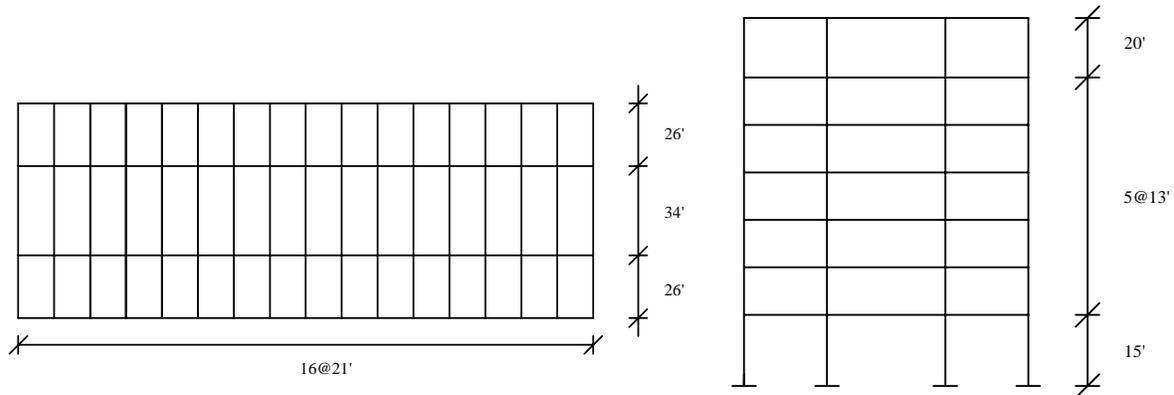


## SAP 2000 Tutorial Session Notes

This tutorial provides the basic steps of performing a frame analysis using SAP 2000. It is based on the design project example shown below, the complete solution of which is provided as a handout.



### Results of Preliminary Analysis:

Slab thickness: 9 in  
 Beam dimensions: 14"x31"  
 Exterior columns: 14"x18"  
 Interior columns: 16"x22"

### Loads:

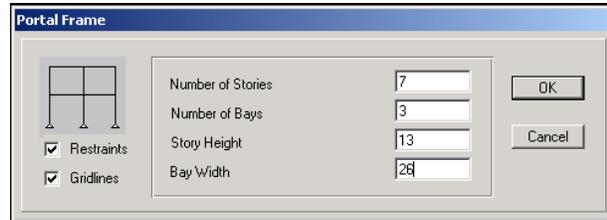
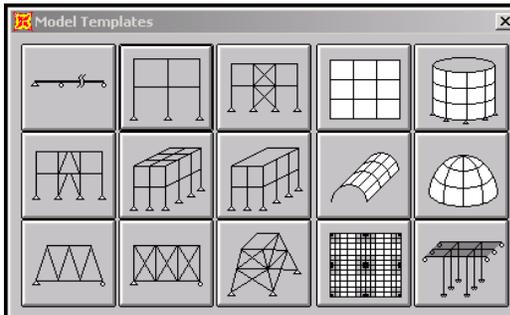
Exterior beams: LL=0.72 kips/ft  
 DL=1.85 kips/ft  
 Interior beams: LL=0.83 kips/ft  
 DL=2.10 kips/ft

Wind and Earthquake loads are specified in the appendix. Only earthquake loads are considered.

