

# SADZAM

Prepared by:

Eng.\ Magdy Genidy  
Eng.\ Abdel Razeq Abdel Rahman  
Eng.\ Walid Abdel Raouf

## INTRODUCTION

### General

The most famous program used in structural analysis in the last 30 years is *Structural Analysis Program (SAP)* which designed by *Computer and Structures, Inc. company (CSI)*. This program had been improved from 1970 up to now. There were many versions had been edited by the company like *SAP, SOLIDSAP, SAPIV, SAP80, SAP90, and SAP2000*.

SAP is a finite element program for solving structures. In the finite element solution, the structure is to be divided into elements interconnected only at a finite number of nodal points. The finite element theory assumes that the behavior of a structure is similar to that of an assemblage of finite elements to which it is divided.

### SAP2000 Versions and Limitations

SAP2000 may be purchased in the following four versions:

#### SAP2000 Standard

- Capacity      1500 Nodes
- Analysis      Static and Dynamic Response Spectrum Analysis
- Elements      FRAME and SHELL Elements only
- Design      Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

**SAP2000 Plus**

- Capacity      No Practical Limit
- Analysis      Static, Dynamic Response Spectrum, Time History and Bridge Analysis
- Elements      FRAME, SHELL, PLANE, SOLID and ASOLID Elements
- Design      Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

**SAP2000 Nonlinear**

- Capacity      No Practical Limit
- Analysis      Static, Dynamic Response Spectrum, Time History, Bridge and Dynamic Nonlinear Time History Analysis
- Elements      FRAME, SHELL, PLANE, SOLID, ASOLID, and NLLINK ( as external Damping , Base Isolators, Gap and Hook Elements)
- Design      Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

**SAP2000 Educational**

- Capacity      30 Nodes
- Analysis      Static, Dynamic Response Spectrum, Time History, Bridge and Dynamic Nonlinear Time History Analysis
- Elements      FRAME, SHELL, PLANE, SOLID, ASOLID, and NLLINK ( as external Damping, Base Isolators, Gap and Hook Elements)
- Design      Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

## **Advantages of SAP2000**

- Has a very simple way to input data.
- Ability to input the geometrical data using AutoCAD program.
- Ability to solve any problem edited by using SAP90 program.
- Ability to use a template models in the program and to modify it.
- Solving much number of equations up to 60 000.
- Ability to design some structural elements.

## STRUCTURAL ANALYSIS PRINCIPALS

### Coordinate system

When defining any structure, it is important to input the joint coordinates values that specify the location of each joint. So it is necessary to use a coordinate system to locate the structure elements and to define the direction of loads, restraints, displacements, reactions, and the resultant forces and stresses.

#### I- Global coordinate system

It is a coordinate system for all over the structure. The following input data is prepared with respect to the global coordinate system:

- 1- Joint coordinate.
- 2- Joint restraints.
- 3- Joint spring supports.
- 4- Joint loading.
- 5- Joint constraints.

The following output data is referred to the global coordinate system:

- 1- Joint displacements.
- 2- Joint reactions.

#### II- Local coordinate system

It is a coordinate system for each element in the structure. The following input data is prepared with respect to the local coordinate system:

- 1- Element section properties.
- 2- Element loading.

The following output data is referred to the local coordinate system:

- 1- Element forces.
- 2- Element stresses.

## **Types of elements**

It is necessary to know the different types of elements used for defining the structure problem such as beams, frames, trusses, slabs, tanks, and rafts.

### **Frame element**

Two joints define this element. It can be used for modeling beams, trusses, and frames.

### **Plate bending element**

At least three joints are needed to define these element max four joints. It can be used for modeling slabs (sections under simple bending).

### **Membrane element**

At least three joints are needed to define these element max four joints. It can be used for modeling domes and folded plates (sections under normal only).

### **Shell element**

At least three joints are needed to define these element max four joints. It can be used for modeling tanks (sections under moment and normal).

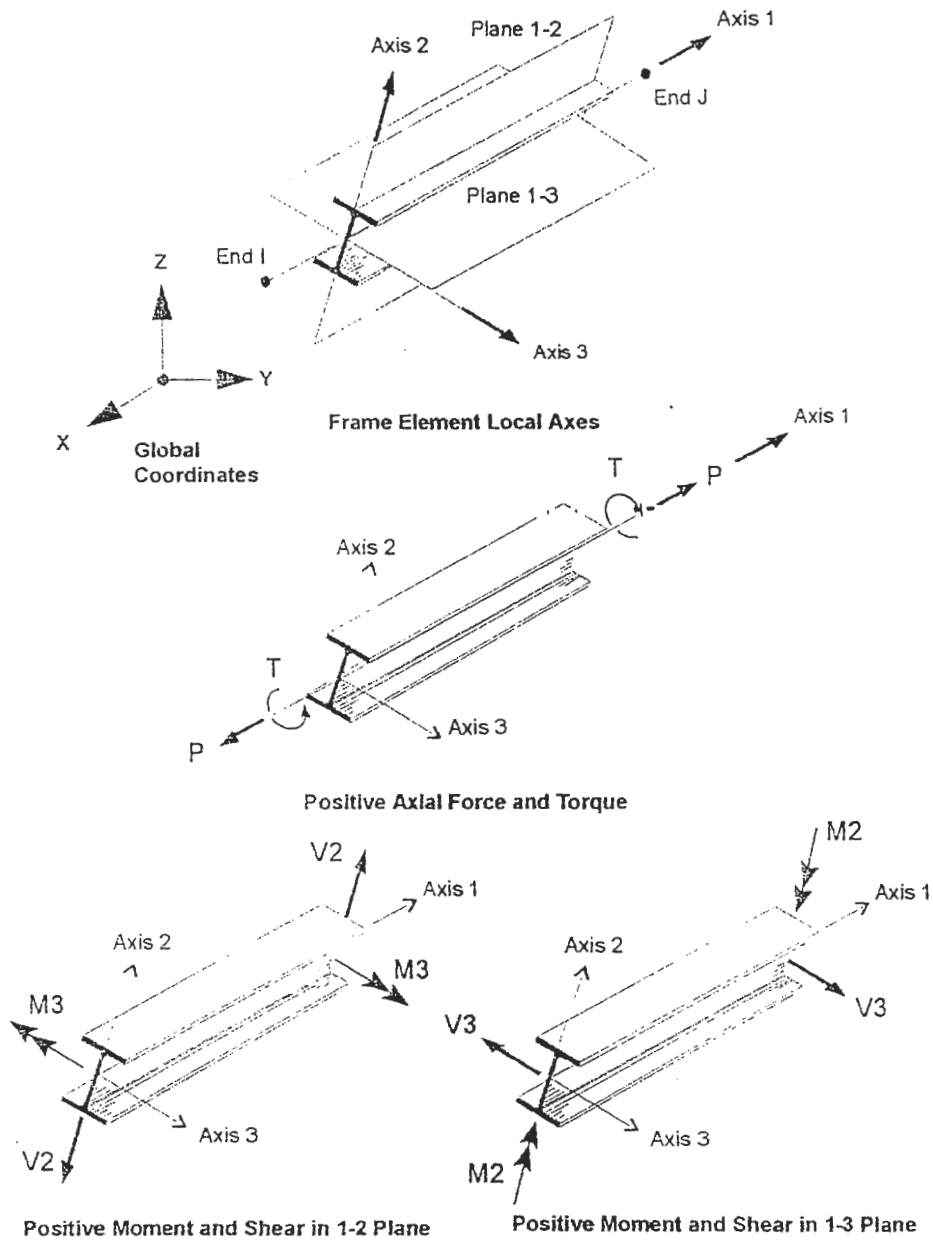
## **Restraints and degrees of freedom**

