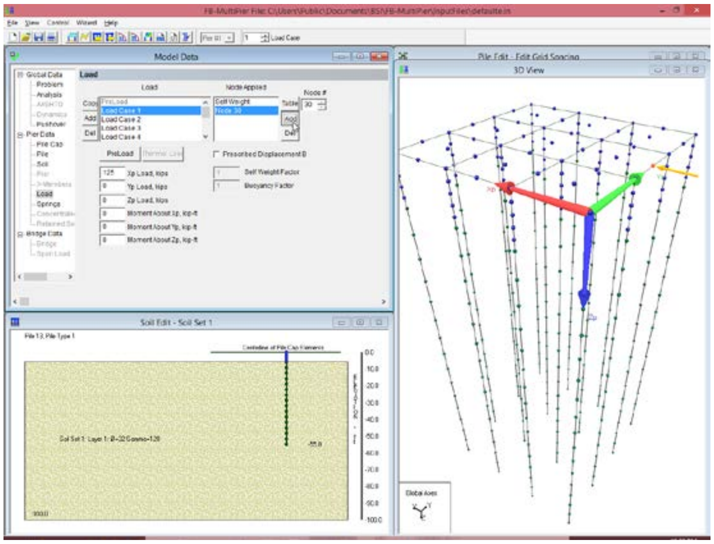
Select Load Case 1. In this problem, the lateral load was applied on the edge of the pile cap. Remove all default loads. Load cases 2 through 8 should have no loads by default.



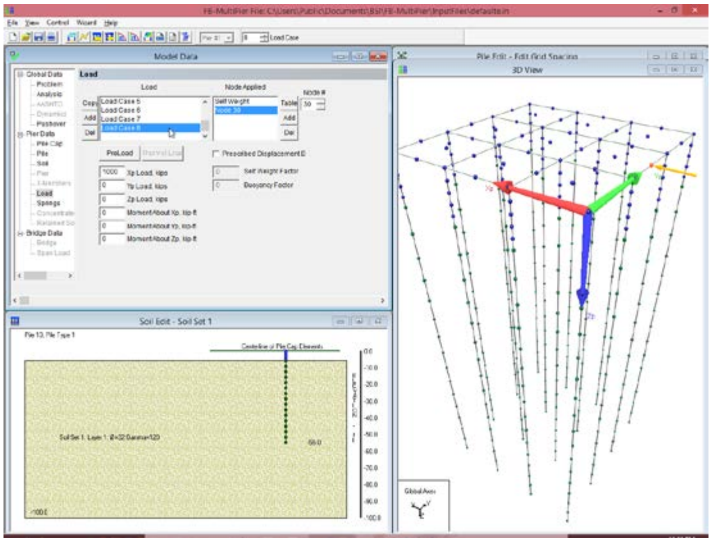
Right click near the pile group in the 3d View window. Select “Picking Node Mouse Control”



Select Load Case 1. Click on the center node on the side of the pile cap.