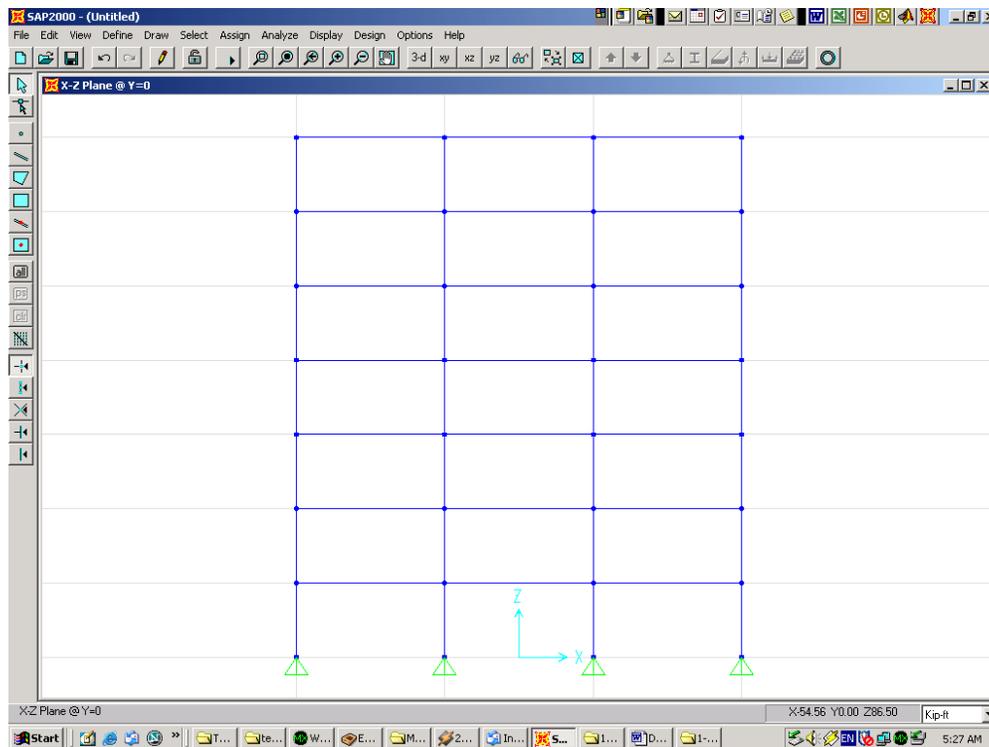
The frame is solved in the short direction only since this direction is more critical.

## Analysis Steps

1. Generate the frame geometry from analysis templates.  
Menu: File > New model from template



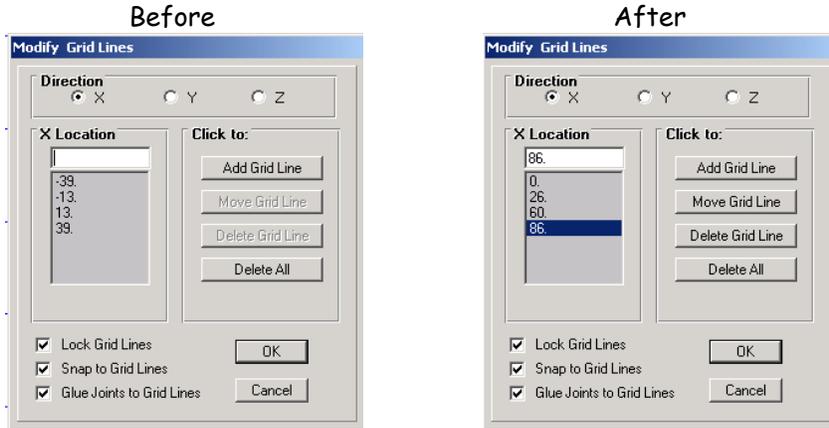
## Result



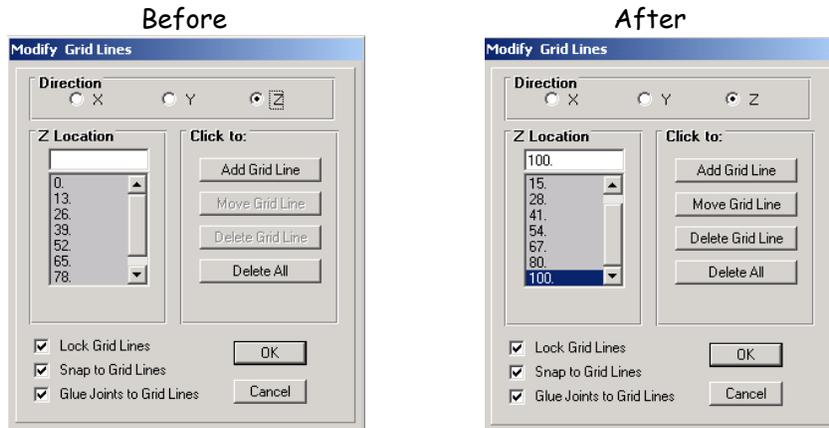
Note: make sure that proper units are selected (shown in the lower right corner)

Since we chose the common floor heights and bay spans, we need to **move** the grids  
 Menu: Draw > Edit Grids