It is known that every point in space have six components of freedom, which are three displacements in the three directions  $U_x$ ,  $U_y$ , and  $U_z$  and three rotations about the three axes  $R_x$ ,  $R_y$ , and  $R_z$ .

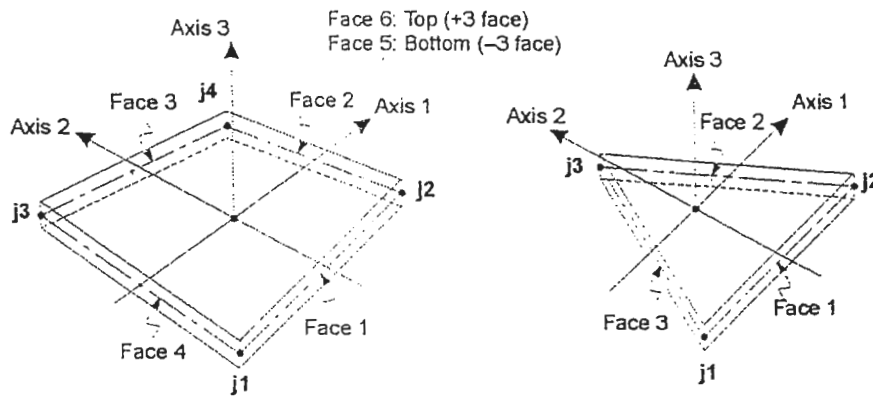
Existing of any displacement in any direction is called a degree of freedom (D.O.F) and inability of displacement in a specified direction means that this freedom is restrained.

**Frame Element**

FRAME element follows the designer's convention.

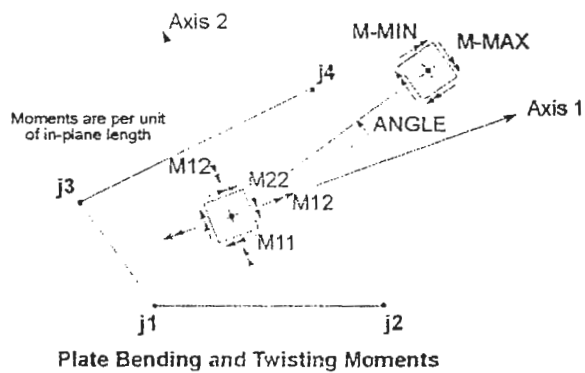
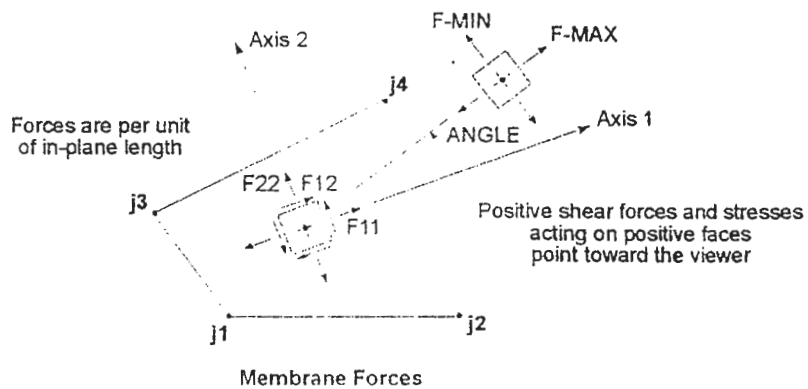
**Frame Element Internal Forces**

## Shell Element



Four-Node Quadrilateral Shell Element

Three-Node Triangular Shell Element



## Shell Internal Forces

FINT = F intermediate

$$FVM = \sqrt{\frac{1}{2}[(FMAX - FINT)^2 + (FMAX - FMIN)^2 + (FINT - FMIN)^2]}$$

Also See:



## PROGRAM APPLICATIONS

### Define Materials

1- On the Define menu, click Materials... This will display the Materials dialog box with default materials, CONC, OTHER and STEEL listed in the Materials list box.

What do you want to do?

Add a New Concrete Material Type

Add a New Steel Material Type

Add a New Other Material Type

### Add a New Concrete Material Type

To Add a New Concrete Material :

1- In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.

