SAP2000 Tutorial

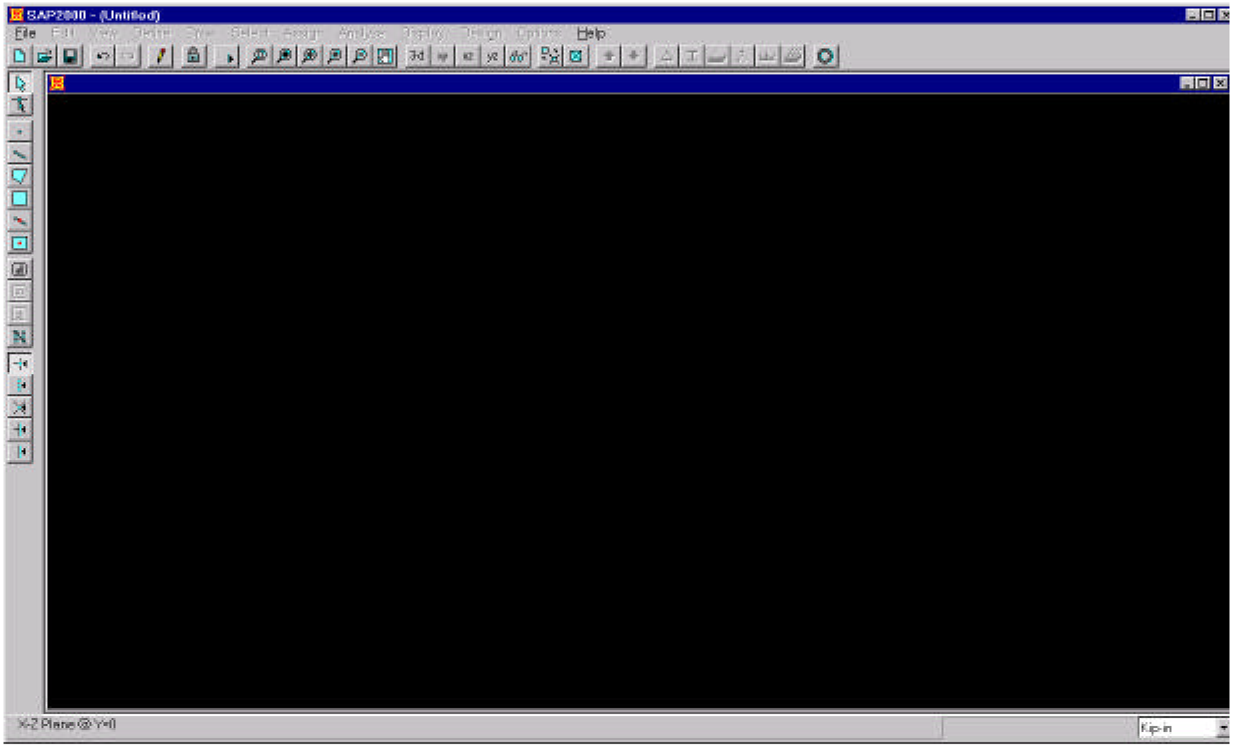
By

Rakesh K. Goel

Department of Civil and Environmental Engineering
California Polytechnic State University
San Luis Obispo, CA 93407

Geometric Model

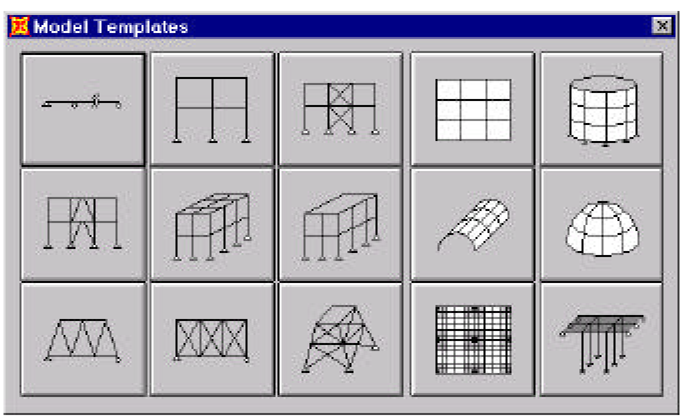
Start the program. The following screen pops.



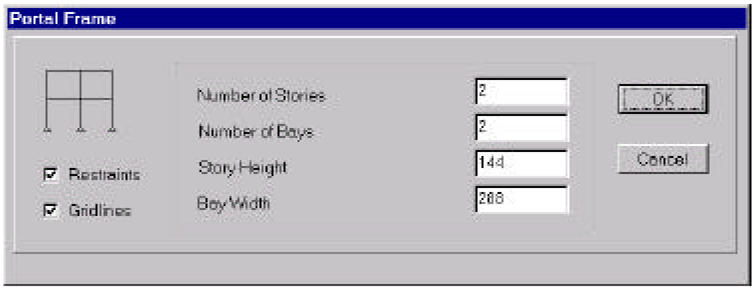
On the bottom right corner, select the units to be used in the analysis.

Click on **File**

Click on **New Model from Template** to get the following screen.

