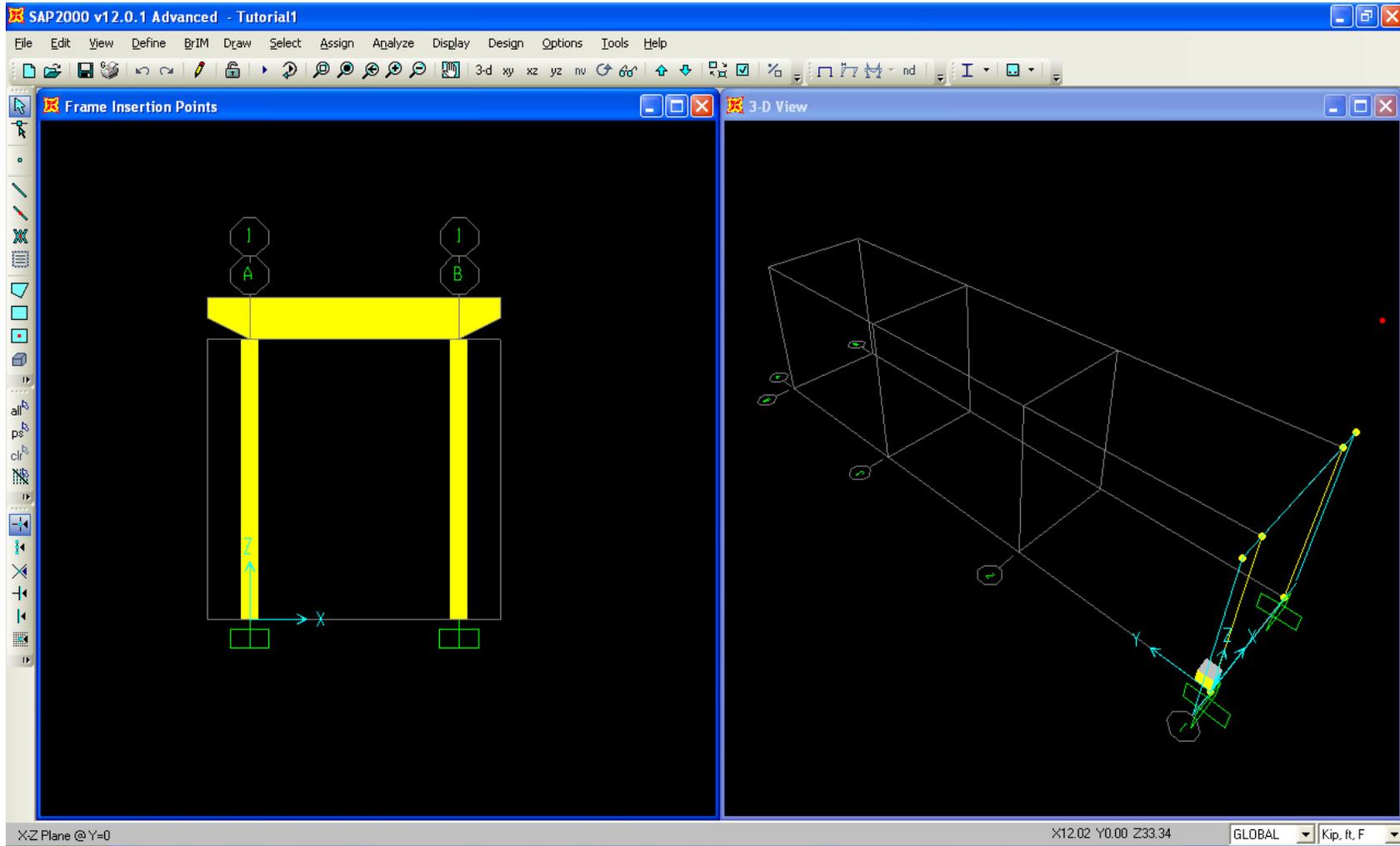
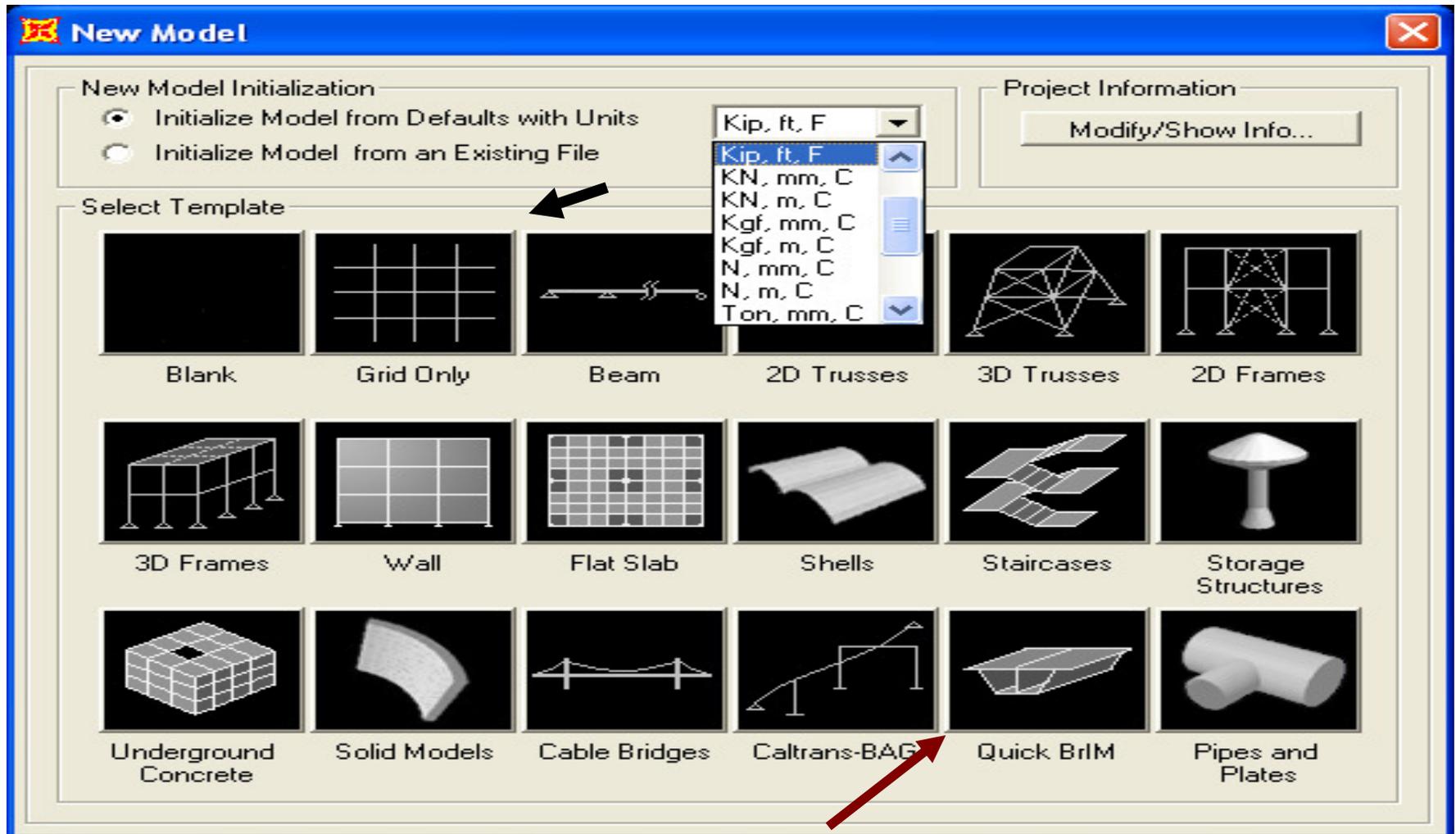


# SAP2000 Getting Started Tutorial

Learn how to define materials, sections, grids, and supports with basic modeling concepts for creating and modifying a concrete bent structure model using a nonprismatic cap beam



Click File>New menu or button/icon  to bring up the initial SAP2000 screen where you select units and choose whether you want to work with grids or one of the templates. For this tutorial, we will work with Grids, but you will probably want to explore the Quick BrIM options for modeling of simple bridges later on. Choose Kip, ft, F units and click 'Grid Only' option



This screen enables you to define equally spaced grids. Later you will learn how to modify and insert irregular spaced gridlines. Please type Number of Grid Lines and Grid Spacing as shown. Note how although we are in Kip-ft units, you can input with architectural units (180" below) at anytime and SAP2000 will automatically convert to current units after you tab.