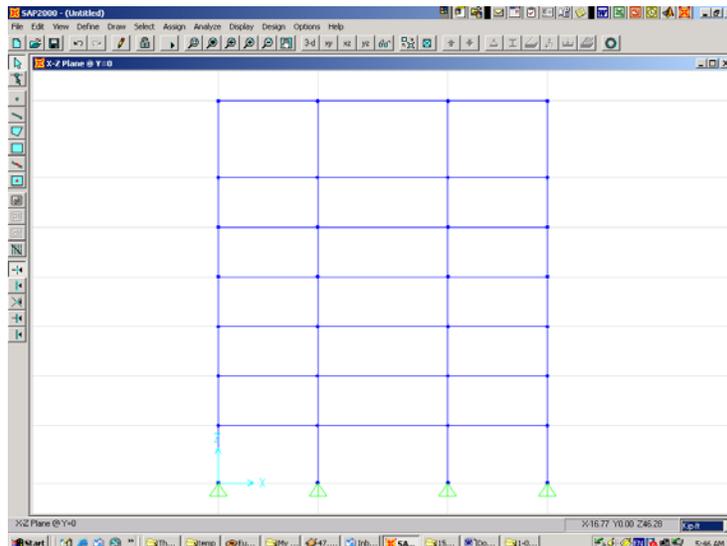
X-grids before and after



Z-grids before and after

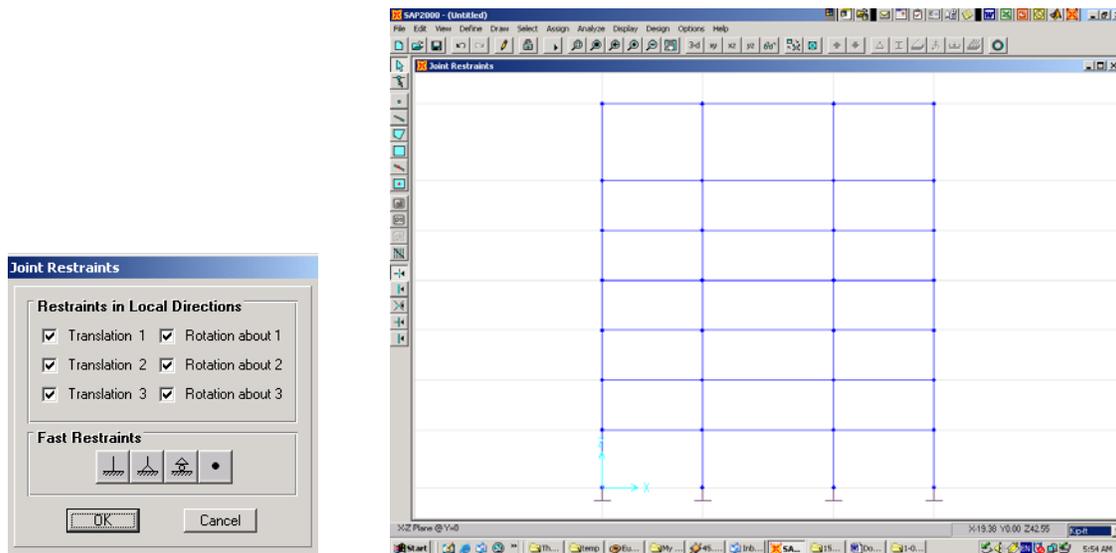


Result:



## 2. Set Boundary Conditions

The default boundary condition in the template is simple supports at the bottom. We need to change those to fixed supports. After selecting these joints, one can either use the menu: Assign > Joint > Restraints, or the quick menu button at the top  (Restraints is the first one).