2 - In the Material Property Data box

- Choose Concrete from the Design Type drop down list.
- Enter the Material name in the Material Name edit box or accept the default name.
- In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume, Modulus of Elasticity, Poisson ' s Ratio and Co-efficient of thermal expansion, if the default values are not acceptable.
- In the Design Property area, enter the value for Reinforcing yield stress ( $f_y$ ), Concrete strength ( $f_c$ ), Shear steel yield stress ( $f_{ys}$ ) and Concrete shear strength ( $f_{cs}$ ) or accept the default values.
- Click OK.

- 3 - The new Material Name will be added to the Materials list box.
- 4 - To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.
- 5 - Click OK.

### **Add a New Steel Material Type**

To Add a New Steel Material:

- 1- In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.
- 2- In the Material Property Data box
  - Choose Steel from the Design Type drop down list.
  - Enter the Material name in the Material Name edit box or accept the default name.
  - In the Analysis Property Data area, enter new values for mass per unit volume, Weight per unit volume, Modulus of Elasticity, Poisson's Ratio and Co-efficient of thermal expansion, if the default values are not acceptable.
  - In the Design Property area, enter the value for Steel yield stress  $f_y$ , or accept the default value.
  - Click OK.
- 3- The new Material Name will be added to the Materials list box.
- 4- To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.
- 5 Click OK

### **Define Static Load Cases**

- 1- On the Define menu, click Static Load Cases... This will display the Static Load Case Names dialog box with a default Load Case, LOAD1 listed in the Loads list box.
- 2- To add a new Load Case name
  - Type in the new name in the Load edit box
  - Choose the load type to be Dead, Live, Quake, Wind, Snow or Other from the Type drop down list
  - Type in a self-weight multiplier in the edit box and
  - Click Add new Load button.
- 3- To change a Load Case name click on the name in the list box and then type over the changes in the edit boxes or choose a different Load Type from the drop down list and click Change Load button.
- 4- To delete a Load Case name click on the name in the list box and click Delete Load button.
- 5- Click OK.

**Tip:** Load Case names defined here are used in Load Combinations. Load Case Types are used by SAP2000 to develop automatic load combinations for Design/Stress Check of the structure.

### **Define Load Combinations**

- 1 On the Define menu, click Load Combinations... This will display the Load Combinations dialog box.
- 2 to add a new Load Combination click Add new Combo button. This will display the Load Combination Data dialog box.

In the Load Combination Data dialog box

- Type in the combination name in the Load Combination Name edit box.
- Select the Load Combination Type: ADD, ENVE, ABS, SRSS
- Type in the title (description of load combo) in the Title edit box.
- Select Analysis Case names from the Case Name drop down list box and type in the multiplier in the Scale Factor edit box and click ADD. This will add the Analysis Case and the multiplier to the list box.
- Similarly select other Analysis Cases and add them to the list box to complete the Load Combination definition.
- It is possible to modify or delete an Analysis Case from the Combination definition by selecting the Analysis Case from the list box and clicking the Modify or Delete button.
- Select whether the combo should be used as a part of the Steel or Concrete design loading combinations.
- Click OK. This will add the Load Combination to the Combinations list box.

3- To include the default design load combinations for the code specified in Concrete and Steel Preferences, click on the Add Default Design Combo Button. The program will add Steel design combos if the Steel Design is selected in the Design menu and Concrete design combos if the Concrete Design is selected in the Design menu. The Type assignment for the Static Load Case is used to determine the appropriate load factors.

4 - To modify a Load Combination, click on the name in the Combinations list box to select it and then click Modify/Show Combo button. This will display the Load Combination Data dialog box. Make the appropriate changes and then click OK.

5- To delete a Load Combination, click on the name in the Combinations list box to select it and then click Delete Combo button. This will delete the Load Combination.

6- Click OK.

**Tip:** Load Combinations defined here are used in Select Design Combos to define design-loading combinations.

### Selecting Design Load Combinations

1- On the Design Menu, click Select Design Combos ... This will display the Design Load Combinations Selection dialog box.

2- In the Design Load Combinations Selection box:

- Select previously defined Combos from the List of Combos
- Click on Add button to add Combos to the Design Combos list
- Clicking on Remove button will remove the selected Combos from the Design Combos list
- Click OK

**Note:** The Design Load Combinations Selection menu lists a series of default load combinations labeled DCON# for Concrete design and DSTL# for Steel design. The number and type of default combinations depends upon the static Load Cases previously defined. Other user defined Load Combos that were

indicated as design combos in the Load Combination Data dialog box will also be included in the Design Combos

If other load combinations have been defined and were not indicated as design combos in the Load Combination Data dialog box, they are shown under List of Combos. The Design Combos list can be modified by removing one or more default Combinations and/or by adding from the List of Combos.

Clicking on the Show button will present the Load Cases and Scale factors for the highlighted Combo.

## FRAME SECTION

Add a Frame Section by defining the dimensions and or properties manually

- 1- On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
- 2 - In the Frame Sections dialog box click on the Add drop down button and choose Add I/Wide Flange or Channel, Tee, Angle etc. by double clicking on one of the section types. This will display a Section Property form that shows the physical shape and dimensions of the section.
- 3- Enter the section name or accept the default name.
- 4 - Enter the physical dimensions of the section or accept the default values.
- 5- Choose the material type (e.g. Steel, Concrete or Other) from the Material drop down list box. For Concrete sections see below.
- 6- It is possible to look at the section properties by clicking on Section Properties.

7 - You can modify the section properties by pressing the Modification Factors button. There you can change the default factor of 1 that the section properties are multiplied by.

8 - Click OK to accept the selected section.

9 - The new section name will be added to the Name list box.

10 - To delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.

11- Click OK.

### Concrete Section

For concrete sections you will need to also perform the following steps:

1- When you select a Concrete material type, an additional Reinforcement button will appear at the bottom of the form. Click the Reinforcement button.

- In the Reinforcement Data dialog box, select Element Class e.g. Column or Beam.

### For Columns

- Select the Configuration of Reinforcement e.g. Rectangular or Circular.
- Enter the Concrete Cover in the text edit box.
- Enter the Number of Bars in the 3-Direction in the text edit box.
- Enter the Number of Bars in the 2-Direction in the text edit box.