**Quick Grid Lines**

Cartesian Cylindrical

Coordinate System Name  
GLOBAL

Number of Grid Lines

X direction	2
Y direction	4
Z direction	2

Grid Spacing

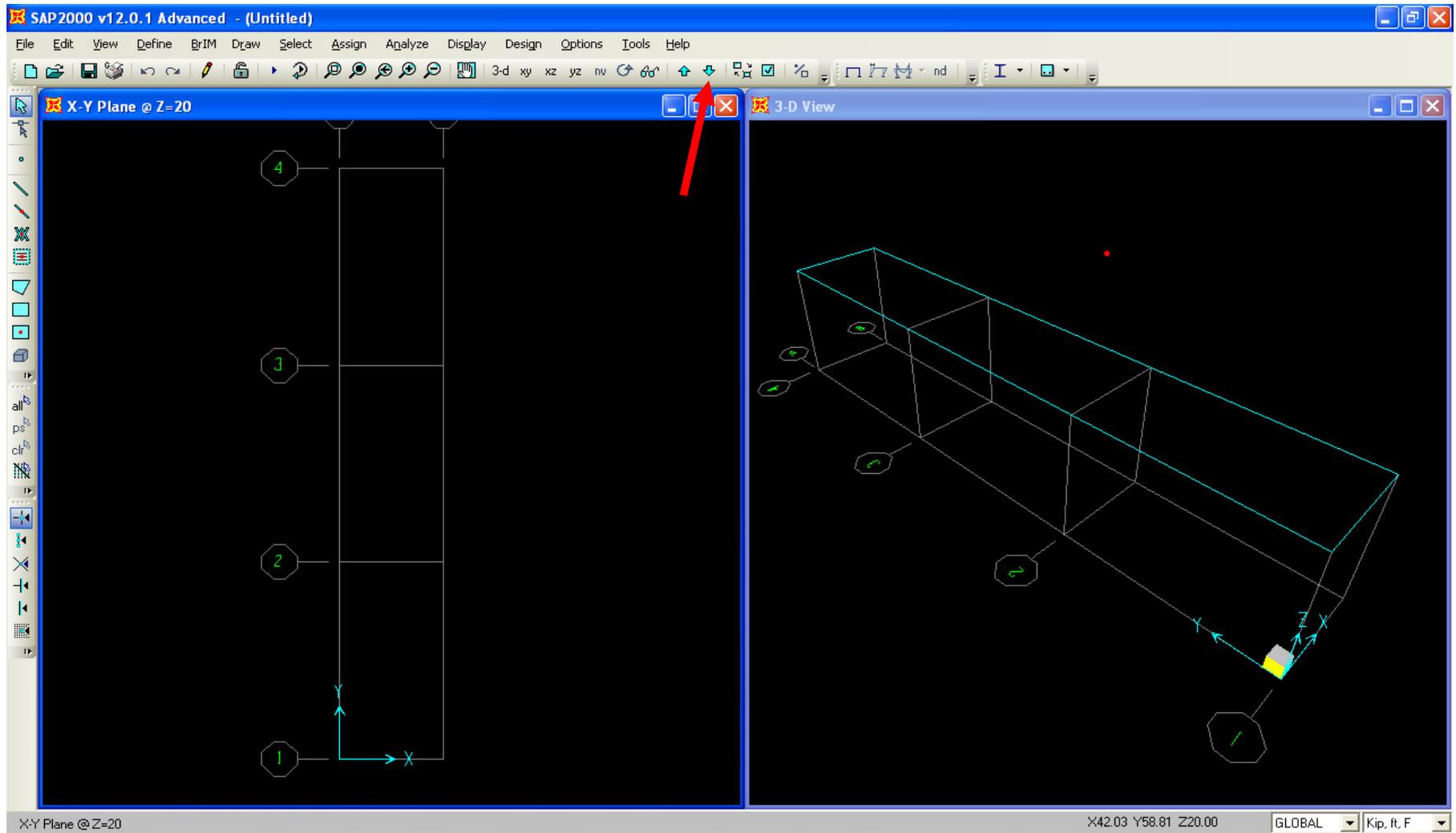
X direction	180"
Y direction	30
Z direction	20

First Grid Line Location

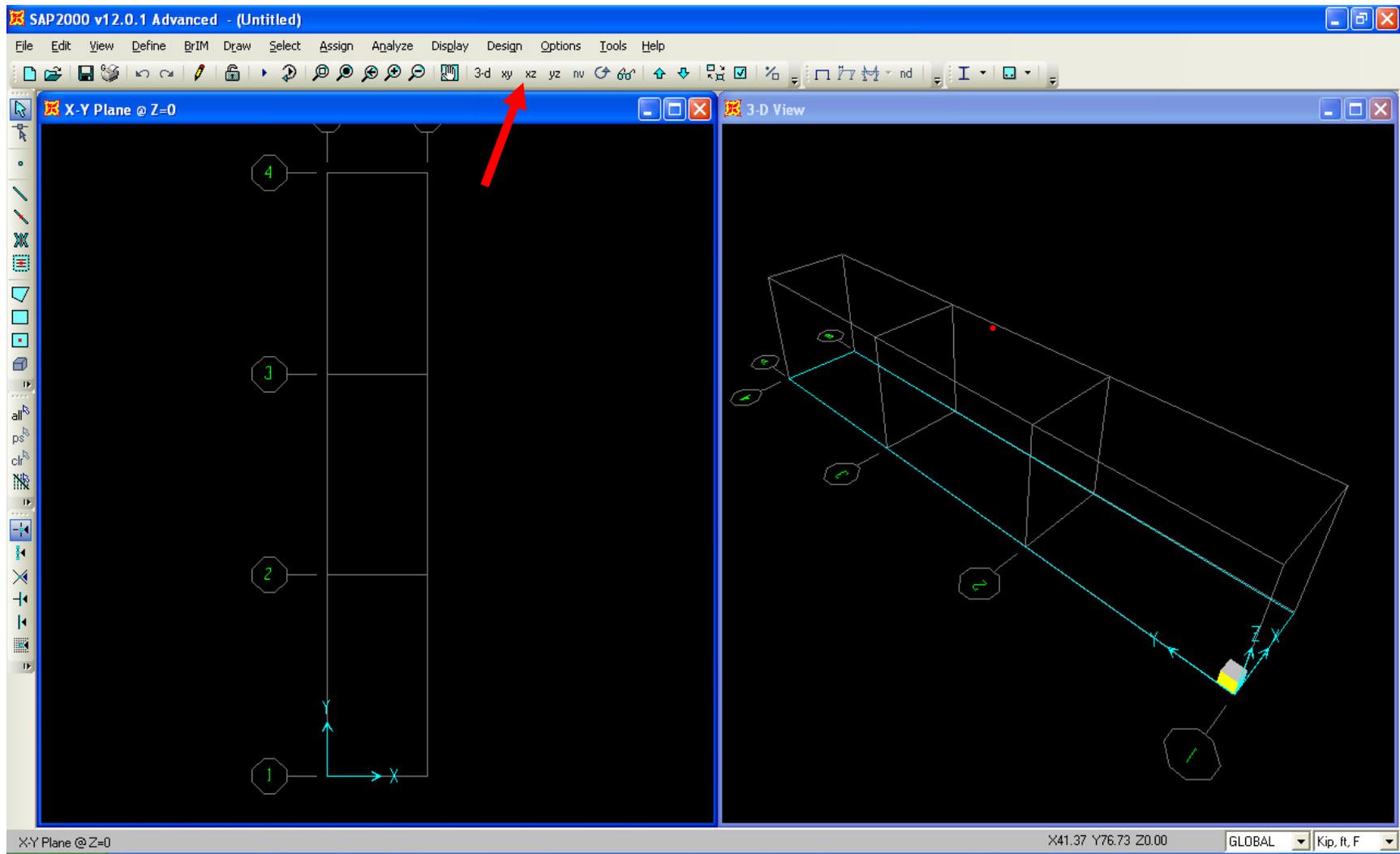
X direction	0.
Y direction	0.
Z direction	0.