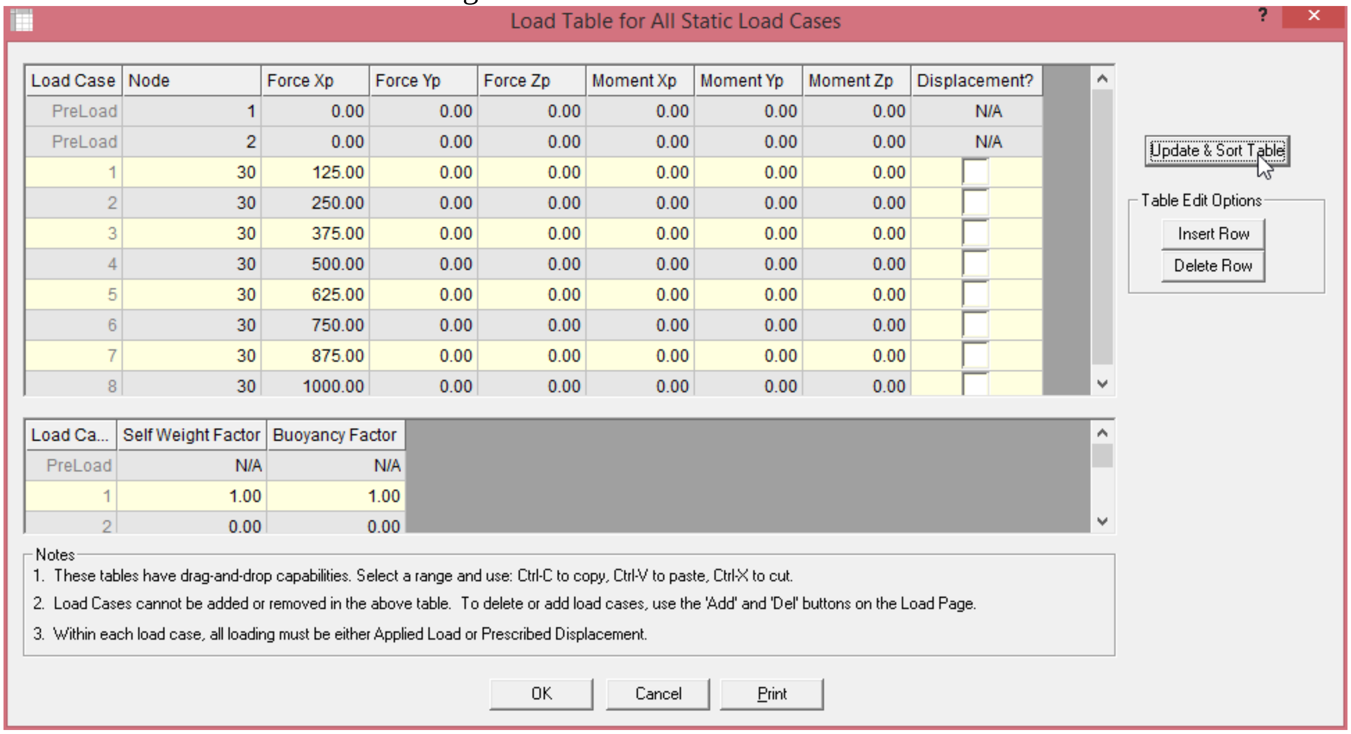


Now under Node Applied, a new node number should appear. Click “Add” to add a load there.



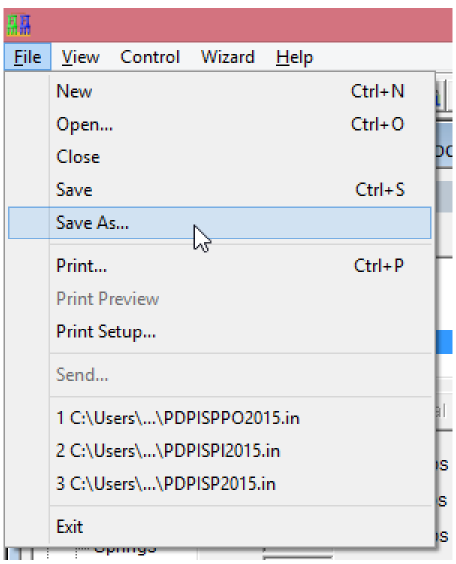
Enter 125 in the x direction. You should see a horizontal vector pointing in the –x direction. Repeat these steps for each of the remaining 7 loads 250, 375, 500, 625, 750, 875, 1000.