Example of Concrete Column 3 bars in the 3-Direction 2 bars in the 2-Direction

- If Circular Reinforcement Configuration is selected then enter total number of bars.
- Select and Enter the Area of 1 Bar or



- Select Design Area of Steel for SAP2000 to automatically find the area of steel required during design mode.

### **For Beams**

- Enter the Top and Bottom Concrete Cover in the text edit boxes.
- If you want to specify top and bottom steel, enter reinforcement area for the section in the appropriate text edit box. Otherwise leave values of zero for SAP2000 to calculate the reinforcement automatically.
- Click OK to return to the Section Definition form.

### **Draw a Frame Element from Joint to Joint**

#### **Draw in 2-D**

- 1- On the View menu click Show Grids. This puts a check mark next to it and works as a toggle to turn on or Off the Grids.
- 2- Select 2-D Plane in which you want to draw the member.
- 3 - Click on to switch to drawing a FRAME element between 2 points.
- 4- Click on a Grid intersection, a previously defined Joint or any point in the plane. Click again on another point to add a single FRAME element. Every subsequent click will add another FRAME element unless a double click on the same joint is performed or the ENTER key is pressed. Pressing the ESC key will also terminate the sequential draw and take you out of the Draw mode.

#### **Draw in 3-D**

- 1 -On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
- 2 - Click on to switch to drawing a FRAME element between 2 points.



3 - In the 3-D view click on a Grid intersection or a previously defined Joint ( a small red ball appears to confirm that the joint has been selected ).

4- Click on another Grid intersection or a Joint to add a FRAME element. Every subsequent click on a selected point will add another FRAME element unless a double click on the same joint is performed or the ENTER key is pressed. Pressing the ESC key will also terminate the sequential draw and take you out of the Draw mode.

**Note:** It is not possible to draw a FRAME element between any two arbitrary points in space in the 3-D view.

## LOCAL AXIS

### Assign Local Axes for Frame Elements

1- Select one or more Frame Elements to which you want to assign the same Local Axes.

2- On the Assign menu clicks Frame and then Local Axes... in the sub-menu.

3- In the Frame Local Axis display box:

- Type in a value for Angle in Degrees. This is an angle by which local axis (2) of the Element will be rotated around local axis 1. Local axis 1 is along the length of the Element. By default local axis 2 is always in the 1-Z plane except if the Element is vertical and then it is parallel to the global X-axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 1 is pointing towards you..
- Check the Reverse start and end connectivity box if you want to flip the I and J ends of the members i.e. flip the local axis 1.
- Click the OK button

**Note:** By default global Z is taken as the 3D View Up Direction. You may change the UP direction from the Option pull down menu. If you change the UP direction, it will only affect the display of the model and does not affect the local axes definition in any way.

### **Assign Gravity Loads to Frames**

This loading method is a way to add the factored self-weight of the members in as a force in any of the global directions. It is recommended that the actual self-weight of the structure be included in the definition of the Static Load Cases.

- 1- Select one or more FRAMES to assign loads to.
- 2- On the Assign menu click Frame Static Loads and then Gravity... in the sub-menu. This will display the Frame Gravity Loads dialog box.
- 3 - In the Frame Gravity Loads dialog box:
  - Select the Load Case
  - Provide Gravity Multipliers
  - Select from Add, Replace, Delete
  - Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

### **Assign Point and Uniform Loads to Frames**

- 1- Select one or more FRAMES to assign loads to.
- 2- On the Assign menu, click Frame Static Loads and then Point and Uniform... in the sub-menu. This will display the Point and Uniform Span Loads dialog box.
- 3- In the Point and Uniform Span Loads dialog box:

- Select Load Case
- Choose Load Type and Direction
- Provide Point Loads and Distances
- Provide Uniform Load
- Select from Add, Replace, Delete

4- Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

### **Assign Trapezoidal Loads to Frames**

1- Select one or more FRAMES to assign loads to.

2- On the Assign menu click Frame Static Loads and then Trapezoidal... in the sub-menu. This will display the Trapezoidal Span Loads dialog box.

3- In the Trapezoidal Span Loads dialog box:

- Select Load Case
- Choose Load Type and Direction
- Provide Loads and Distances
- Select from Add, Replace, Delete

4- Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

**Assign Temperature Loads to Frames**

- 1- Select one or more FRAMES to assign loads to.
- 2- On the Assign menu, click Frame Static Loads and then Temperature... in the sub-menu. This will display the Frame Temperature Loading dialog box.
- 3- In the Frame Temperature Loading dialog box:
  - Select Load Case
  - Select Type from Temperature, Temperature Gradient 2-2, Temperature Gradient 3-3
  - Select Temperature by Element or Joint Pattern
  - Select from Add, Replace, Delete
- 4 - Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

**Assign Prestress to Frame Static Load**

Once the prestress has been assigned to the frame member the load from that prestress needs to be assigned to a Load Case for SAP2000 to be able to include it in the analysis.

- 1- Select the FRAME Elements for which you want the prestress load assigned to a defined Load Case.
- 2 - On the Assign menu, click Frame Static Loads and then Prestress ... in the sub-menu. This will display the Frame Prestress Loads dialog box.
  - Select the Load Case from the drop down list box.
  - Set a scale factor for the prestress load. This value is independent of scale factor for Load Combinations.
  - Select Add, Replace, and Delete existing load.
- 3- Click OK

### Assign Frame Prestress

Assigning Frame Prestress loads is a way to model the prestress cable load on a frame member.

- 1- Select the FRAME Elements to which you want to assign a Prestress load.
- 2- On the Assign menu, click Frame and then prestress... in the sub-menu. This will display the Frame Prestressing Patterns dialog box.
- 3- In the Frame Prestressing Patterns dialog box enter:
  - The Cable Tension in the text edit box
  - The Start, Middle and End cable locations.
  - Select from Add, Replace or Delete.
- 4- Click OK.