OK Cancel

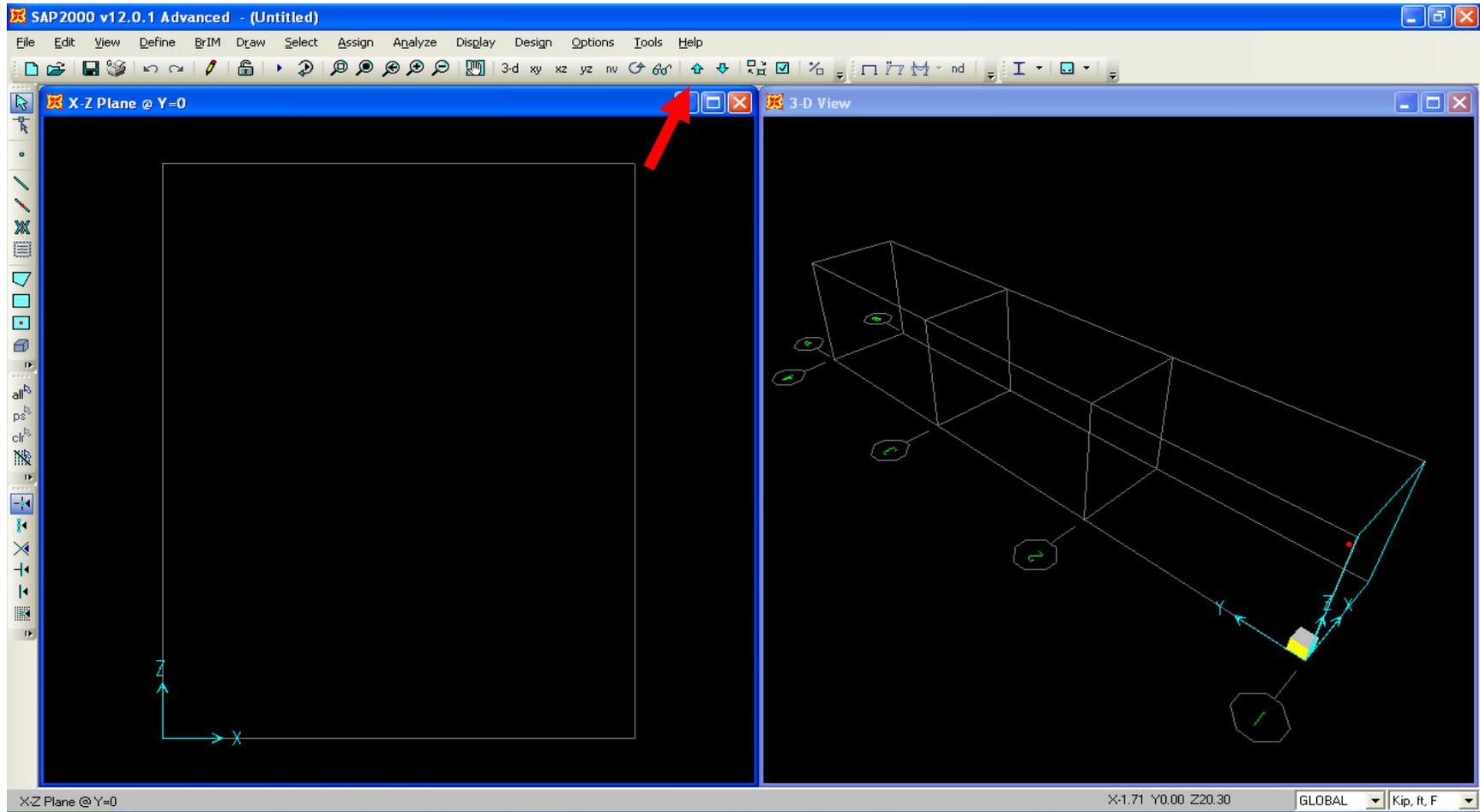
Below you will see the default split-screen view. Note that the planar view on the left (X-Y plan view) is highlighted in blue on the 3D model on the right. As an exercise, click the down arrow as shown below



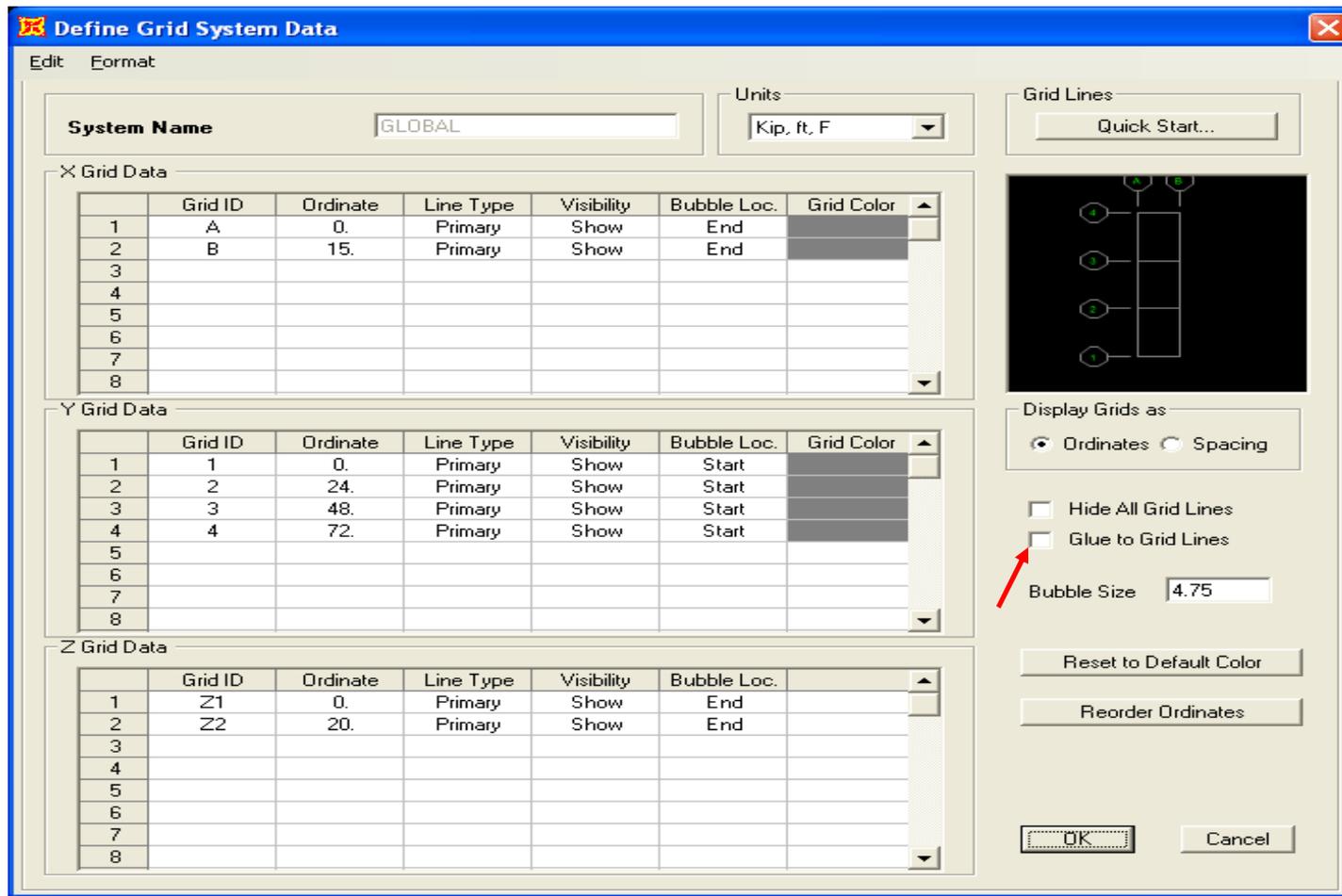
By pressing the down arrow, you move to the base,  $Z = 0$ , which is highlighted on the right window. Next, click xz button to switch to elevation view