## 3. Define Material Properties

$$f'_c = 4 \text{ ksi} = 576 \text{ ksf}$$

$$E_c = 57000\sqrt{4000} = 3605 \text{ ksi} = 519120 \text{ ksf}$$

$$f_y = 60 \text{ ksi} = 8640 \text{ ksf}$$

$$E_y = 29000 \text{ ksi} = 4176000 \text{ ksf}$$

Menu: Define > Materials > Conc

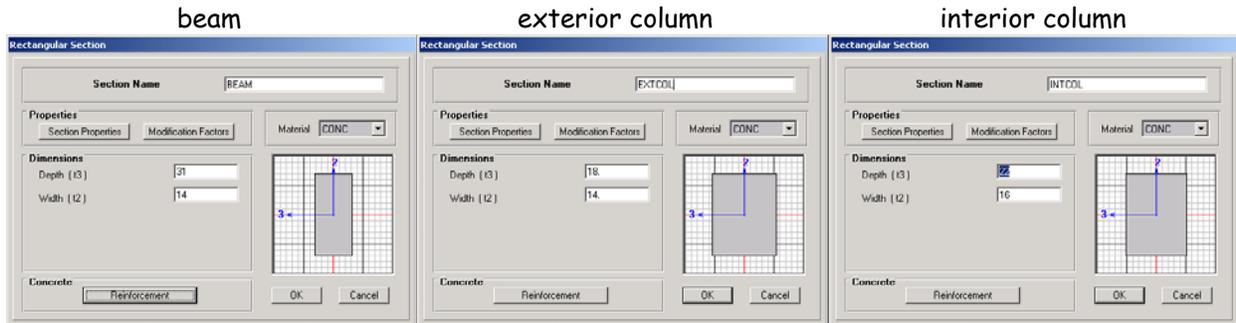
Material Property Data															
Material Name: <input type="text" value="CONC"/>															
Type of Material: <input checked="" type="radio"/> Isotropic <input type="radio"/> Orthotropic <input type="radio"/> Anisotropic	Type of Design: <input type="text" value="Concrete"/>														
<table border="0"> <tr> <td><b>Analysis Property Data</b></td> <td><b>Design Property Data</b></td> </tr> <tr> <td>Mass per unit Volume: <input type="text" value="0"/></td> <td>Reinforcing yield stress, <math>f_y</math>: <input type="text" value="8640"/></td> </tr> <tr> <td>Weight per unit Volume: <input type="text" value="0.1501"/></td> <td>Concrete strength (Cylinder), <math>f_c</math>: <input type="text" value="576"/></td> </tr> <tr> <td>Modulus of Elasticity: <input type="text" value="519120"/></td> <td>Shear steel yield stress, <math>f_{ys}</math>: <input type="text" value="8640"/></td> </tr> <tr> <td>Poisson's Ratio: <input type="text" value="0.2"/></td> <td>Concrete shear strength, <math>f_{cs}</math>: <input type="text" value="300"/></td> </tr> <tr> <td>Coeff of Thermal Expansion: <input type="text" value="5.500E-06"/></td> <td></td> </tr> <tr> <td>Shear Moduli: <input type="text" value="261067.96"/></td> <td></td> </tr> </table>		<b>Analysis Property Data</b>	<b>Design Property Data</b>	Mass per unit Volume: <input type="text" value="0"/>	Reinforcing yield stress, $f_y$ : <input type="text" value="8640"/>	Weight per unit Volume: <input type="text" value="0.1501"/>	Concrete strength (Cylinder), $f_c$ : <input type="text" value="576"/>	Modulus of Elasticity: <input type="text" value="519120"/>	Shear steel yield stress, $f_{ys}$ : <input type="text" value="8640"/>	Poisson's Ratio: <input type="text" value="0.2"/>	Concrete shear strength, $f_{cs}$ : <input type="text" value="300"/>	Coeff of Thermal Expansion: <input type="text" value="5.500E-06"/>		Shear Moduli: <input type="text" value="261067.96"/>	
<b>Analysis Property Data</b>	<b>Design Property Data</b>														
Mass per unit Volume: <input type="text" value="0"/>	Reinforcing yield stress, $f_y$ : <input type="text" value="8640"/>														
Weight per unit Volume: <input type="text" value="0.1501"/>	Concrete strength (Cylinder), $f_c$ : <input type="text" value="576"/>														
Modulus of Elasticity: <input type="text" value="519120"/>	Shear steel yield stress, $f_{ys}$ : <input type="text" value="8640"/>														
Poisson's Ratio: <input type="text" value="0.2"/>	Concrete shear strength, $f_{cs}$ : <input type="text" value="300"/>														
Coeff of Thermal Expansion: <input type="text" value="5.500E-06"/>															
Shear Moduli: <input type="text" value="261067.96"/>															
<input type="button" value="OK"/> <input type="button" value="Cancel"/>															