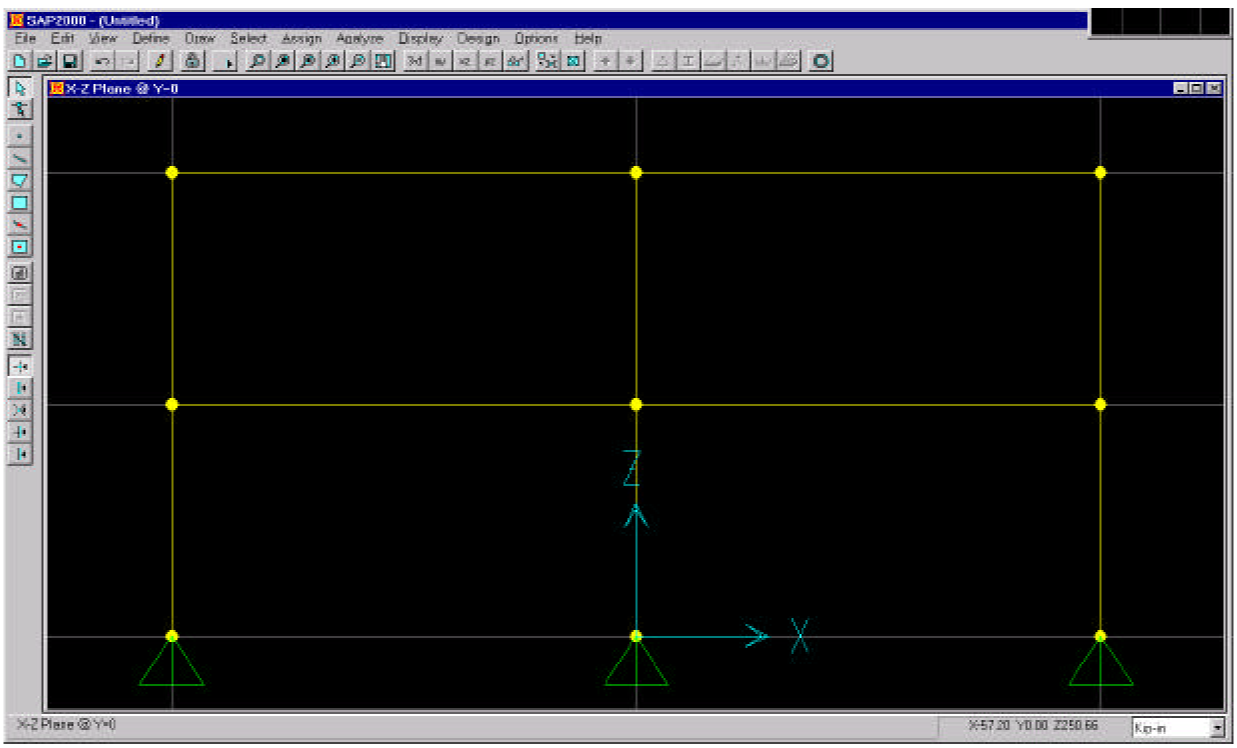


Choose a template that fits your need. For example, to generate a two-dimensional frame, click on the second icon in the first row. The following information screen pops up.

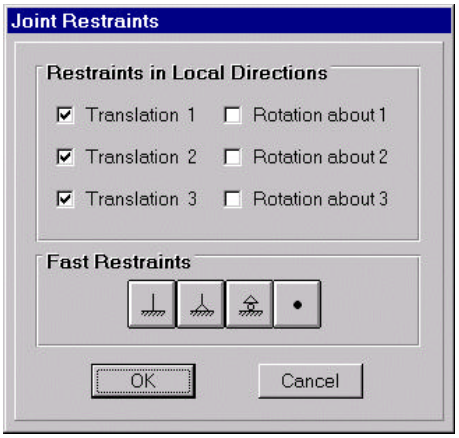


Specify the number of stories, number of bays, story height, and bay width. The units should be consistent with those selected in the first step.

Click on **OK** and a basic outline of the model would appear on screen. This model has correct coordinates of all nodes.



To change the boundary conditions, select joints and then click on the joint restraint icon (little triangle on the icon bar) to get the following screen.



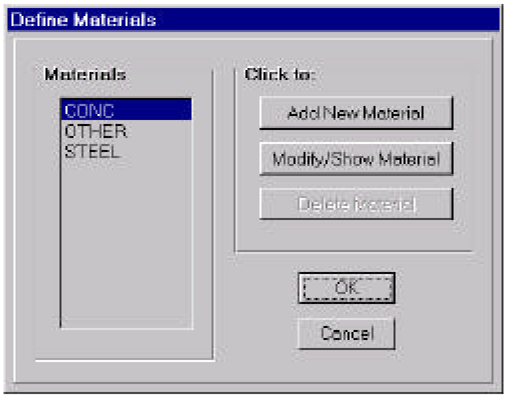
Click on the appropriate **Fast Restraint** icon and click **OK**.

Material Properties

If more than one material is used, repeat the process for each new material.

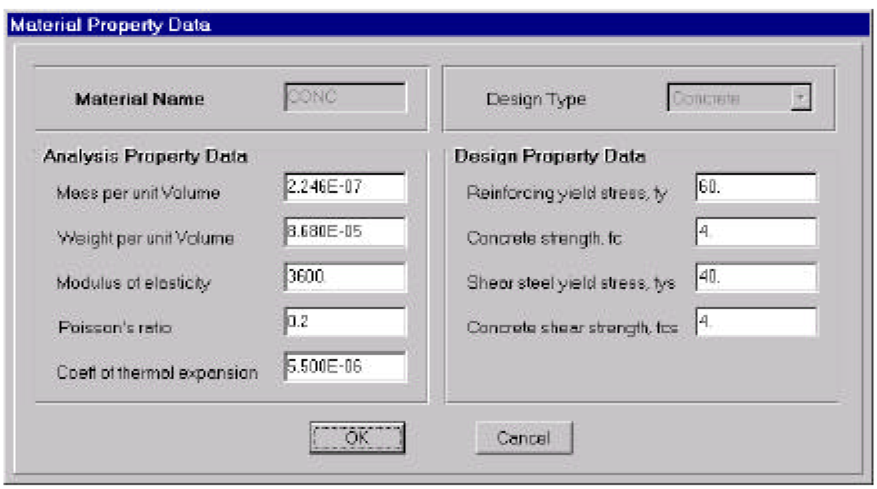
Click on **Define**

Click on **Material** to get the following screen.



The program has default properties of concrete and steel. If you want, you can add new material by clicking on **Add New Material** and following the directions.

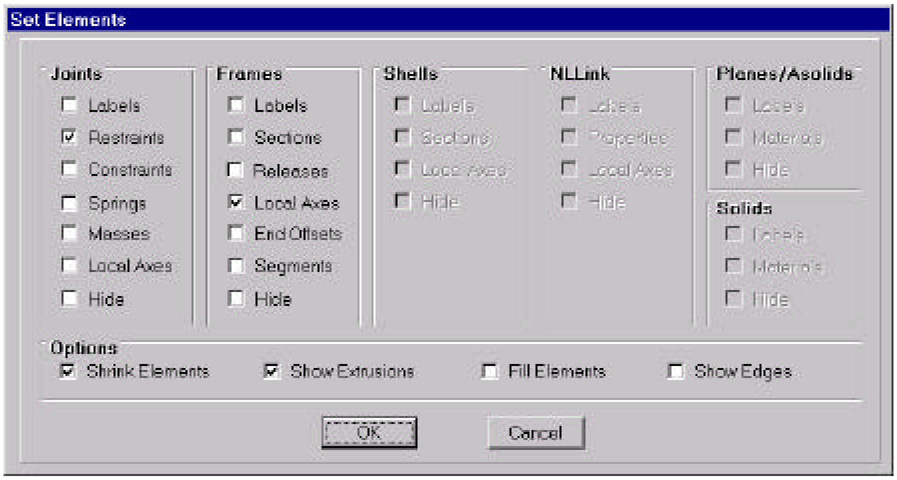
For checking or modifying properties of steel or concrete, click on **Modify/Show Material**. The following screen pops up. Check to see if these material properties are acceptable. If not, you can modify them by going to the appropriate box and re-entering the value.



CAUTION: If you specify non-zero value of weight per unit volume, the program by default would calculate the weight of structural elements and use them in analysis. If you specify value of Dead Load that contains the self-weight of structural elements, specify zero unit-weight (or turn it off latter). Similarly, if your nodal masses contain mass due to self-weight, specify zero unit-mass.

Element Data

Define the element view options. Click on **View** and then on **Set Elements …** to get the following screen.