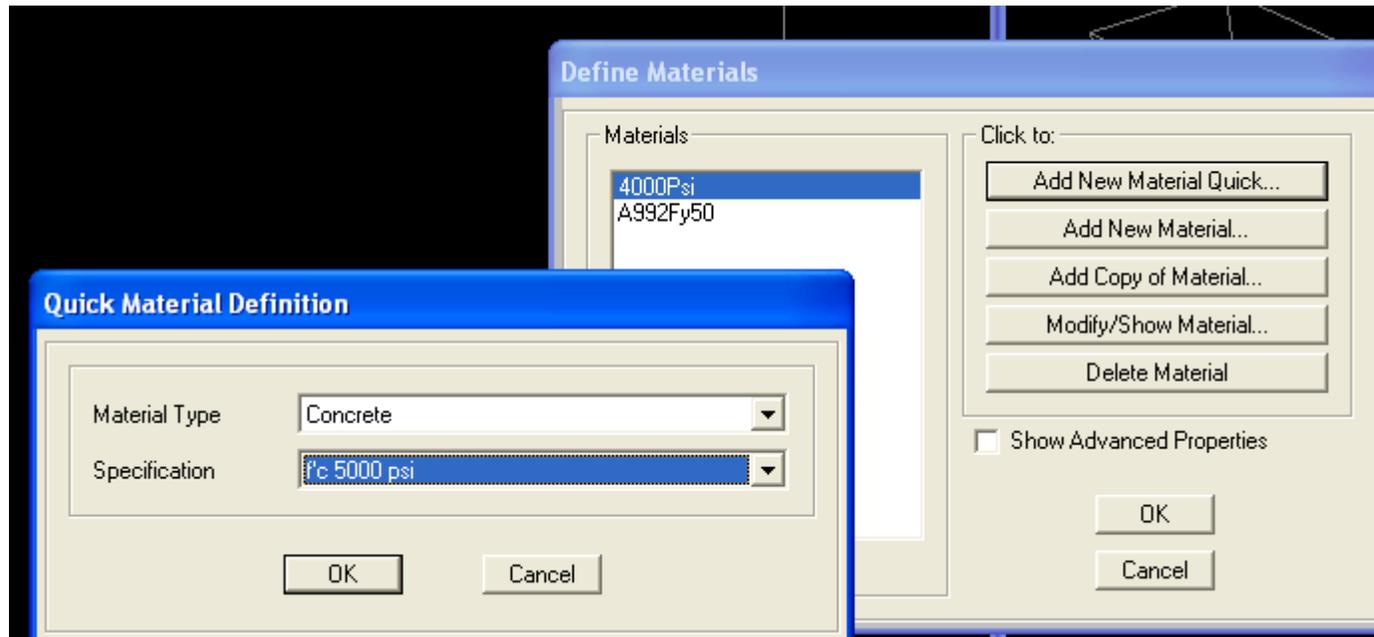
This changes the view from plan to X-Z elevation. Press the up arrow to move to X-Z elevation view Y=0 as shown



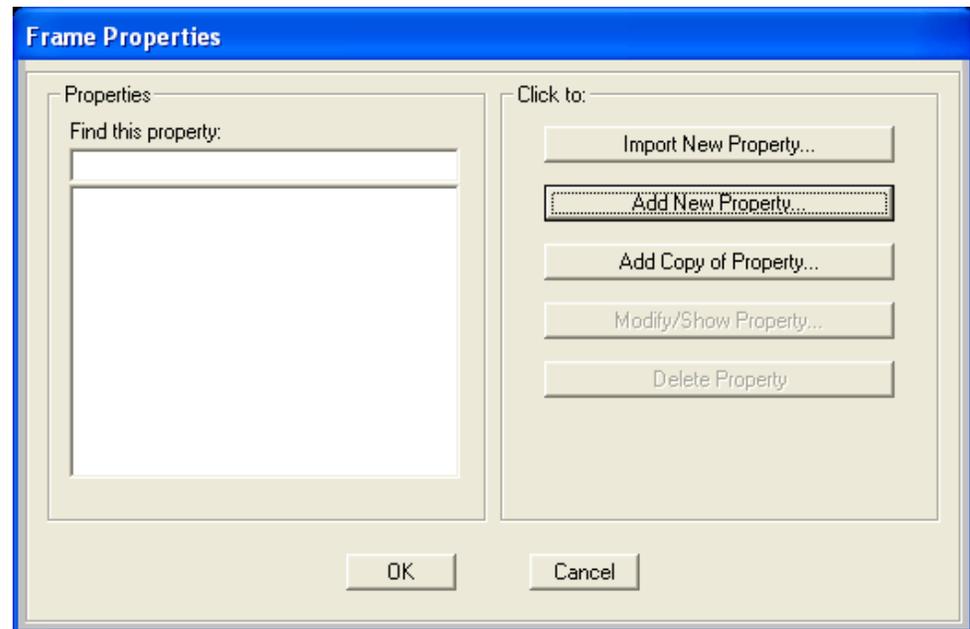
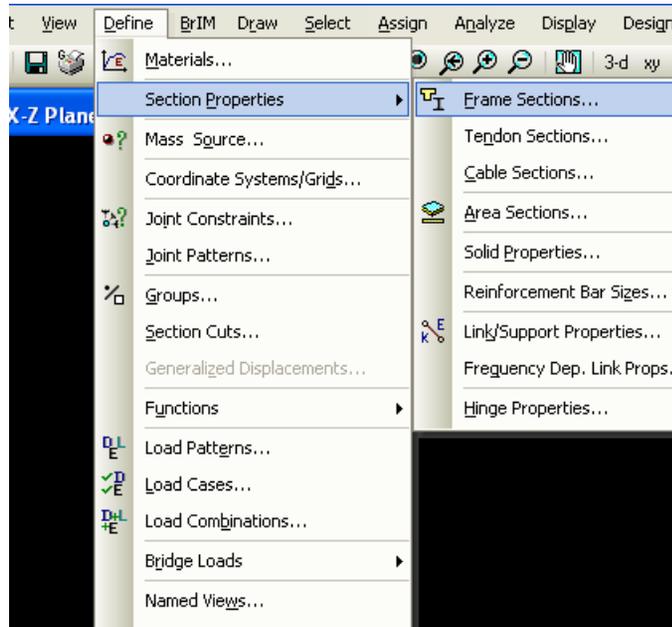
Double click any gridline to display the Define grid system data dialogue. Here you can add, modify or delete gridlines at any time. Grids can be added out of numerical sequence and SAP2000 will automatically reorder them. Select rows and use Edit menu to delete. The 'Glue to Grid Lines' option, when activated, will move and stretch elements associated with a particular gridline that is modified. Cancel out of this dialogue to return to the main screen.