### 4. Define and assign frame sections

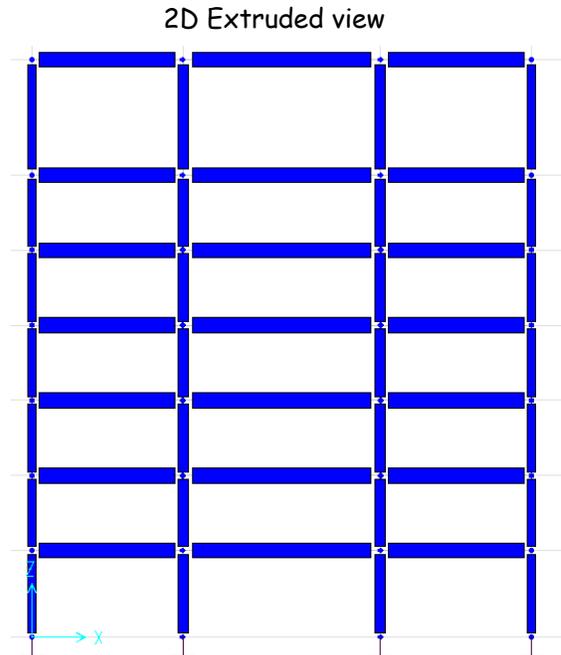
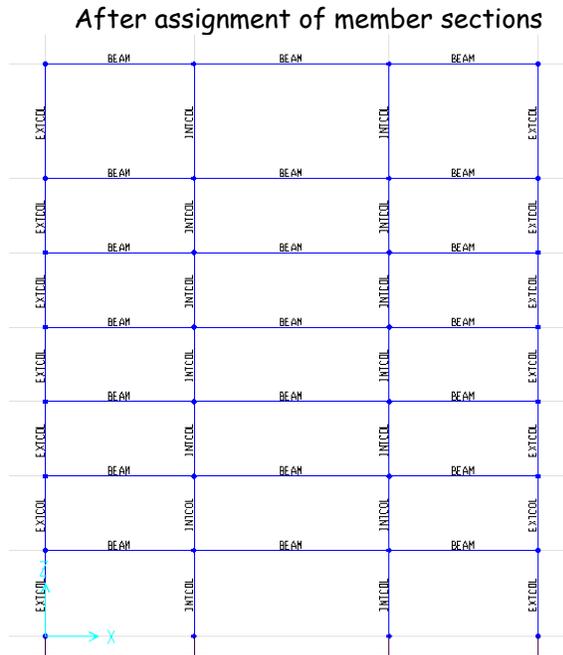
Define the beam, exterior column, and interior column sections

Menu: Define > Frame Sections > Add Rectangular

Note: Don't forget to change the units to Kips-in before specifying member dimensions



After the member sections are defined, select members with common sections and assign the respective section from menu: Assign > Frame > Sections. After assigning the sections, 2D/3D Extruded shape of the members can be seen from Menu: View > Set Elements > Extruded shape or using the shortcut menu button .



## 5. Define static load cases

Define the type of load cases considered in design. Dead load (DL), live load (LL), and earthquake loading (E) are considered in this example. New load cases can be added from the below menu item.

Menu: Define > Static Load Cases

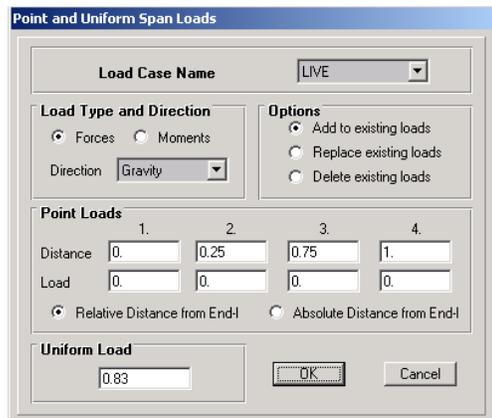


## 6. Assign member and joint loads for each load case

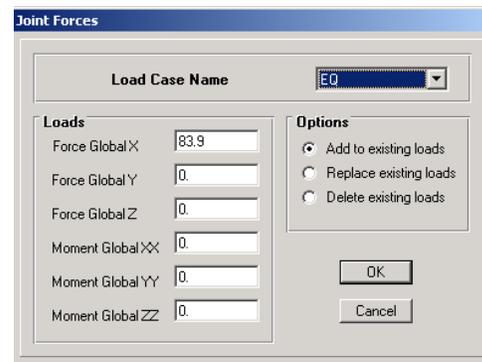
Select members or joints with similar load type and values, and assign proper loading using quick menu buttons  or below menu items.

