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

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

Before

After

Z-grids before and after

Before

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).

![Joint Restraints](image)

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](image)
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
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.

![Analysis window](image)

**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.