You can edit some information using the load table

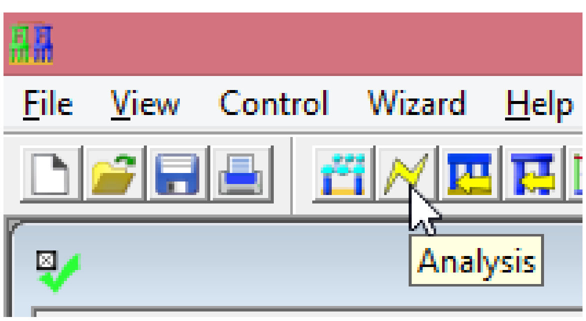


Load Case	Node	Force Xp	Force Yp	Force Zp	Moment Xp	Moment Yp	Moment Zp	Displacement?
PreLoad	1	0.00	0.00	0.00	0.00	0.00	0.00	N/A
PreLoad	2	0.00	0.00	0.00	0.00	0.00	0.00	N/A
1	30	125.00	0.00	0.00	0.00	0.00	0.00	
2	30	250.00	0.00	0.00	0.00	0.00	0.00	
3	30	375.00	0.00	0.00	0.00	0.00	0.00	
4	30	500.00	0.00	0.00	0.00	0.00	0.00	
5	30	625.00	0.00	0.00	0.00	0.00	0.00	
6	30	750.00	0.00	0.00	0.00	0.00	0.00	
7	30	875.00	0.00	0.00	0.00	0.00	0.00	
8	30	1000.00	0.00	0.00	0.00	0.00	0.00	

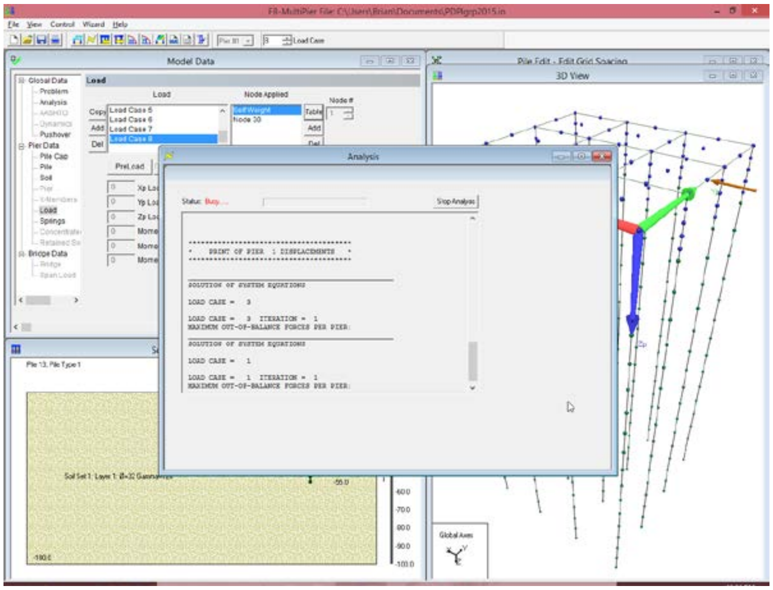
Load Ca...	Self Weight Factor	Buoyancy Factor
PreLoad	N/A	N/A
1	1.00	1.00
2	0.00	0.00



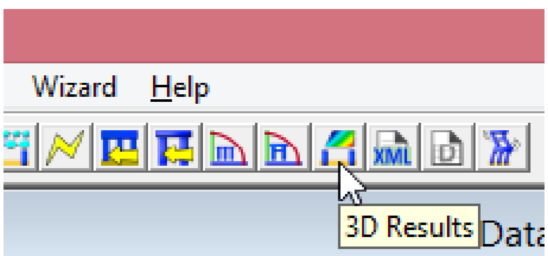
Use the “Save As…” command to save your problem before the analysis.



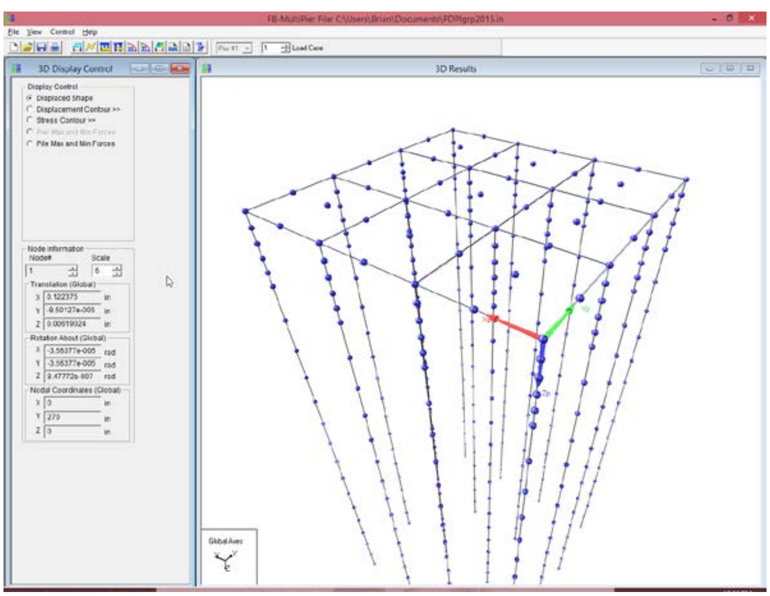
The next step will be to perform the analysis. Click on the lightning bolt icon to run the analysis. You will get a warning that you will force a file save, accept and continue