Uniform loading of frames: Menu: Define > Frame Static Loads > Point and Uniform

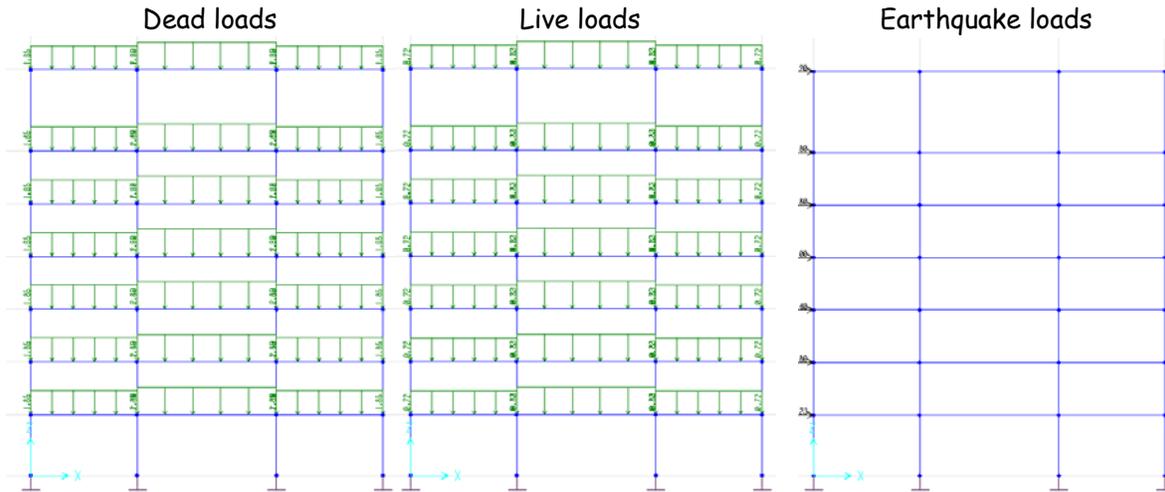
Point loading of joints: Menu: Define > Joint Static Loads > Forces



Uniform load definition for frames



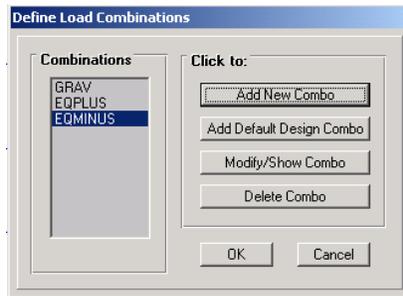
Point load definition for joints



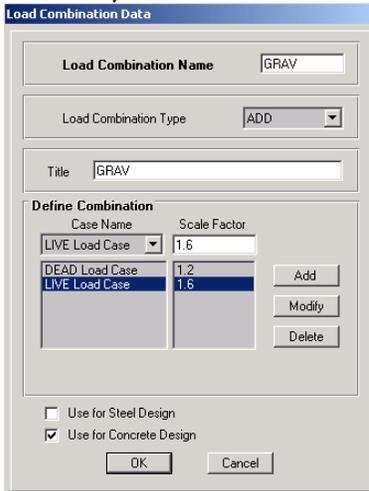
### 7. Define load combinations

Define load combinations with proper load factors from Menu: Define > Load Combinations