**Note:** You can assign more than one Prestress load to an individual frame member. Once you have assigned prestress loads to elements you will need to assign the frame prestress loads to static load case.

### Assign Frame Releases

Releases are assigned to selected Frame Elements.

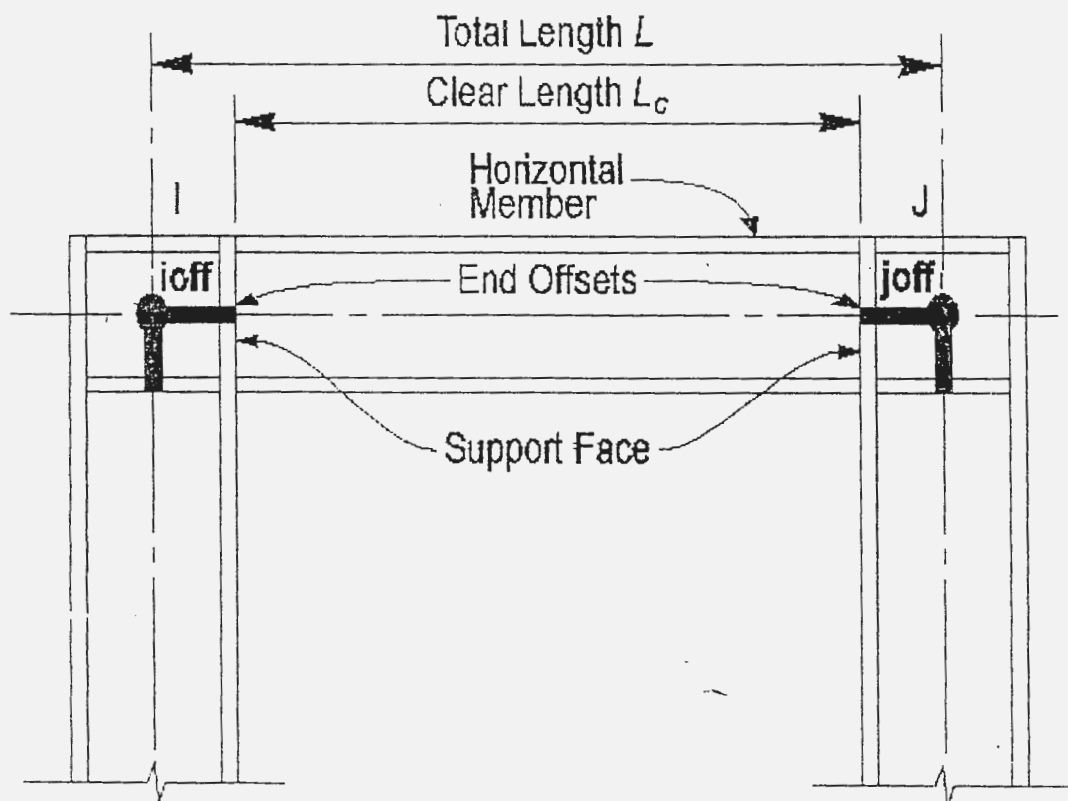
- 1- Select the FRAME Elements to which you want to apply the releases.
- 2- On the Assign menu click Frame and then releases... in the sub-menu. This will display the Frame Releases dialog box.
- 3- In the Frame Releases dialog box:
  - For each end of the FRAME Element check the type of release desired (Axial, Shear Force 2 (Major), Shear Force 3 (Minor), Torsion, Moment 22 (Minor) and Moment 33 (Major).
  - If no releases are desired then check the No Releases box.
- 4- Click OK.

**Note:** The releases specified always replace the existing releases.

### Assign Frame End Offsets

Frame End Offsets are assigned to selected Frame Elements.

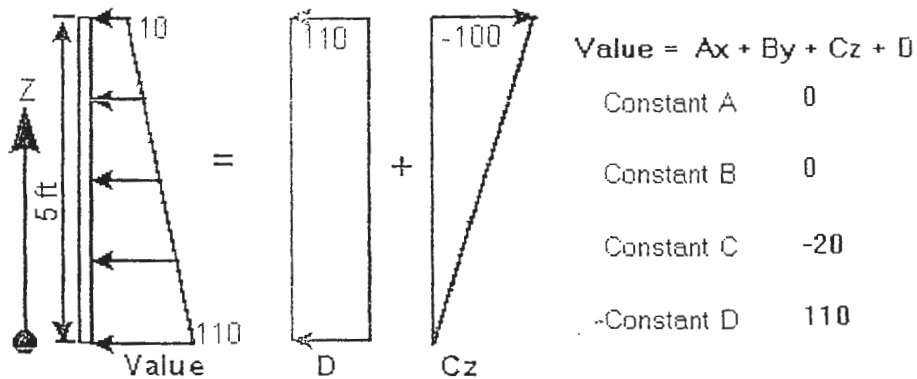
- 1- Select the FRAME Elements to which you want to apply the Frame End Offsets.
- 2- On the Assign menu, click Frame and then End offsets ... in the sub-menu. This will display the Frame End Offsets dialog box.
- 3- In the Frame End Offsets dialog box:
  - If you want to use User Specified End Offsets then check Define Lengths and type in the values for the Offsets at End I and End J.
  - If you want the program to calculate the End offsets from the connectivity of the model then check Update lengths from current connectivity. The program will automatically calculate the End offsets from the Depth (Major) and Width (Minor) specified for the Frame Element properties.
  - Specify a Rigid Zone Factor in the edit box. This is a factor used to define the percentage of the Zone specified through End Offsets to be taken as fully rigid. 0 means no rigid zone and 1 means that the entire zone is taken as rigid.



### Define Joint Patterns

- 1- On the Define menu, click Joint Patterns... This will display the Pattern Names dialog box with a default pattern, DEFAULT listed in the Patterns list box.
- 2- To add a new pattern name type in the new name in the Patterns edit box and click Add new Pattern Name.
- 3- To change a pattern name click on the name in the list box and then type over the changes in the edit box and click Change Pattern Name.
- 4- To delete a pattern name click on the name in the list box and click Delete Pattern Name.
- 5- Click OK.