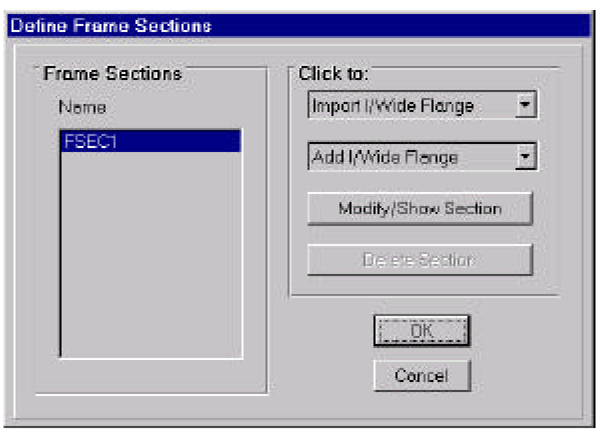
Check all appropriate boxes under the **Joint** sections and **Frame** section.

It is also useful to check Options for **Shrink Elements** and **Show Extrusions**. The last option is useful in visually checking the correct orientation of beams and columns (more later on this).

Click **OK**.

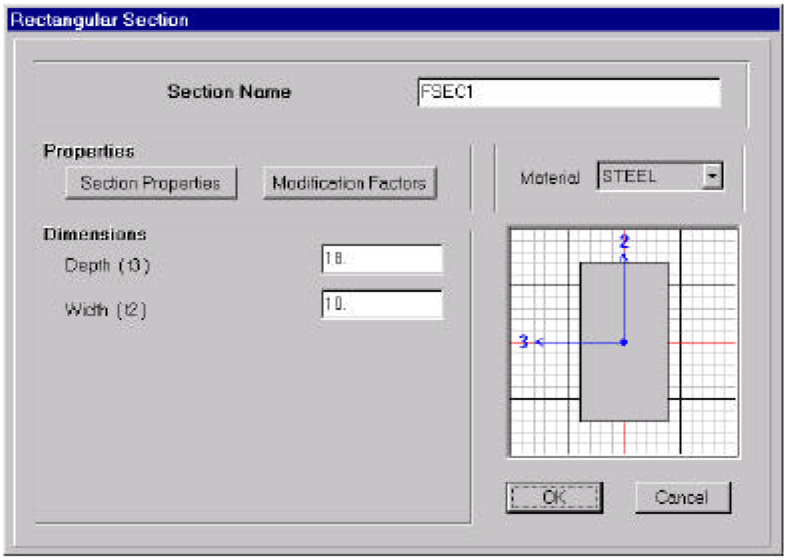
To define frame section properties, repeat the following process as many times as different sections to be used in your analysis.

Click on **Define** and then on **Frame Sections** to get the following screen.



There is default rectangular section FSEC1. You can modify this section, or you can Import or Add section.

To modify the existing section, select **FSEC1** and then click on **Modify/Show Section** to get the following screen.

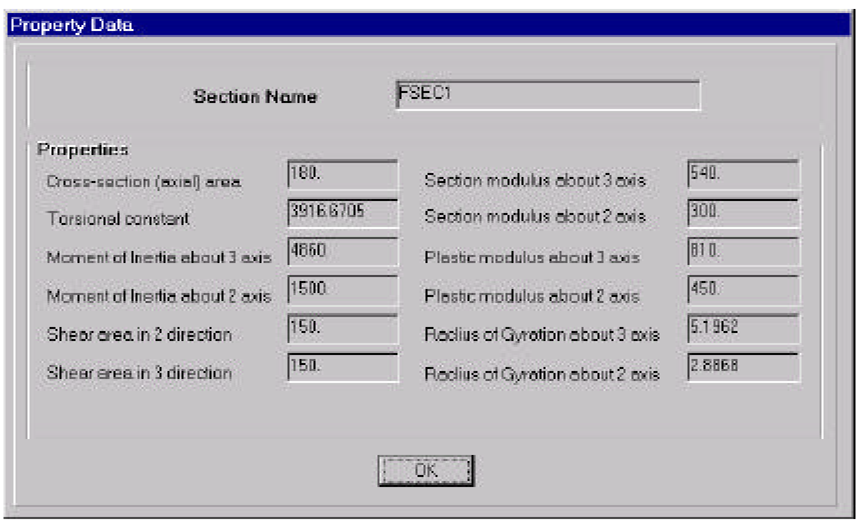


Specify a name for your section in **Section Name** box.

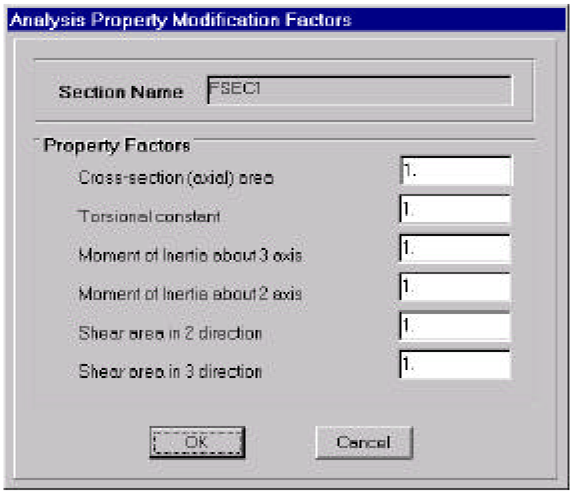
Select the appropriate **Material**. The material propoerties were defined earlier.

Modify the **Depth** and **Width**.

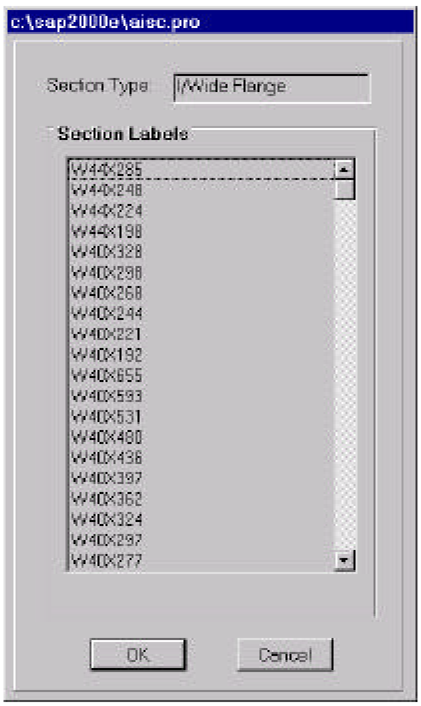
View the calculated properties of the section by clicking on **Section Properties**.



If you want to modify the properties, you can do so by clicking on **Modification Factor** and specifying the appropriate multipliers. The properties shown are based on gross section. For reinforced concrete, it is usual to consider moment of inertia to be a fraction of the gross intertia. That fraction could be enetered as **Property Factor** in the Modification Factor screen.