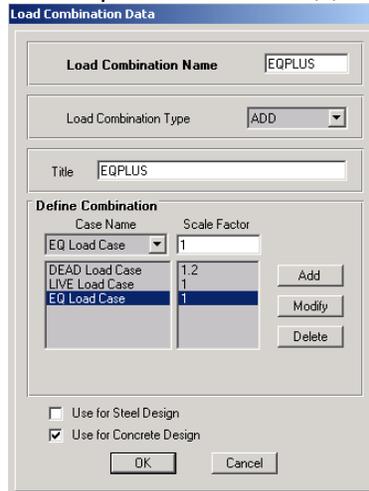
#### Load combinations



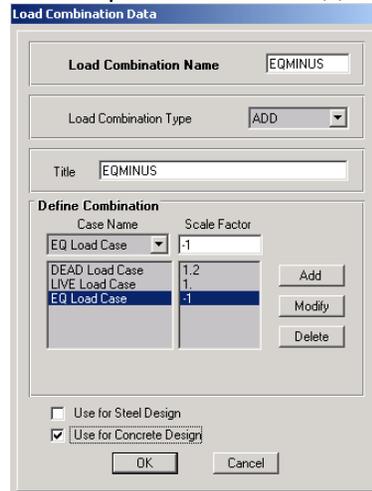
#### Gravity load combination



#### Earthquake load comb. (+)

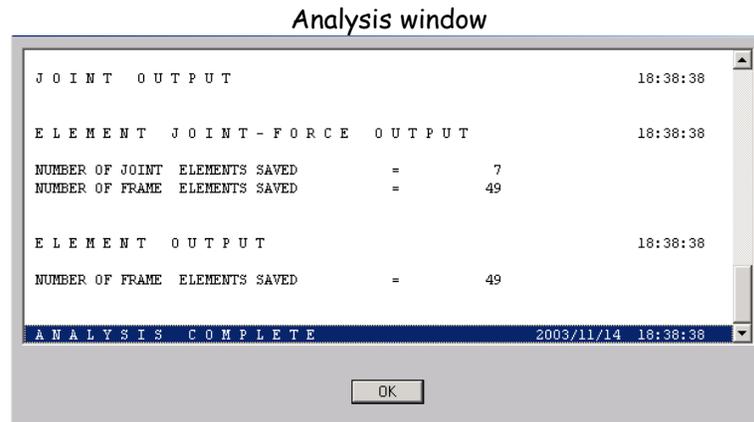


#### Earthquake load comb. (-)

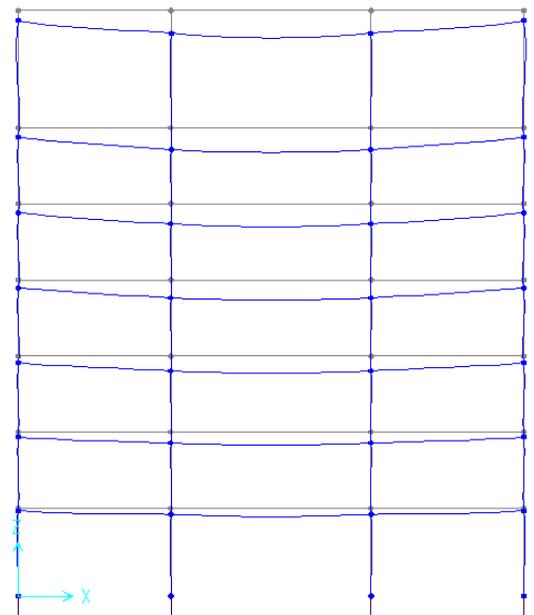


## 8. Run Analysis

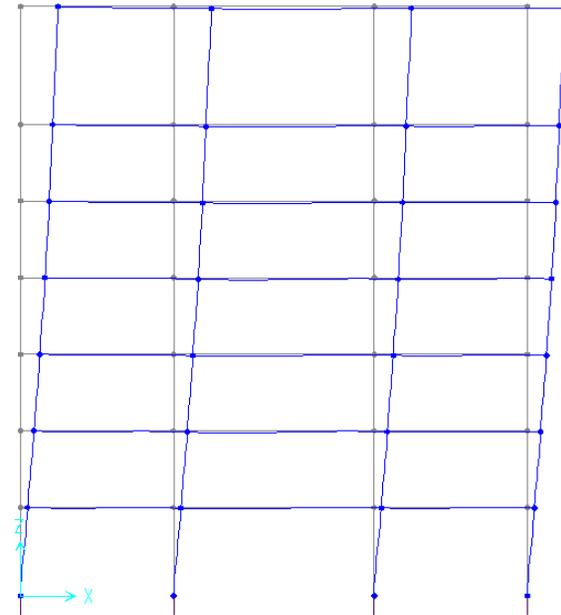
We are now ready to run the analysis either from Menu: Analyze > Run, or using the quick menu button 



Deformed shape under gravity load combo



Deformed shape under earthquake load combo



## 9. Print Output Tables

Member displacements and frame forces can be printed to tables from Menu: File > Print Output Tables. You can either select the envelope option to determine the maximum values, or the spreadsheet format to check values for each load combination.