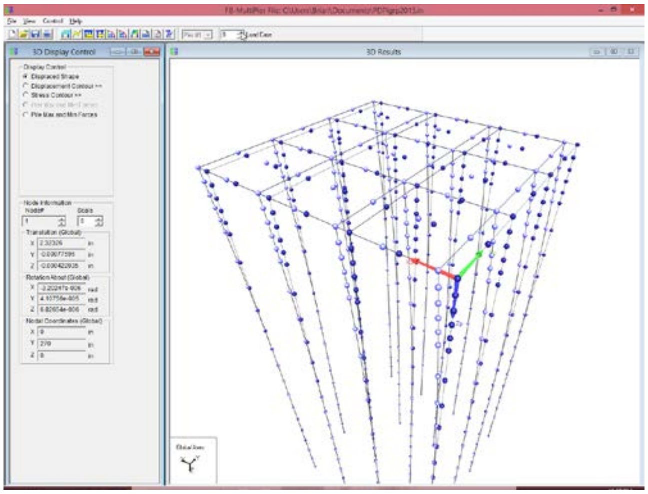
The iterative process of FB-MultiPier will be shown while the analysis progresses.



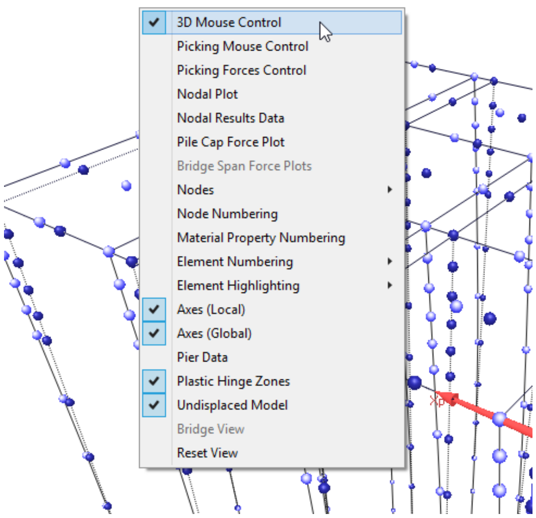
Once the analysis is complete, click on the 3D Results icon to see a graphical depiction if the analysis.



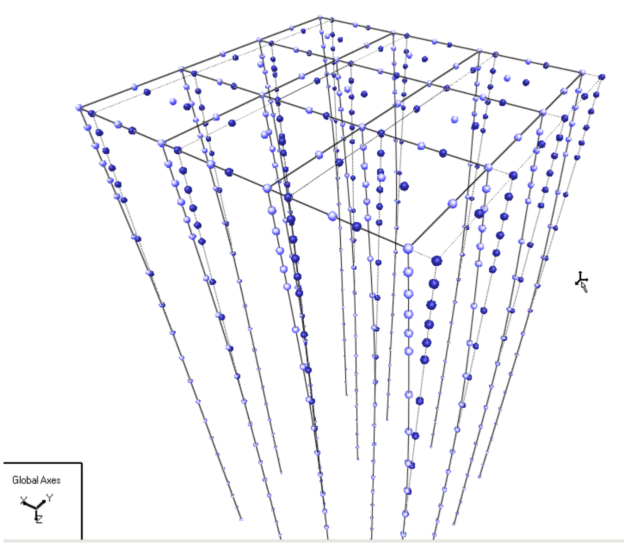
In the tool bar area, you can select the load case to view. In the 3D Display Control pane, you can select which node to display displacements or rotations, as well as the nodal coordinates. A 3 Dimensional representation of the results is shown in the 3D View pane.