You can also import one of the predefined sections, such as AISC standard section. For this purpose, go to the import drop box and click on a section type. For example, clicking on **Import/Wide Flange** gives the following screen



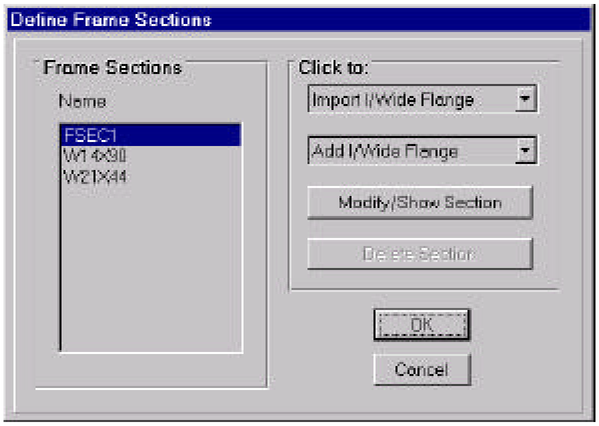
Choose the appropriate section, and click **OK**. If you are doing this for the first time, the program would ask you to choose from standard data files. For example, AISC properties are in aisc.pro.

You can view the section from the Modify/Show option of the **Define Frame Section** box.

To assign section properties to each frame member, select all elements to which you want to assign the same section.

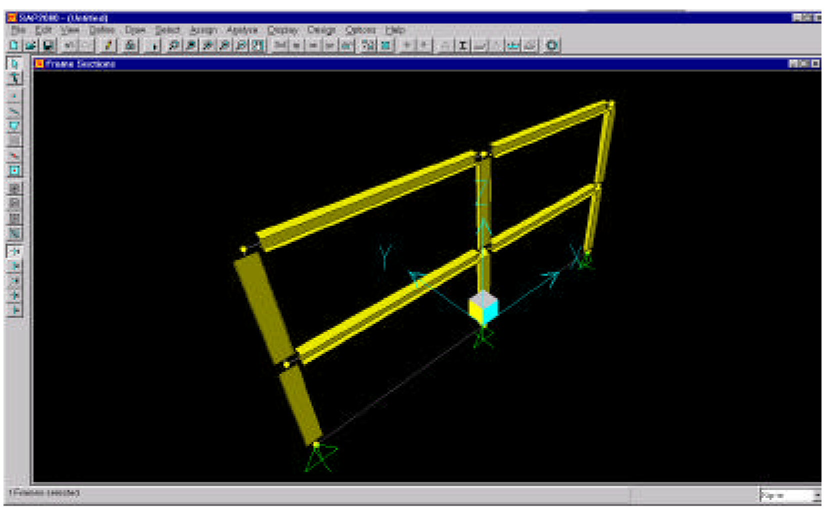
Click on **Assign**, then on **Frame Section** to get the following screen.

Choose the frame section that you want to assign to the selected frame elements and then click **OK**.



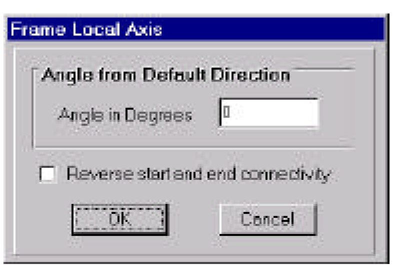
Completing the process gives you, for example the following structural model. Note that SAP by default places frame elements for strong axis bending.

The axis can be changed by the following process, which is described by changing the local axis for one of the columns.



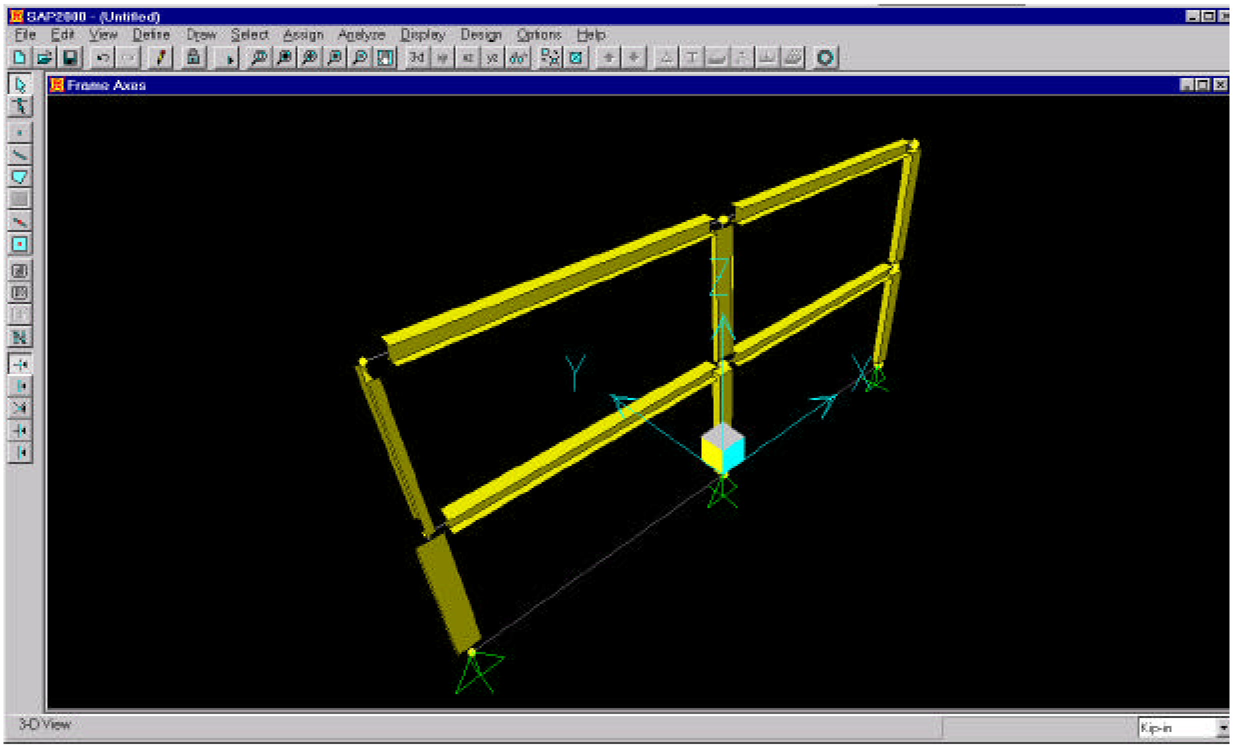
Select the element you want to change the local axis for.