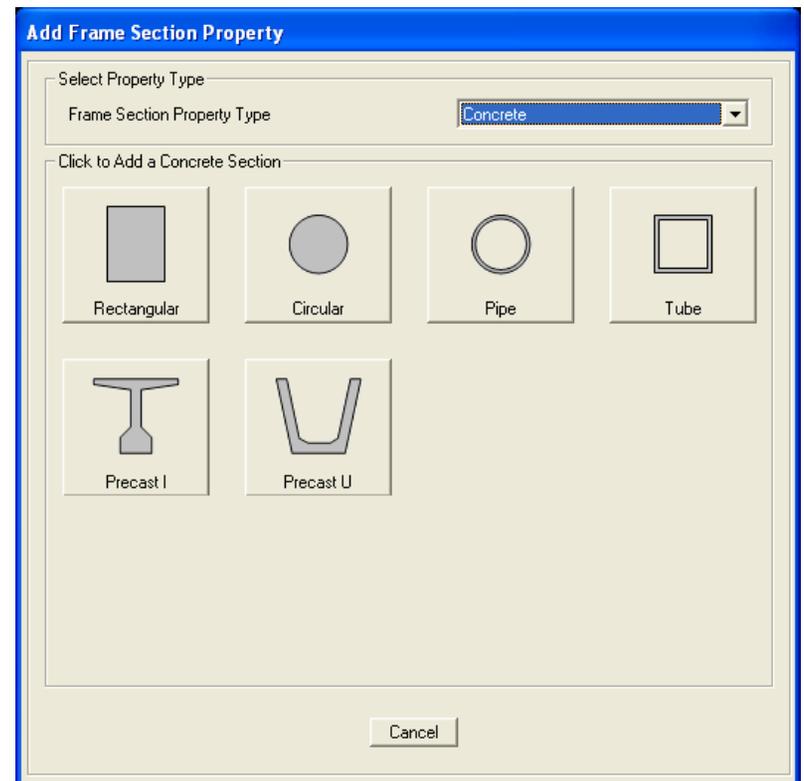
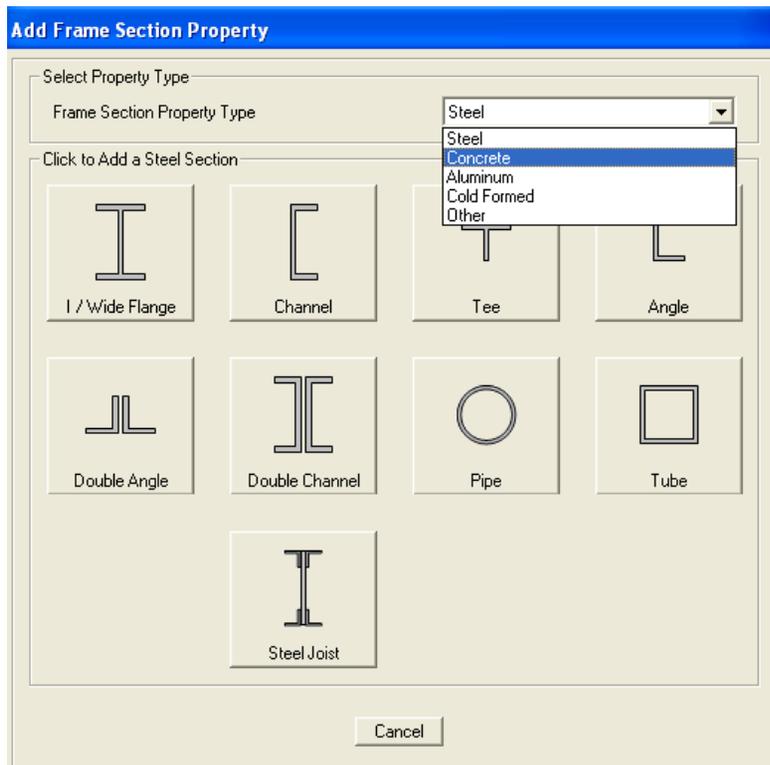
Go to Define>Materials, press 'Add New Material Quick', Material type Concrete and add f'c 5000 psi concrete and press OK to add this new material. SAP2000 offers several libraries of materials for steel, concrete, aluminum and cold formed steel. To define a material not in our standard library, click 'Add New Material' button or 'Add copy of material' from an existing material and input material properties



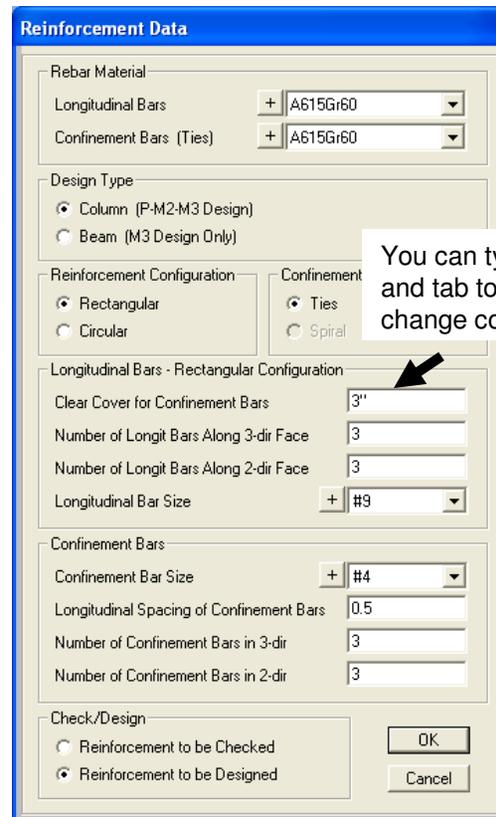
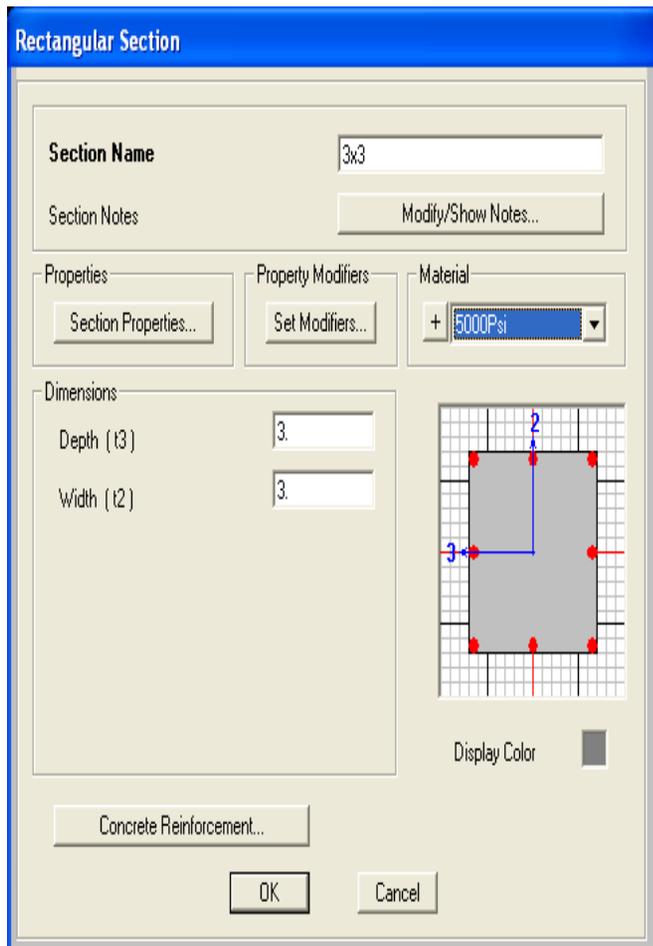
Next we will define frame sections. Go to Define>Section properties>Frame Sections. In this dialogue, you would click 'Import New Property' only if you want to import standard steel sections from a library (AISC and international libraries of sections). For nonstandard steel sections such as Plate Girders, nonprismatic sections, or concrete sections, you use the 'Add New Property'. Click the 'Add New property' button



Select Concrete from the pull down Frame section property type. Please note that any shape in the steel shape list can be changed to concrete, and conversely, any concrete shape can be modeled as steel. For example, you can define concrete I/Wide flanges using the steel shape options, or define solid circular steel rods using the concrete circular shape option.