**Tip:** Pattern names defined here are used in Assign Joint Patterns.



### Assign Joint Patterns

- 1- Select the Joints to which you want to assign the pattern.
- 2- On the Assign menu, click Joint Patterns... This will display the Pattern Data dialog box.
- 3- Click on the Pattern Name drop downs box and choose a previously defined pattern name by clicking on it.
- 4- Type in the values in the edit boxes for Constants A, B, C and D, relevant to defining the Value of  $Ax+By+Cz+D$  (x, y and z are joint coordinates relative to the origin of the current coordinate system). It is this Value that will be assigned to the Joints. It can define the variation of Temperature for Frame Elements and variation of Pressure for Shell Elements. These Values are then subsequently multiplied by the Temperature values assigned from the Frame Static Loads or Pressure values assigned from the Shell Static Loads. For instance if you want to assign a pattern which represents the variation of soil pressure on a wall:
- 5 - Options
  - To add computed Values from Constants to existing Values from previously assigned patterns, click on Add to existing values under Options.
  - To replace computed Values from Constants to existing Values from previously assigned patterns, click on Replace existing values under Options.
  - To delete previously assigned patterns, click on Delete existing values under Options.
- 6 - Depending on the pattern you are trying to define select either:
  - Use all values if all values are desired
  - Zero Negatives Values if negative variation is not desired.
  - Zero Positive Values if positive variation is not desired.
- 7- Click OK.



**Note:** Once you have assigned the pattern you will still need to assign a load to that pattern by selecting from the assign menu, Shell Static Loads... Pressure....

## **SHELL SECTION**

Add a Shell Section by defining the dimensions and / or properties

- 1- On the Define menu, click Shell Sections... This will display the Shell Sections dialog box.
- 2- In the Shell Sections dialog box click on the Add New Section button. This will display Shell Sections Data dialog box.
- 3- In the Shell Sections Data dialog box
  - Enter the Section Name or accept the default name.
  - Select the material type by clicking on the Material drop down box.
  - Enter the Membrane and Bending Thickness.
  - Choose the shell type to be Shell, Membrane or Plate.
  - Click OK.
- 4- The new section name will be added to the Shell Sections list box.
- 5- To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.
- 6- Click OK.

## **Add a Shell Section by defining the dimensions and / or properties**

- 1- On the Define menu, click Shell Sections... This will display the Shell Sections dialog box.
- 2- In the shell Sections dialog box click on the Add New Section button. This will display Shell Sections Data dialog box.
- 3- In the Shell Sections Data dialog box

- Enter the Section Name or accept the default name.
- Select the material type by clicking on the Material drop down box.
- Enter the Membrane and Bending Thickness.
- Choose the shell type to be Shell, Membrane or Plate.
- Click OK.

4- The new section name will be added to the Shell Sections list box.

5- To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.

6- Click OK.

### **Draw a Shell Element between 4 Joints**

#### **Draw in 2-D**

1- On the View menu click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.

2- Select 2-D Plane in which you want to draw the member.

3- Click on to switch to drawing a SHELL element between 4 Joints.

4- Click on a Grid intersection, a previously defined Joint or any point in that plane.

5- Click again on 3 other points either clockwise or counterclockwise to add a SHELL element.

**Note:** If you want to add a triangular element the fourth point clicked should be the same as the first.

#### **Draw in 3-D**

1- On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.

- 2- Click on to switch to drawing a SHELL element between 4 Joints.
- 3- In the 3-D view click on a Grid intersection or a previously defined Joint (a small red ball appears to confirm that the point has been selected).
- 4- Click on 3 other similar points either clockwise or counterclockwise to add a SHELL element.

**Note:** It is not possible to draw a SHELL element between any 4 arbitrary points in space in the 3-D view.

### **Assign Local Axes for Shell Elements**

- 1- Select one or more Shell Elements to which you want to Assign the same Local Axes.
- 2- On the Assign menu, click Shell and then Local Axes... in the sub-menu.
- 3- In the Shell Local Axis display box:
  - Type in a value for Angle in Degrees. This is an angle by which local axis (2) of the Element will be rotated around local axis 3. Local axis 3 is normal to the surface of the SHELL Element. By default local axis 2 is always in the 3-Z plane (and lies in the plane of the SHELL) except if the Element is horizontal and then it is parallel to the global Y-axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 3 is pointing towards you.
  - Check the Reverse direction of normal box if you want to flip the direction of local axis 3.
  - Click the OK button

**Note:** The positive direction of the local axis 3 is determined by the definition of the SHELL connectivity. See Sign Convention for more info.

### **Assign Gravity Loads to Shells**

This loading method is a way to add the factored self-weight of the members in as a force in any of the global directions. It is recommended that the actual self-weight of the structure be included in the definition of the Static Load Cases.

- 1- Select one or more SHELLS to assign loads to.
- 2- On the Assign menu, click Shell Static Loads and then Gravity... in the sub-menu. This will display the Shell Gravity Loads dialog box.
- 3- In the Shell Gravity Loads dialog box:
  - Select the Load Case
  - Provide Gravity Multipliers
  - Select from Add, Replace, Delete
- 4- Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

### **Assign Uniform Loads to Shells**

- 1- Select one or more SHELLS to assign loads to.
- 2- On the Assign menu, click Shell Static Loads and then Uniform... in the sub-menu. This will display the Shell Uniform Loads dialog box.
- 3- In the Shell Uniform Loads dialog box:
  - Select Load Case
  - Provide Uniform Load value and Direction
  - Select from Add, Replace, Delete
- 4- Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

### **Assign Pressure Loads to Shells**

- 1- Select one or more SHELLS to assign loads to.
- 2- On the Assign menu, click Shell Static Loads and then Pressure... in the sub-menu. This will display the Shell Pressure Loads dialog box.
- 3- In the Shell Pressure Loads dialog box:
  - Select Load Case
  - Choose if pressure will be applied by Element or by Joint Pattern
  - If by Element, then specify pressure value
  - If by Joint Pattern, then choose the pattern and multiplier
  - Select from Add, Replace, Delete
- 4 - Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

### **Assign Temperature Loads to Shells**

- 1- Select one or more SHELLS to assign loads to.
- 2- On the Assign menu, click Shell Static Loads and then Temperature... in the sub-menu. This will display the Shell Temperature Loading dialog box.
- 3- In the Shell Temperature Loading dialog box:
  - Select Load Case
  - Choose the type of Temperature Loading, i.e. Temperature or Gradient
  - Choose if temperature to be applied by Element or by Joint Pattern
  - If by Element, then specify temperature value
  - If by Joint Pattern, then choose pattern and multiplier
  - Select from Add, Replace, Delete
- 4- Click OK.