Click on **Assign** then on **Frame** and then on **Local Axes** to get the following screen



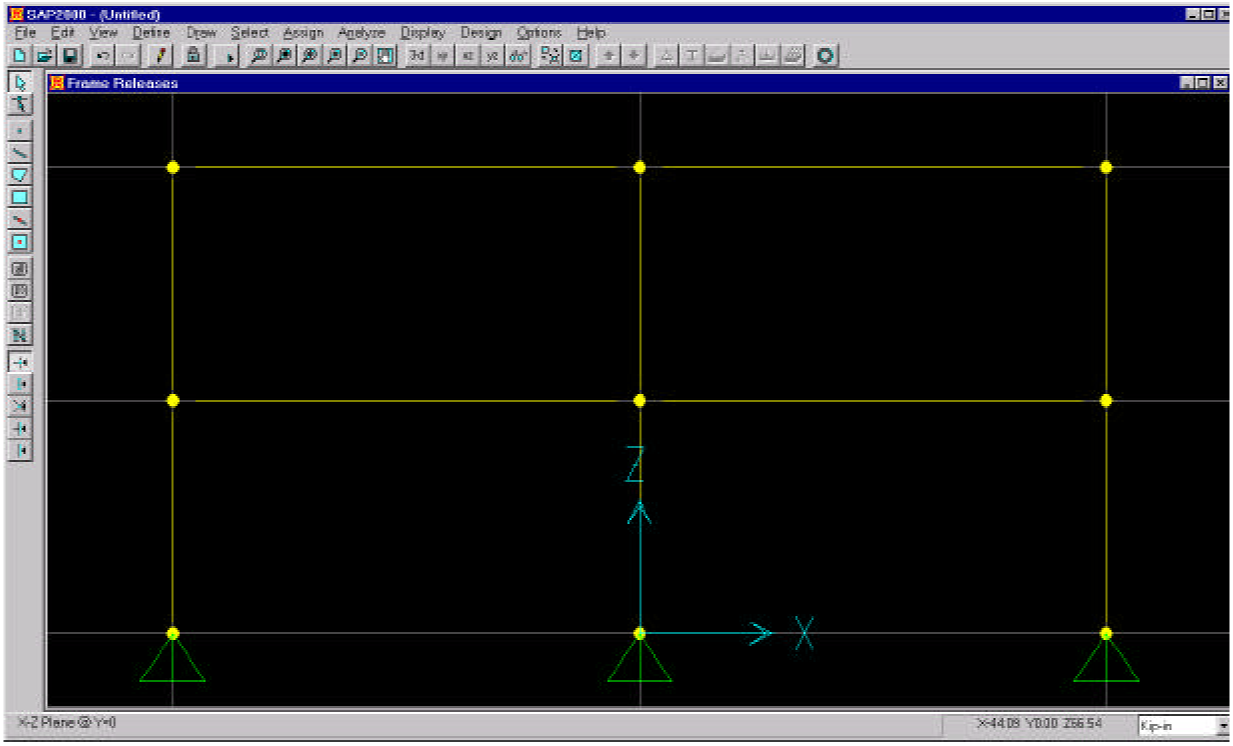
To rotate the angle from default direction, say by 90 degrees, enter the desired **Angle in Degrees** and click **OK**.

You would see the change in the direction of the columns as shown below.

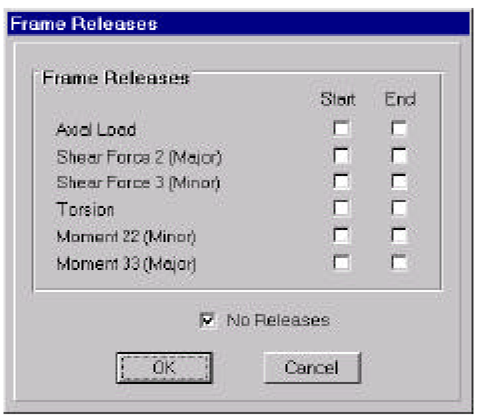


After you have specified all elements and checked the orientation, it is useful to go back to the line digarm. This can be done by clicking **View** then **Set Elements** and then clearing the **Show Extrusion** box.

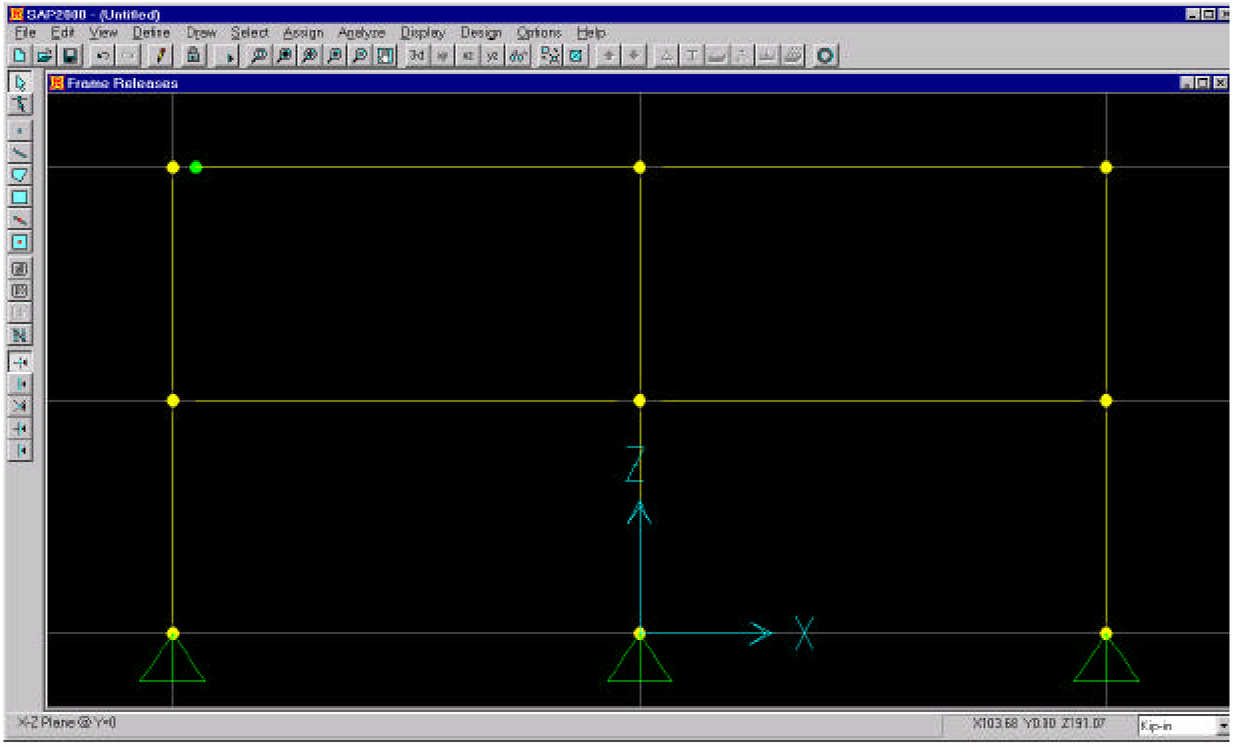
You may also want to check on **Frame Release** to be able to view any member end releases. Shown below is the two dimensional model.