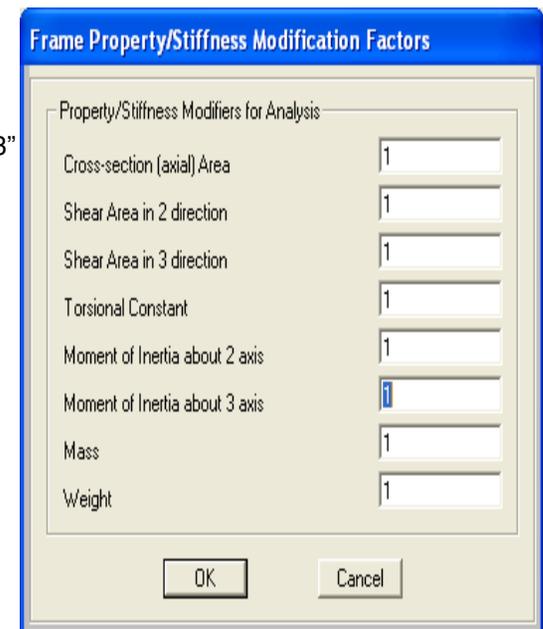


Click the Rectangular shape option, name the section 3x3 to remember it, change the Material to 5000 psi concrete which we defined earlier, and input 3ft in both the depth and width dimensions. Click 'concrete reinforcement' to modify or review design parameters. Then click 'Set modifiers' button. Here you can assign reduction factors to EI for analysis of cracked sections, or assign a multiplier to reduce or increase any section property in this dialogue. Press OK without changing any bars factors and press OK again to save this section

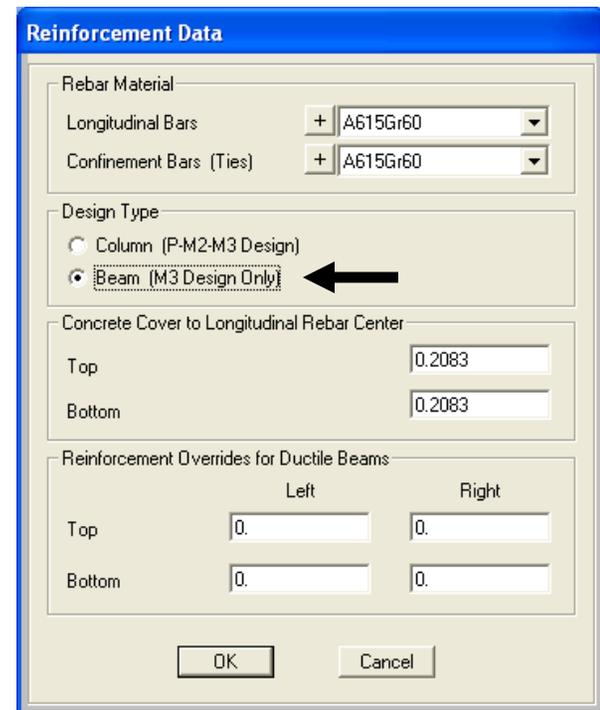
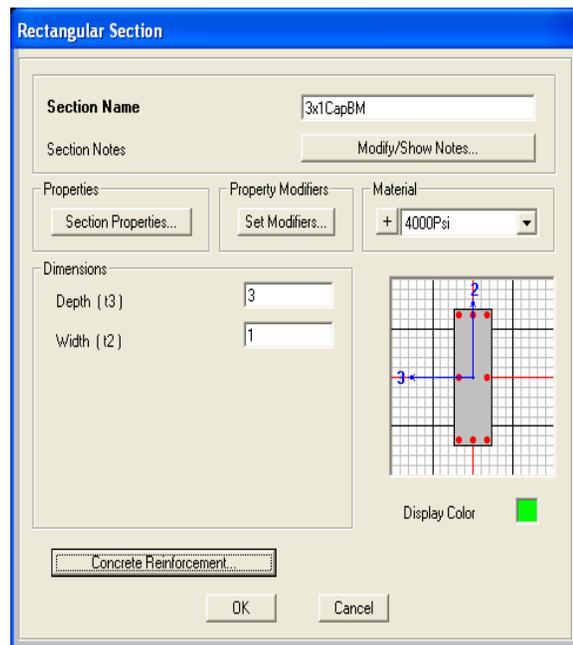
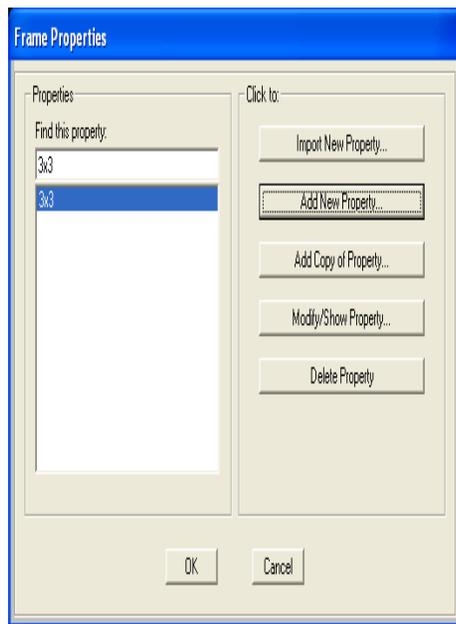


You can type 3" and tab to change cover

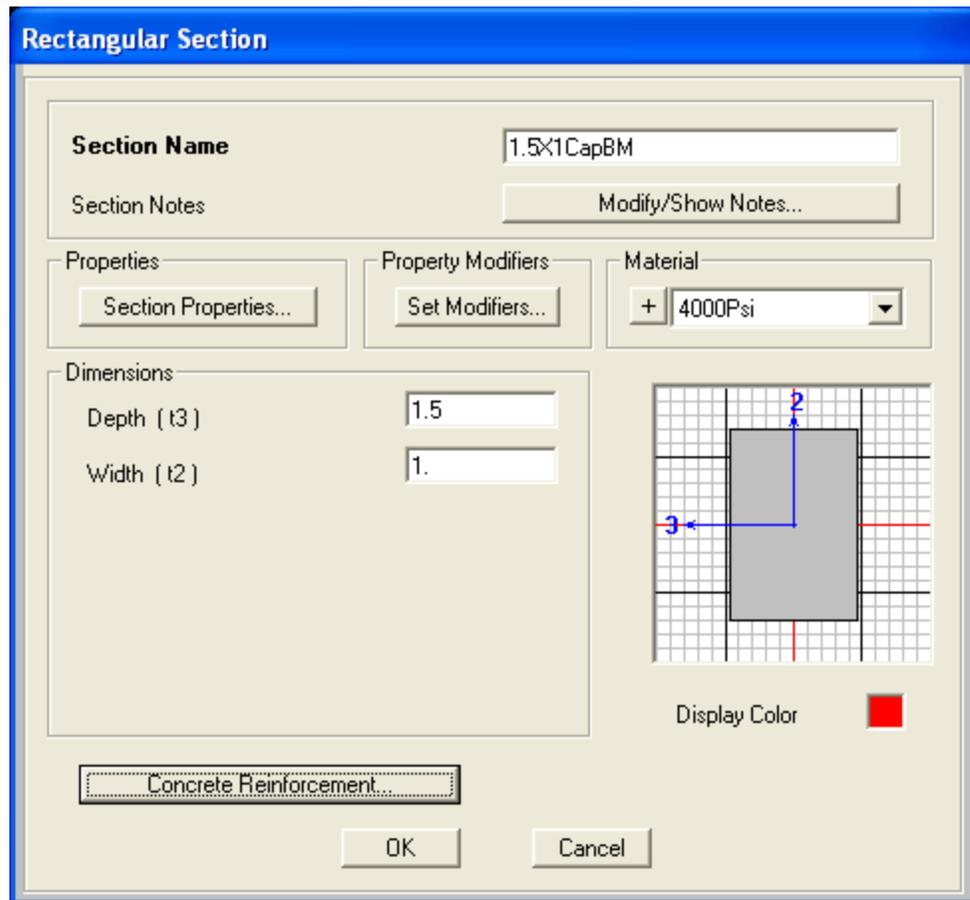
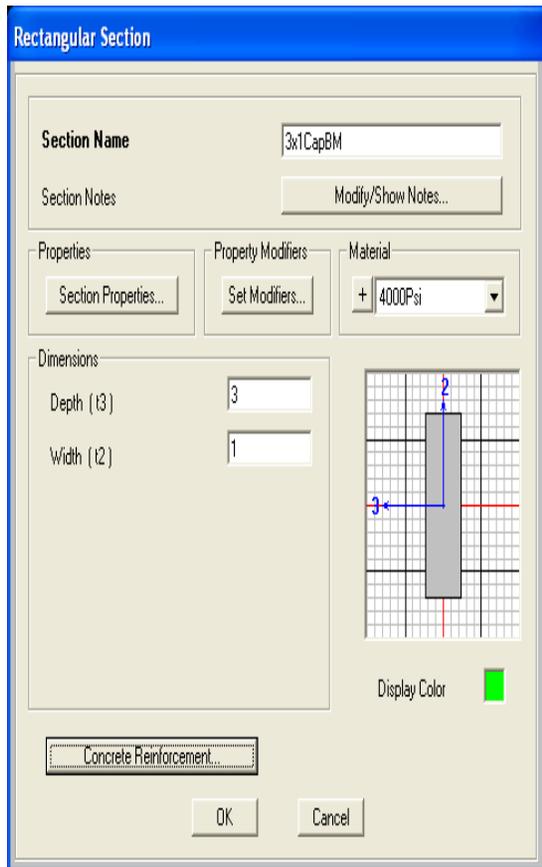
Commonly used to apply reduction factors to EI for cracked sections