The deflected shape for load case 8 is shown here.

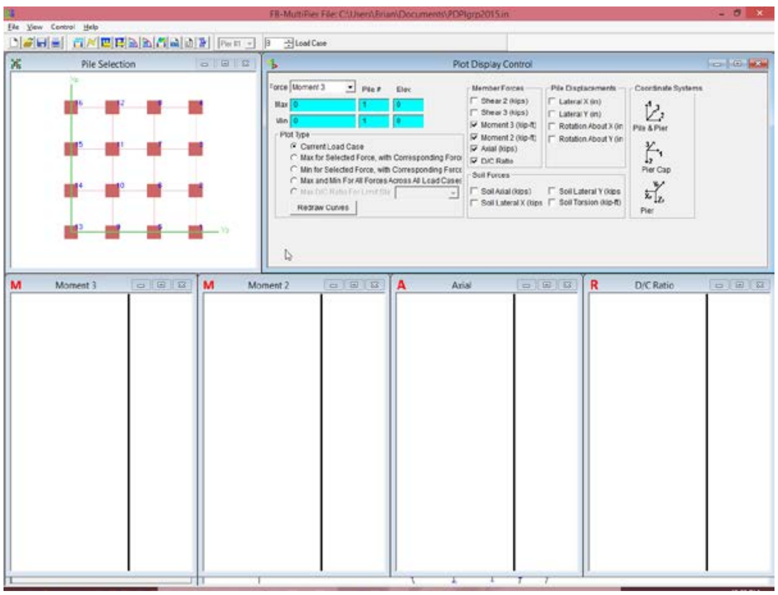


It may be useful to rotate, resize, or move the model. Right click in the 3D Results window and choose 3D Mouse Control.

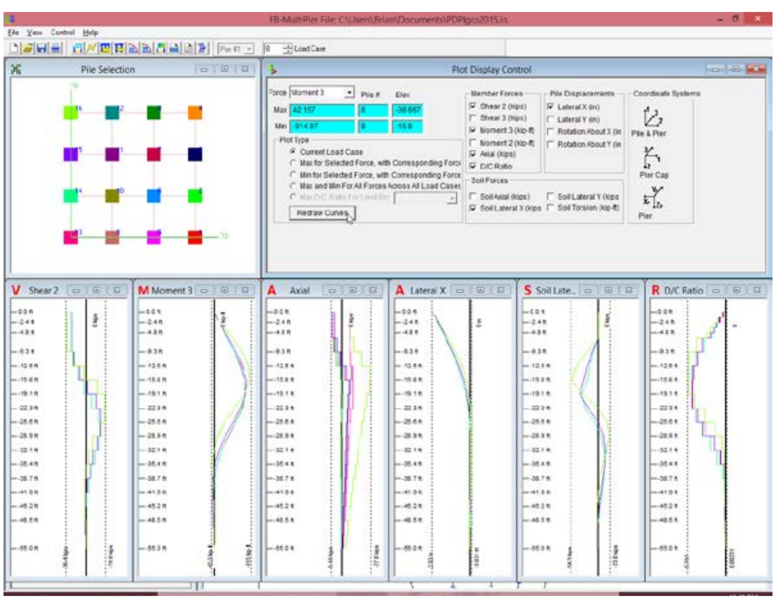


By clicking the left mouse and dragging, the model can be rotated in all directions. Holding shift and left click dragging will translate the model. Control and left click dragging will allow you to resize the model. Note this also works in the 3D View window of the input portion of the program.

Next, click on the Pile Results button in the toolbar. This will open a series of pile results windows.



The pile results window is shown. In the Plot Display Control window, select the quantities you would like to view plotted versus pile depth. In the Pile Selection window, select the piles you would like to plot.



The selected quantities will be color coded with the selected pile shown in each of the individual plot windows.