By default, SAP also considers that both ends of the frame member have rigid connections. A pin connection at any or both end, can be specified by selecting an element, then clicking on **Assign** then on **Fram**e and then on **Releases** to get

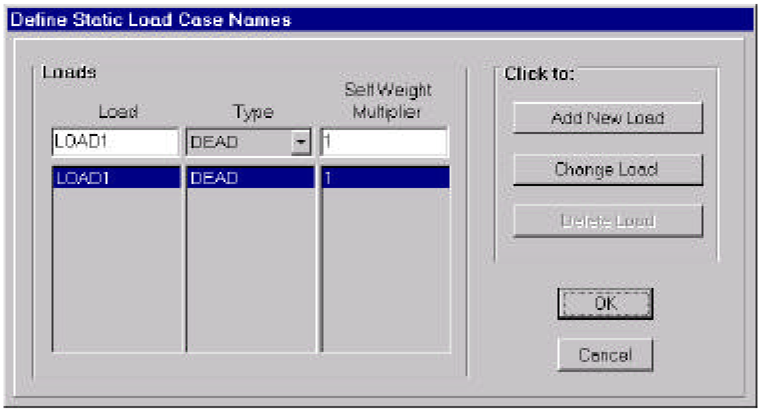


Pin at ends can be introduced by releasing the moment in appropriate direction. It is also possible to release other quantities from the same screen. Following figure shows moments released at starting end of the beam at second story of first bay.


SAP2000 - (Untitled)
File Edit View Define Draw Select Assign Analyze Display Design Options Help
Frame Releases
Z
X
X-Z Plane @ Y=0
Kip-in

Static Analysis

Define static load cases by clicking on **Define** and then on **Static Load Case** to get the following screen.

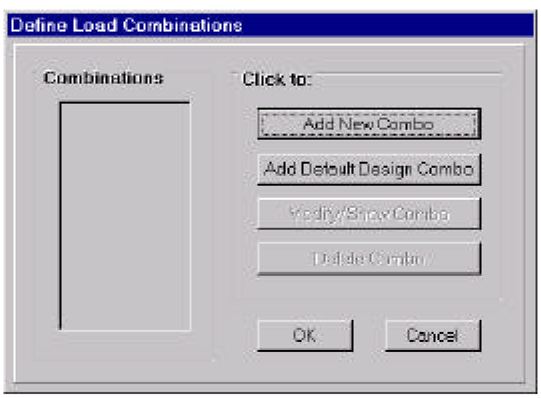


You can specify a name for your load (e.g., DL, LL, EQ etc.) in the **Load** box and specify the load type in the **Type** box.

Caution: If your specified load values include self-weight of the elements, you should specify **Self Weight Multiplier** to be zero. Otherwise, the self weight would be counted twice.

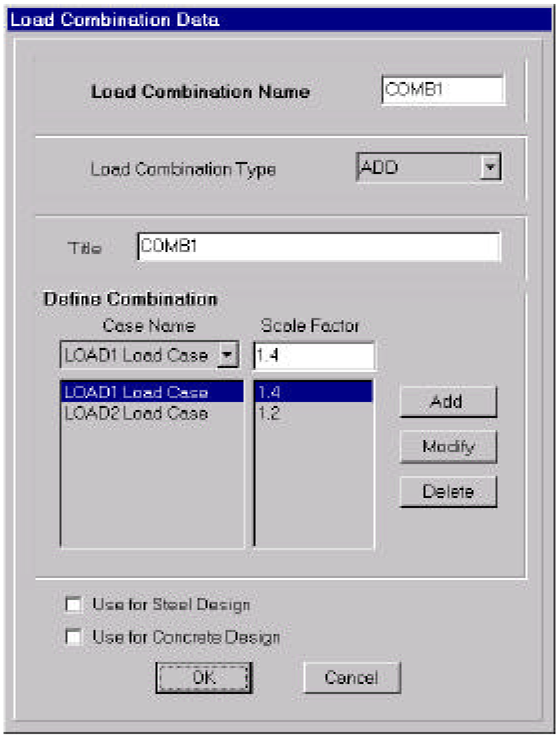
You can add as many load types as you need for your analysis.

Load cobminations can also be specified. For this purpose, click on **Define** and then on **Combination** to get the following screen.



Define as many combinations as desired.

To add a new combination, click on **Add New Combo** to get the following screen.

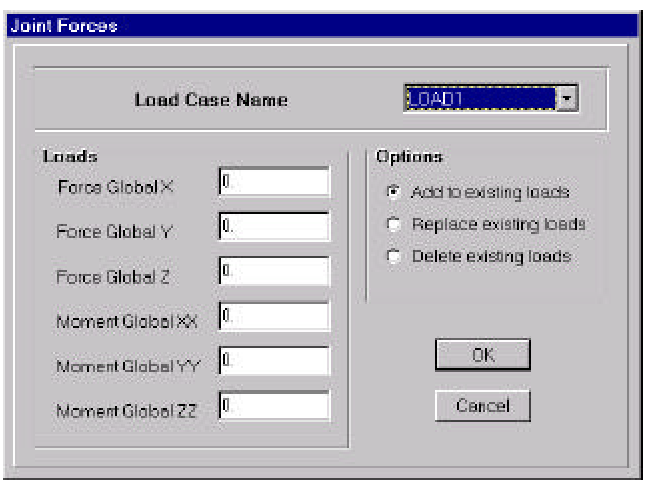


Specify a **Load Cobination Name**.

From the **Case Name** drop box, select the load case and from the **Scale Factor** specify the multiplier.

Built up your load combination by adding other load case. The figure shows a selection of 1.4 × Load 1 + 1.2 × Load 2.

To specify loads at joints, select the joint, click on **Asign,** click on **Joint Static Loads,** and click on **Forces** to getthe following screen.



Input the load value in appropriate box. Note that positive force is along positive axis direction.

Select the **Load Case Name** from the drop box to associate the specified values to a particular load case.

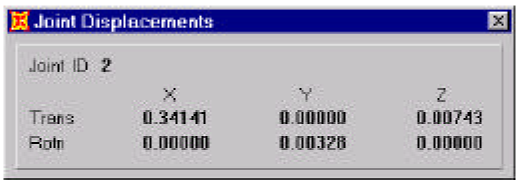
You can also modify an existing load by adding to it, replacing it, or deleting it by checking the appropriate box.

Click **OK** to finish specifying load at the selected joint.

Repeat the process for all joints that have applied loads.

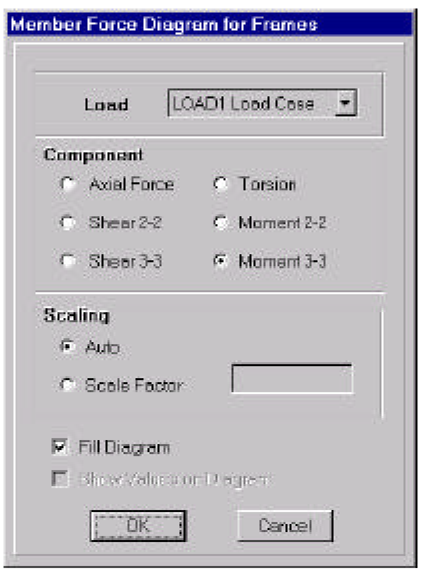
Static analysis can be performed by clicking on **Analyze** and then on **Run** or by clicking on the ▶ icon from the toolbar.

By default, SAP displays the deformed shape for the first load case. Displacements at a joint can be displayed by selecting the joint and then right clicking on the mouse.