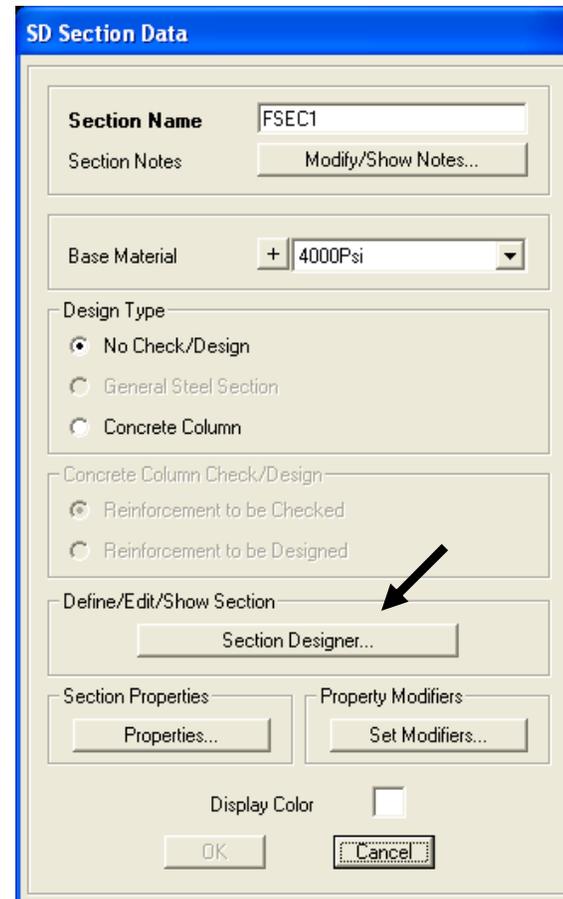
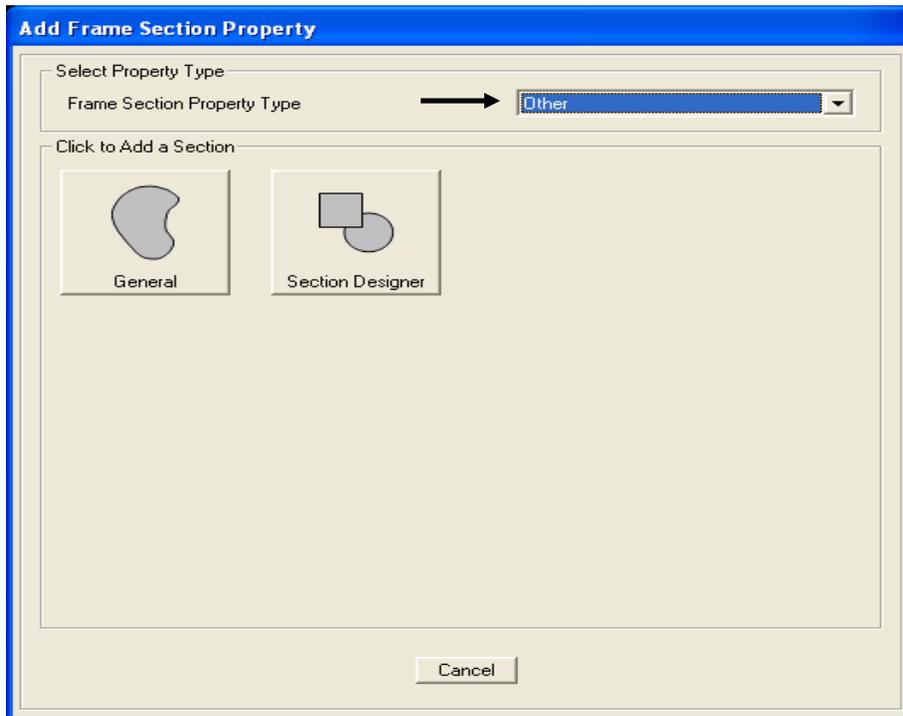
By clicking OK, the section is added to the working list on the left. Next, click 'Add New Property' button and click 'Rectangular' to add another section. Name it 3x1CapBM and type in Depth 3ft. And width of 1 foot. Press 'concrete reinforcement' button and change from Column to Beam as shown and press OK



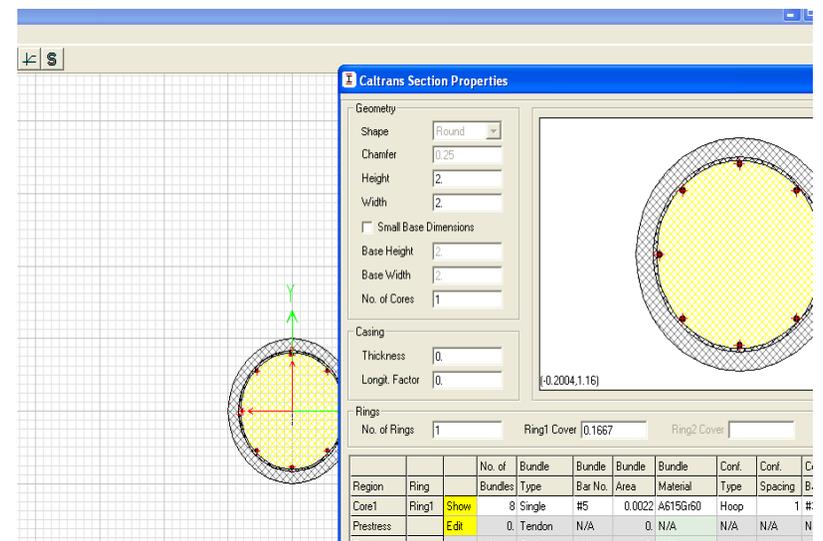
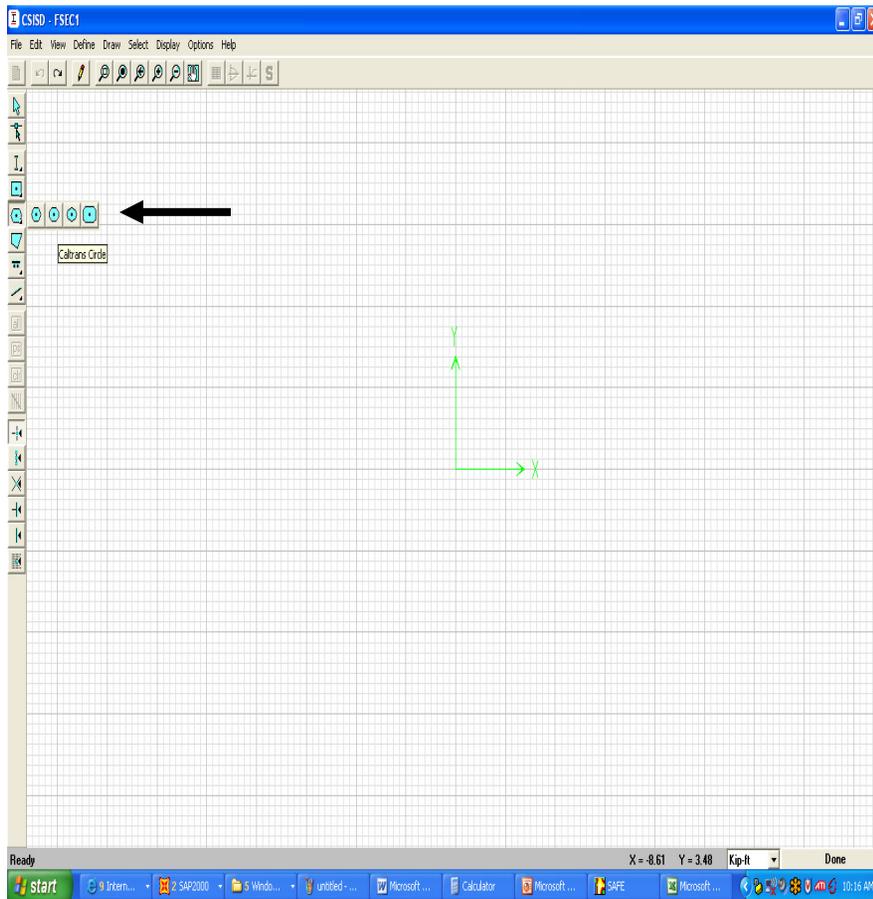
Click OK to accept this new section, click 'Add New Property' and 'Rectangular' once more to add another rectangular section. Name it 1.5x1CapBM, type in the dimensions shown below, click the Concrete reinforcement button and change to beam design type and press OK twice to add this section to the working list