## **Reshape Element**

Reshape Element option is used to edit element forms or shapes. With the help of this option it is possible to move FRAME or SHELL elements, stretch or shrink FRAME elements and reshape SHELL elements. To move or reshape an element do the following:

### **Add in 2-D**

- 1- On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
- 2- Select 2-D Plane in which you want to reshape the member.
- 3- Click on to switch to Reshape Element mode.
- 4- Click once on a FRAME or a SHELL element to select it. This will display the member end handles.
- 5- Grabbing one of the handles of the element and moving it will stretch, shrink, rotate or reshape the element.
- 6- Grabbing the element (any point away from the handles) and dropping it at another location will allow you to move the element.

### **Add in 3-D**

In 3-D, the element editing works the same way as 2-D except that the handle may only be dropped at another pre-defined JOINT or Grid intersection.

**Note:** Only one Element at a time may be moved or reshaped through this option.

### **Add Special Joints**

In building a model in SAP2000 it is not necessary to pre-define Joints. The Joints are automatically added to the ends or corners of the elements. Special

Joints are those Joints, which are added by the user. These Joints may be necessary to add in rare cases such as at one end of an NLLink Element - an end where there is no other element present and hence no automatically generated Joint. To add a Special Joint do the following:

### **Add in 2-D**

- 1- On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
- 2- Select 2-D Plane in which you want to add a Joint.
- 3- Click on to switch to Add Special Joint mode.
- 4- Click on a Grid intersection or any other point in that plane to add a Joint.

### **Add in 3-D**

- 1- On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
- 2- Click on to switch to Add Special Joint mode.
- 3- In the 3-D view, click on a Grid intersection to add a Joint.

**Note:** The Joints may only be added at grid intersections in 3-D view.

Right Click on a Joint will open Joint Information edit box. You may edit the Joint location in this edit box to locate the Joint precisely in the desired location.

### **Edit Grids**

After defining your coordinate system you can edit the location of the individual grids in the active coordinate system.

- 1- On the Draw menu, click Edit Grid or double click on a grid line in the graphics screen.

2- In the Modify Grids dialog box, Select Direction of the grids you want to edit.

3-In the location area:

#### **To Move a Grid**

- Select the grid location from the list.
- Edit the grid 's location in the text edit box.
- Click the Move Grid Line button.

#### **To Add a new Grid**

- Enter the grid 's location in the text edit box.
- Click the Add Grid Line button.

#### **To Delete a Grid**

- Select the grid location from the list.
- Click the Delete Grid Line button.

4- The Delete All button will delete all the grids in the coordinate system.

5- Check the Lock Grid Lines if you want to lock the grid lines from being moved using the method discussed in the note below.

6- Check the Snap to Grid lines if you want the feature on.

7- Check Glue Joints to Grid Lines if you want the joints to move with the grids. This is a very powerful way to quickly edit the structure without having to redefine joint locations.

8- Click OK when finished.

**Note:** From the display screen with the Reshape Element option set, you can add a new grid line by holding the CTRL-key down and clicking on an existing grid line and dragging away a duplicate.



### **Assign Joint Springs**

Springs are added to selected joints.

1- Select the Joints to which you want to apply springs.

2- On the Assign menu, click Joint and then Springs... in the sub-menu.

This will display the Joint Springs dialog box.

3- In the Joint Springs dialog box:

- Type in the Spring Stiffness values in the local directions for the three Translations and the three Rotations.
- In the Options area select the appropriate option - Add to existing springs, Replace existing springs or Delete existing springs.
- If required specify the upper half of the Coupled 6X6 spring matrix, accessed by clicking the advanced button.

4 Click OK.

### **Equal Constraint**

An Equal Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the constraint local coordinate system. The other degrees of freedom are unaffected.

The Equal Constraint differs from the rigid-body types of Constraints in that there is no coupling between the rotations and the translations. This Constraint can be used to:

- Model symmetry and anti-symmetry conditions with respect to a plane
- Partially connect together different parts of the structural model, such as at expansion joints and hinges

- For fully connecting meshes, it is better to use the Body Constraint when the constrained joints are not in exactly the same location.

### **Joint Connectivity**

Each Equal Constraint connects a set of two or more joints together. For a given Constraint, if any of the selected degrees of freedom are negative (i.e., opposite), only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space, but for best results all joints should share the same location in space if used for connecting meshes. Otherwise, moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Such restrained moments may also be generated when Equal Constraints are used for symmetry purposes. They are necessary to enforce the desired symmetry or anti-symmetry of the displacements when the applied loads are not correspondingly symmetric or anti-symmetric.

### **Local Coordinate System**

Each Equal Constraint uses a fixed coordinate system, `csys`, that you specify. The default for `csys` is zero, indicating the global coordinate system. The axes of the fixed coordinate system are denoted X, Y, and Z.

### **Selected Degrees of Freedom**

For each Equal Constraint you may specify a list, `cdofs`, of up to six degrees of freedom in coordinate system `csys` that are to be constrained. The degrees of freedom are indicated as UX, UY, UZ, RX, RY, and RZ.

A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., -UX.

### **Add Joints to an existing Constraint**

- 1- Select the Joints you want to add to an existing Constraint.
- 2- On the Assign menu, click Joint and then Constraints... in the sub-menu. This will display the Constraints dialog box.
- 3- In the Constraints dialog box:
  - Choose the existing Constraint from the Constraints list box
  - Click OK
- 4 The Joints will be added to the existing Constraint.

