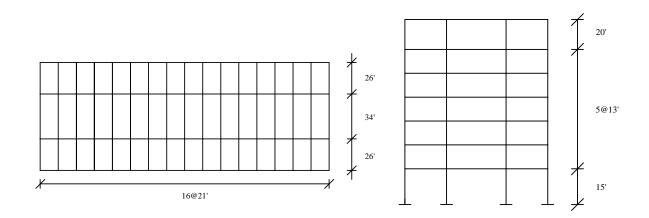
# SAP 2000 Tutorial Session Notes

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"×31" Exterior columns: 14"×18" Interior columns: 16"×22"

### Loads:

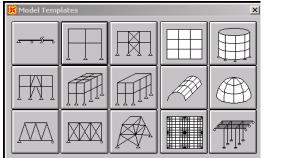
Exterior beams:	LL=0.72 kips/ft
	DL=1.85 kips/ft
Interior beams:	LL=0.83 kips/ft
	DL=2.10 kips/ft

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.

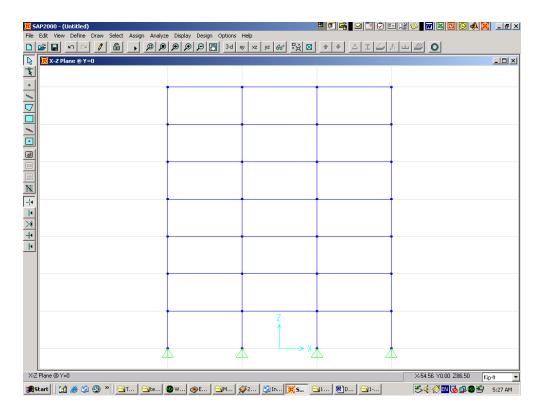
# Analysis Steps

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



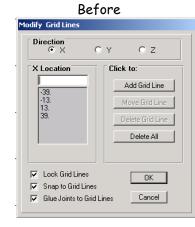
✓ Restraints ✓ Gridlines	Number of Stories Number of Bays Story Height Bay Width	7 3 13 26	OK Cancel

Result



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

#### X-grids before and after



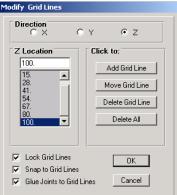
# Z-grids before and after

Direction	oy oz
Z Location 0. ▲ 13. 26. 39. 52. 65. 65. 78. ▼	Click to: Add Grid Line Move Grid Line Delete Grid Line Delete All
<ul> <li>✓ Lock Grid Lines</li> <li>✓ Snap to Grid Line:</li> <li>✓ Glue Joints to Grid</li> </ul>	

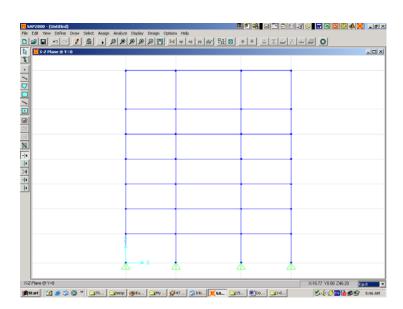
Modify Grid Lines
Direction
X Location         Click to:           86.         Add Grid Line           0.         Add Grid Line           26.         Move Grid Line           86.         Delete Grid Line
Glue Joints to Grid Lines Cancel

After





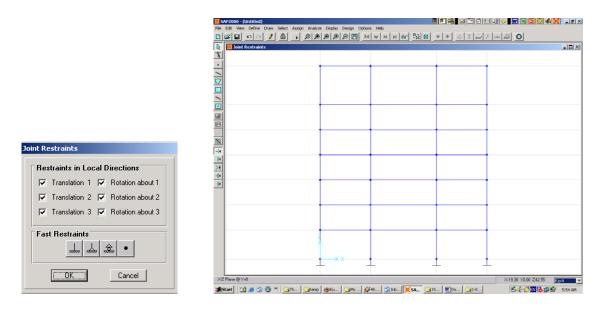
#### Result:



#### Fall 2003

# 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  $\boxed{A \equiv \Box = A \equiv \Box = 0}$  (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 Name CONC			
Type of Material • Isotropic O Orthotropic	C Anisotropic	Type of Design Design	Concrete
Analysis Property Data Mass per unit Volume	0.	Design Property Data Reinforcing yield stress, fy	8640
Weight per unit Volume	0.1501	Concrete strength (Cylinder), fc	576
Modulus of Elasticity Poisson's Ratio	519120	Shear steel yield stress, fys Concrete shear strength, fcs	8640
Coeff of Thermal Expansion	5.500E-06		,
Shear Modulii	261067.96		

# 4. Define and assign frame sections

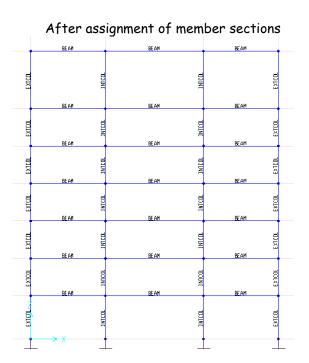
Define the beam, exterior column, and interior column sections