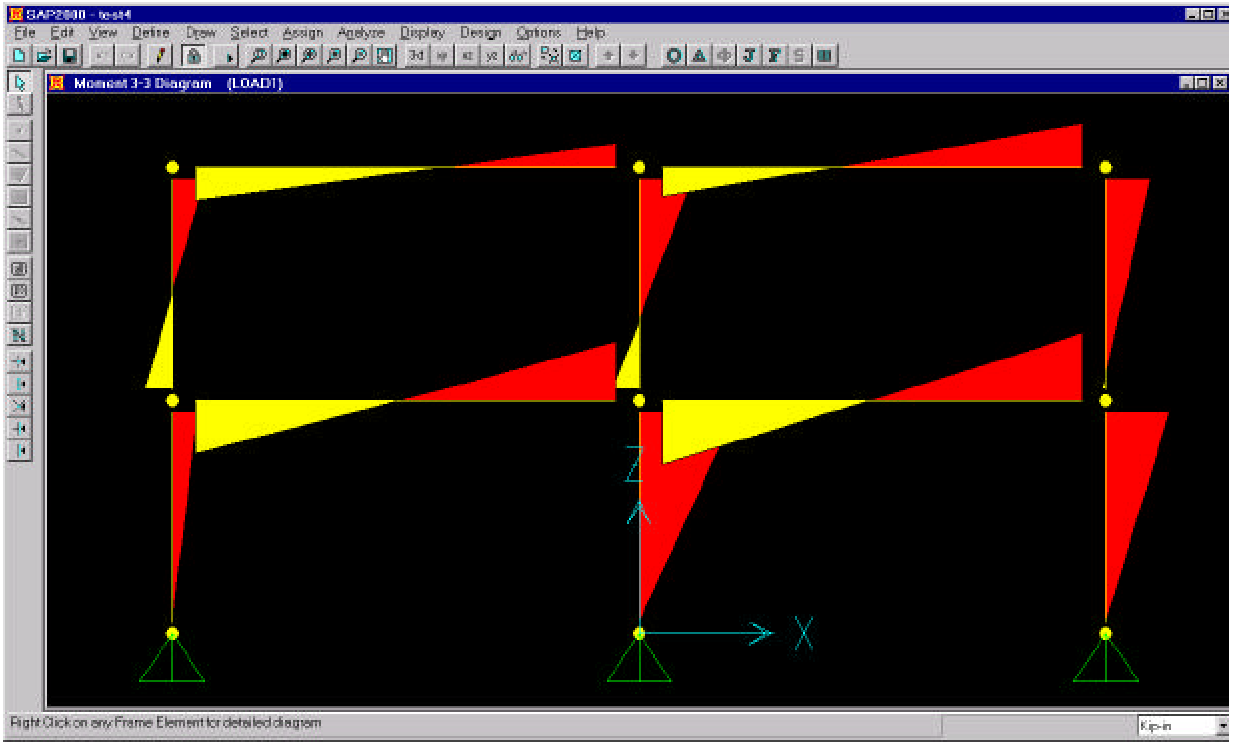


To display other quantities, click on **Display** and picking the desired option.

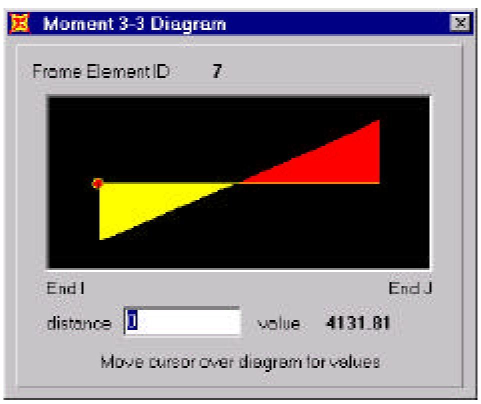
To display element forces, click on **Display**, **Show Element Forces/Stresses,** and **Frame** to get the following screen.



Select the load case or cobmination and the component for which you want to display the element forces. For example, to display bending moment about strong axis, select **Moment 3-3** to get the following diagram on screen.



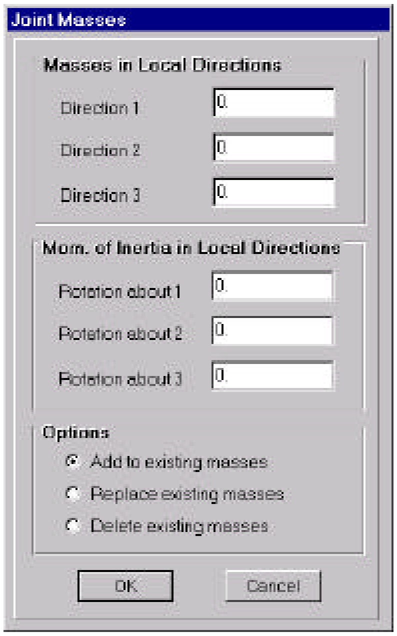
To display numerical values, select an element and then right click on the mouse to get a smalle screen as shown below.



Values at different location can be displayed by moving the red dot along the element length.

Dynamic Analysis

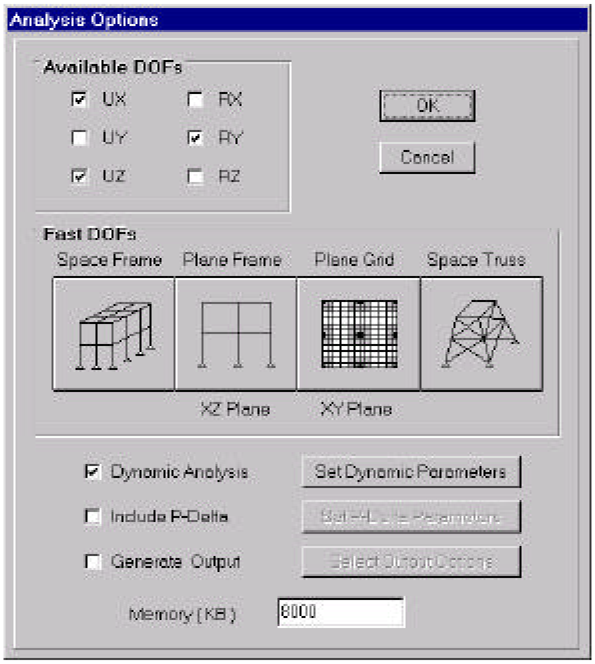
Specify lumped masses at joint. For this purpose, click on **Assign, Joint,** and **Masses** to get the following screen.



Enter appropriate mass values. The units should be consistent with the units specifed for youe analysis. Do not specify mass in units of kips; the units should be kips-sec^2/inch (or any other appropriate value).