### **Assign Joint Constraints**

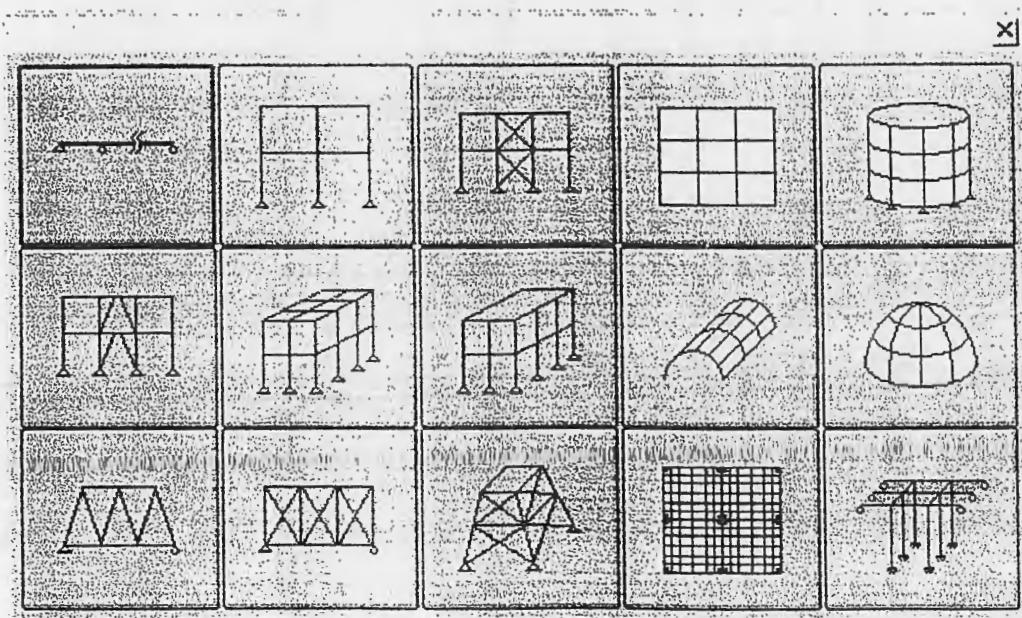
Constraints are applied to selected joints.

- 1- Select the Joints to which you want to apply Constraints.
- 2- On the Assign menu, click Joint and then Constraints... in the sub-menu. This will display the Constraints dialog box.
- 3- In the Constraints dialog box:
  - Either select default names for different types of Constraints such as Beam, Body, Diaphragm, Equal, Local, Plate, Rod or Weld or
  - Add a new Constraint by clicking on the Add drop down list and selecting the appropriate type of constraint. Type in the new name or accept the default name, choose the axis (X, Y or Z) or Auto to define the direction of the constraint and then click OK.
- 4- Click OK.

### Getting a Head Start With Templates

New models may be created with very little effort using pre-programmed templates.

Click on any of the template buttons to learn more about them.




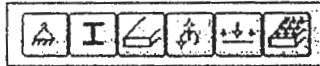
#### To Base a new model on a template:

- 1 On the File menu, click New Model from Template... This will display the possible templates shown above.
- 2 Click on a template most appropriate for your model.
- 3 Choose or change the default values in the edit boxes.
- 4 Click OK.

#### To add one or more templates to the existing model:

- 1 On the Edit menu, click Add to Model From Template... This will display the template options similar to the one shown above.
- 2 Click on the template you want.
- 3 Fill in the parameters in the edit boxes and then click on to specify an insertion point.
- 4 Click OK.

**Note:** Remember to save your model periodically. Click on 

**Assign Options**

Clicking on any of these buttons from the allows you to Assign properties, loads and restraints/constraints to your selection. Members or Joints have to be selected first in order to make an assignment.

Select various Assign options by clicking as follows:

**To Assign****Do this**

Supports or Restraints to Joints

Click on



Sections to Frame Elements

Click on



Thickness to Shell Elements

Click on



Loads on Joints

Click on



Span and Point Loads on Frame Elements

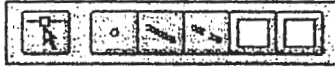
Click on



Uniform Loads on Shell Elements

Click on



**Draw Mode**

Clicking on any of these buttons from the  puts SAP2000 in the Draw mode.


The Draw mode allows for drawing of new members and editing one element or a joint at a time.

Draw Mode is also the default mode when a NEW model is started.

Select various Draw options by clicking as follows:

**To Draw or Edit****Do this**

Edit/Shape/Move Element

Click on  and click on a joint or Element. Use the handles to move or stretch.

An extra or Special Joint

Click on  and click on any point to add a Joint

A Frame Element (joint to joint)



Quick draw Frame Elements



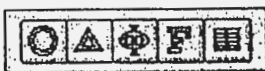
A Shell Element (joint to joint)



Quick draw Shell Elements



## Display Options



Clicking on any of these buttons from the

or some buttons from the Toolbar

allows you to Display your selection with varied options.  
Select various Display options by clicking as follows:

To DisplayDo this

Undeformed geometry

Click on



Isometric view (toggle)

Click on



2-D View (XY, XZ or YZ plane)

Click on any one of the buttons



to display an elevation view in that specific plane.

3-D View

Click on



Deformed Shape

Click on



or Display menu and Show Deformed Shape

Element Stress Diagram

Click on



or Display menu and Show Element Stress Diagram.

Right click will show the details.

Joint, Frame or Shell Loads

Click on Display menu and Show Loads.

Patterns

Click on Display menu and Show Patterns.

Tabular Input

Click on Display menu and Show Input Tables.

Mode Shapes

Click on



or Display menu and

Tabular Output

Click on



or Display menu and

Response Spectrum

Click on Display menu and

Time History

Click on Display menu and

Shrunk Elements (toggle)

Click on



or click on



and select shrink.

Selective Elements

Click on



or View menu and