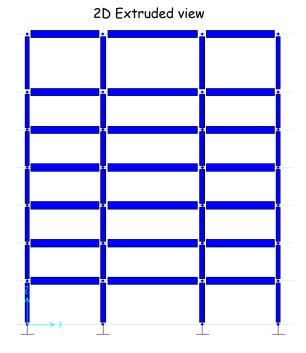
Menu: Define > Frame Sections > Add Rectangular

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

beam	exterior column	interior column
Rectangular Section	Rectangular Section	Rectangular Section
Section Name BEAM	Section Name [EXTCOL]	Section Name INTCOL
Properties Modification Factors Material CONC	Properties Section Properties Modification Factors Material CONC	Properties         Modification Factors         Material         CONC         Image: Conc and a concentration factors         Image: Conconcentration factors </th
Depth (13)         I           Widh (12)         T4	Dimensions         P           Depth (13)         18           width (12)         14.           3 4	Dimensions         P           Drph (13)         Image: Comparison of the second s
Concrete Derrforcement OK Cancel	Concrete	Concrete

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

Loads		Self Weight	Click to:
Load	Туре	Multiplier	Add New Load
DEAD	DEAD	0	
DEAD LIVE	DEAD	0	Change Load
EQ	UVE QUAKE	0	Delete Load
			OK
,	)	,	Cancel

# 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 for 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

Load Case Name	LIVE
Load Type and Direction Forces Moments Direction Gravity	Options Add to existing loads Replace existing loads Delete existing loads
Point Loads         1.         2.           Distance         0.         0.25           Load         0.         0.	3.         4.           0.75         1.           0.         0.
Relative Distance from End-I	C Absolute Distance from End-I

Uniform load definition for frames

Load Cas	e Name	
Loads Force Global X	83.9	Options
Force Global Y Force Global Z	0.	<ul> <li>Replace existing loads</li> <li>Delete existing loads</li> </ul>
Moment alobal AA	0.	ОК

Point load definition for joints

Add

Modify

Delete

Dead loads	Live loads	Earthquake loads
		• • • • • • • • • • • • • • • • • • •

# 7. Define load combinations

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



#### Gravity load combination Earthquake load comb. (+) Earthquake load comb. (-) Load Combination Data Load Combination Data Load Combination Data GRAV EQPLUS EQMINUS Load Combination Name Load Combination Name Load Combination Name ADD ADD 💌 ADD 💌 Load Combination Type • Load Combination Type Load Combination Type Title GRAV Title EQPLUS Title EQMINUS Define Combination Define Combination Define Combination Case Name Sc EQ Load Case 💽 1 Case Name Scale Case Name So EQ Load Case 💌 1 Scale Factor Scale Factor Scale Factor DEAD Load Case LIVE Load Case EQ Load Case DEAD Load Case 1.2 LIVE Load Case 1. EQ Load Case -1 DEAD Load Case LIVE Load Case 1.2 1.2 Add Add Modify Modify Delete Delete Use for Steel Design Use for Steel Design Use for Steel Design 🔽 Use for Concrete Design Use for Concrete Design 🔽 Use for Concrete Design OK OK OK Cancel Cancel Cancel

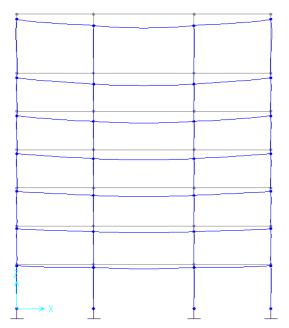
Fall 2003

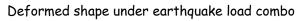
# 8. Run Analysis

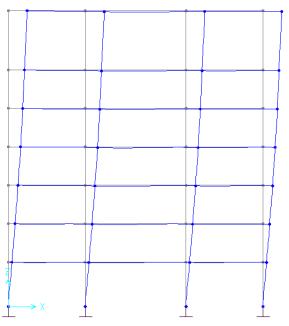
We are now ready to run the analysis either from Menu: Analyze > Run, or using the quick menu button

Analysis window	
JOINT OUTPUT	18:38:38
ELEMENT JOINT-FORCE OUTPUT	18:38:38
NUMBER OF JOINT ELEMENTS SAVED = 7 NUMBER OF FRAME ELEMENTS SAVED = 49	
ELEMENT OUTPUT	18:38:38
NUMBER OF FRAME ELEMENTS SAVED = 49	
ANALYSIS COMPLETE 200	3/11/14 18:38:38 🔽
OK	

Deformed shape under gravity 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.

Displacements	🔲 Shell Forces 📃 Soli	d Stresses
Reactions	🗖 Shell Stresses 🗖 Soli	d Joint Forces
Spring Forces	🔲 Shell Joint Forces 📃 Gro	up Force Sum
Frame Forces	Plane Stresses	
Frame Joint Forces	Plane Joint Forces	
NLLink Forces	Asolid Stresses Se	lect Loads
NLLink Joint Forces		
E Selection Only	Envelopes Only	
Print to File	🔲 Spreadsheet Format 🔲 App	end
File Name	C:\Documents and	