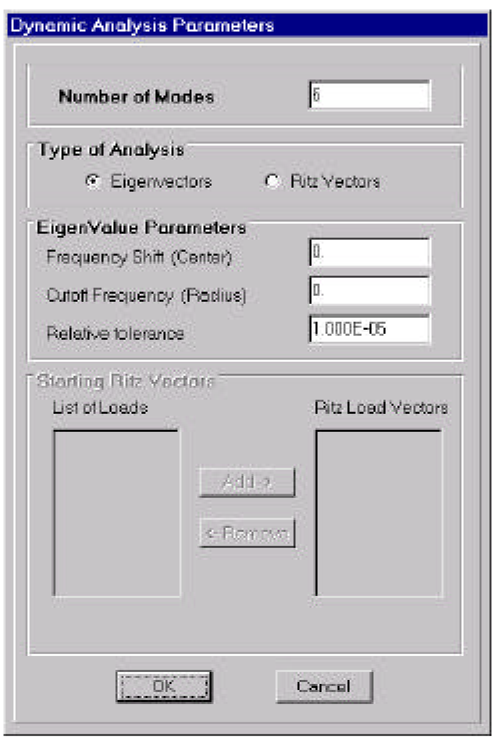
You can add this mass to existing mass, replace existing mass, or delete existing mass from the same screen.

To get the mode shapes and frequencies, click on **Analyze** and then on **Set Options** to get the following screen.

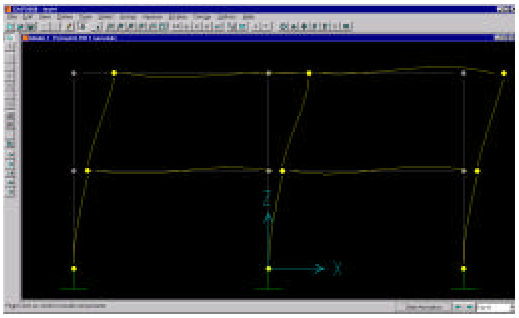


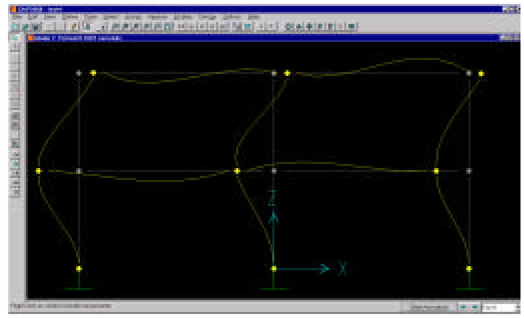
Select the Available DOFs either by individually checking on various DOF or by clicking on Fast DOFs option. For example, for analysis of a frame in two dimensions, you would select **Plane Frame**.

Check the box for **Dynamic Analysis** and then click on **Set Dynamic Parameters** to specify the number of modes to be determined.



Run the model to get mode shapes and frequencies.





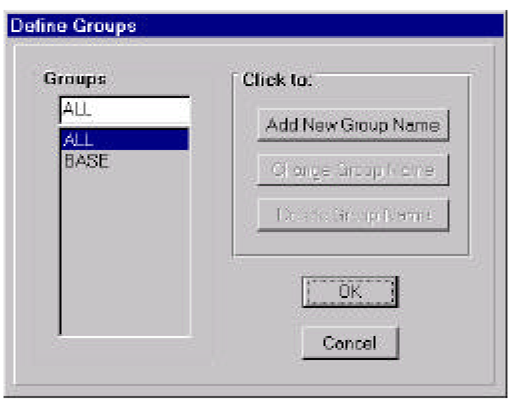
You can change the mode shape that is displayed on screen from the **Display** tool or from clicking on the ➡ symbol at the bottom right corner of the screen.

For dynamic response spectrum analysis, **Define** the **Response Spectrum Function** and then **Response Spectrum Case** using the procedure similar to that for static case.

In dynamic analysis, it is useful to display base shear and base overturning moment for comparison with results from equivalent static analysis. This can be done as follows.

Define a group consisting of joints at the base and all elements connected to these joints.

Click on **Define** and **Groups** to get the following screen.

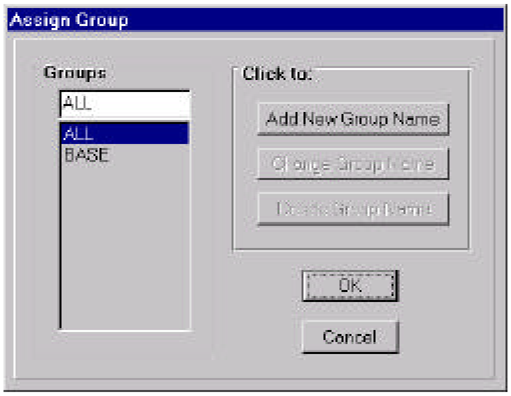


Specify a name for the group in the **Groups** box and click on **Add New Group Name**.

Click **OK**.

Select a group of joints and the elements directly connected to only one side of them. For example if the base shear for a structure is required, the group should consist of the joints at the base of the structure and the frame elements above them that are directly connected to them.

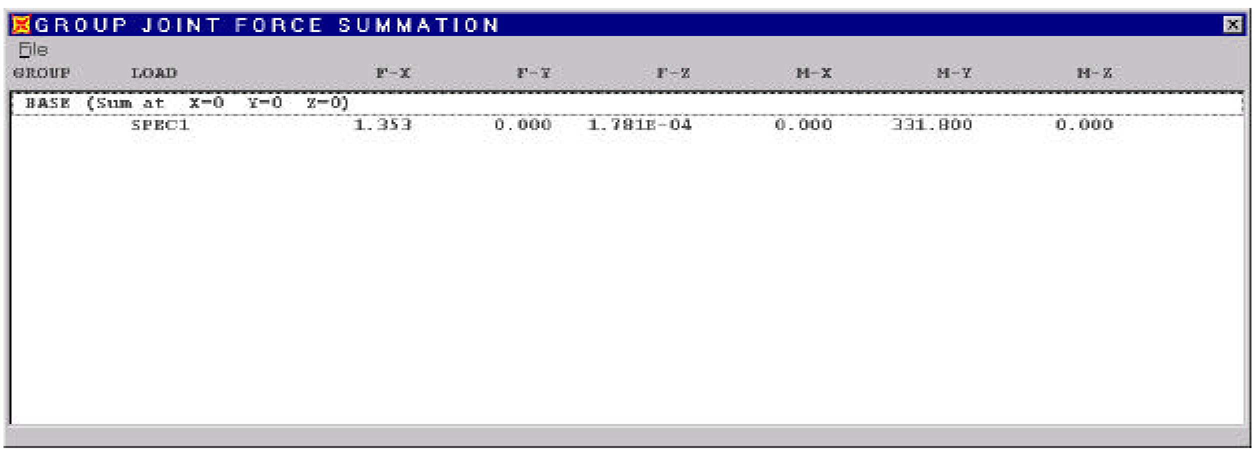
Click on **Assign** and **Group Name** to get the following screen.



Select the name in the **Groups** box that needs to be assigned to the selected group.

Click **OK**.

To display results for this group, after the analysis has been completed, click on **Display** and **Show Group Joint Force Sum**. The following screen would display the results.



Modifying Model

After you have run the analysis once, the model should be unlocked from the toolbar. This would cause all the analysis results to be deleted.

Make whatever changes are necessary and rerun the model.