Next, to review SAP2000's section designer, click 'Add new property' again with Frame section property type 'Other', click Section Designer, then in the SD Section data dialogue click the 'Section Designer' button



Click one of the Caltrans section options and then click on the origin in the middle to draw it. Next, right click the section to review options and click OK without making any changes.



Click options to display section properties, P-M-M curves, and moment curvature. The moment curvature plot also enables you to calculate cracked sections based on load if you select the 'Caltrans idealized model' option. Click Done and Ok until you exit the section designer as we wanted to introduce the SD capability within this tutorial, but we will not use a SD section for this exercise.

**Properties**

Base Material: 4000Psi  
 Xcg: 0.  
 Ycg: 0.  
 Axis Angle: 90

A	3.1214
J	1.5507
I33	0.7754
I22	0.7754
I23	0.
AS2	2.8179
AS3	2.8179
S33(+face)	0.7754
S33(-face)	0.7754
S22(+face)	0.7754
S22(-face)	0.7754
Z33	1.3205
Z22	1.3205
r33	0.4984
r22	0.4984
d33pna	0.
d22pna	0.

OK

**Moment Curvature Curve (Limits: P(comp.) = -1668.628, P(ten.) = 148.8)**

Edit

**Curvature**

**Strain Diagram**

Concrete Strain: -5.026E-03  
 Steel Strain: 4.661E-03  
 Neutral Axis: 0.079

Select Type of Graph: Moment-Curvature  
 Specify Scales/Headings...: (5.455E-03, 370.21)

Plot Exact-Integration Curve  Show Numerical Results for Exact-Integration Curve  
 Plot 3x3 Fiber Model Curve  Show Numerical Results for Fiber Model Curve

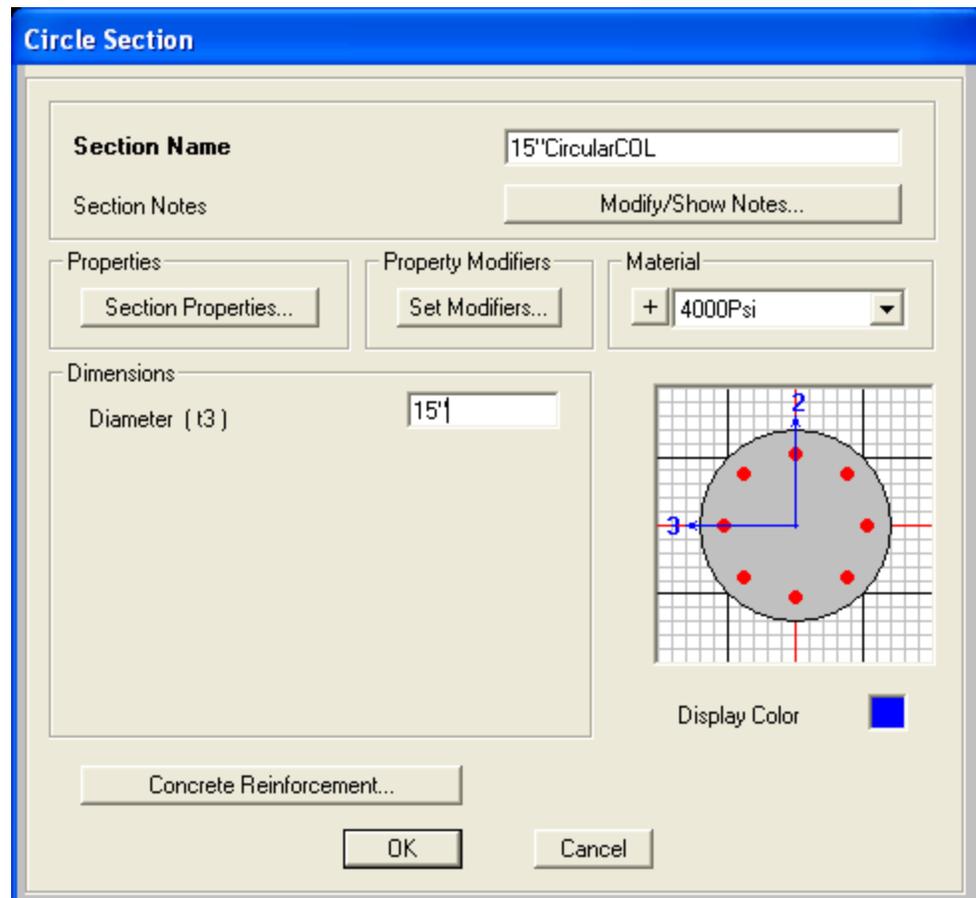
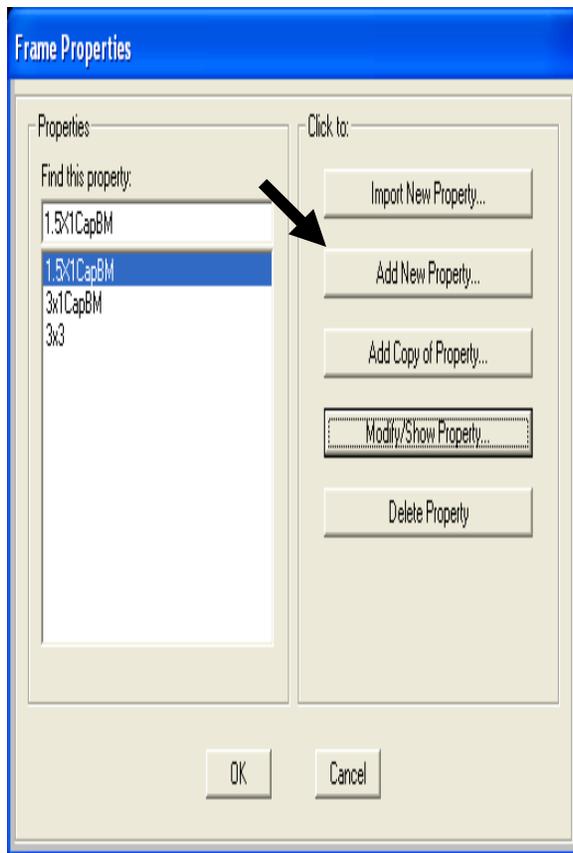
Caltrans Idealized Model No. of Points: 20  
 P [Tension +ve]: -600 Angle (Deg): 0.  
 Max Curvature: 8.747E-03 Mmax = 284.927  
 Phi-Conc = .00874705 M-Conc = 284.927  
 Phi-Steel = N/A M-Steel = N/A  
 Phi-yield(initial) = .00264524 M-yield = 379.325  
 Phi-yield(idealized) = .00264524 Mp = 379.3254  
 ICrack = .276

Analysis Control  
 Concrete Failure (Lowest Ultimate Strain)  
 Concrete Failure (Highest Ultimate Strain)  
 First Rebar Failure  
 User Defined Curvature

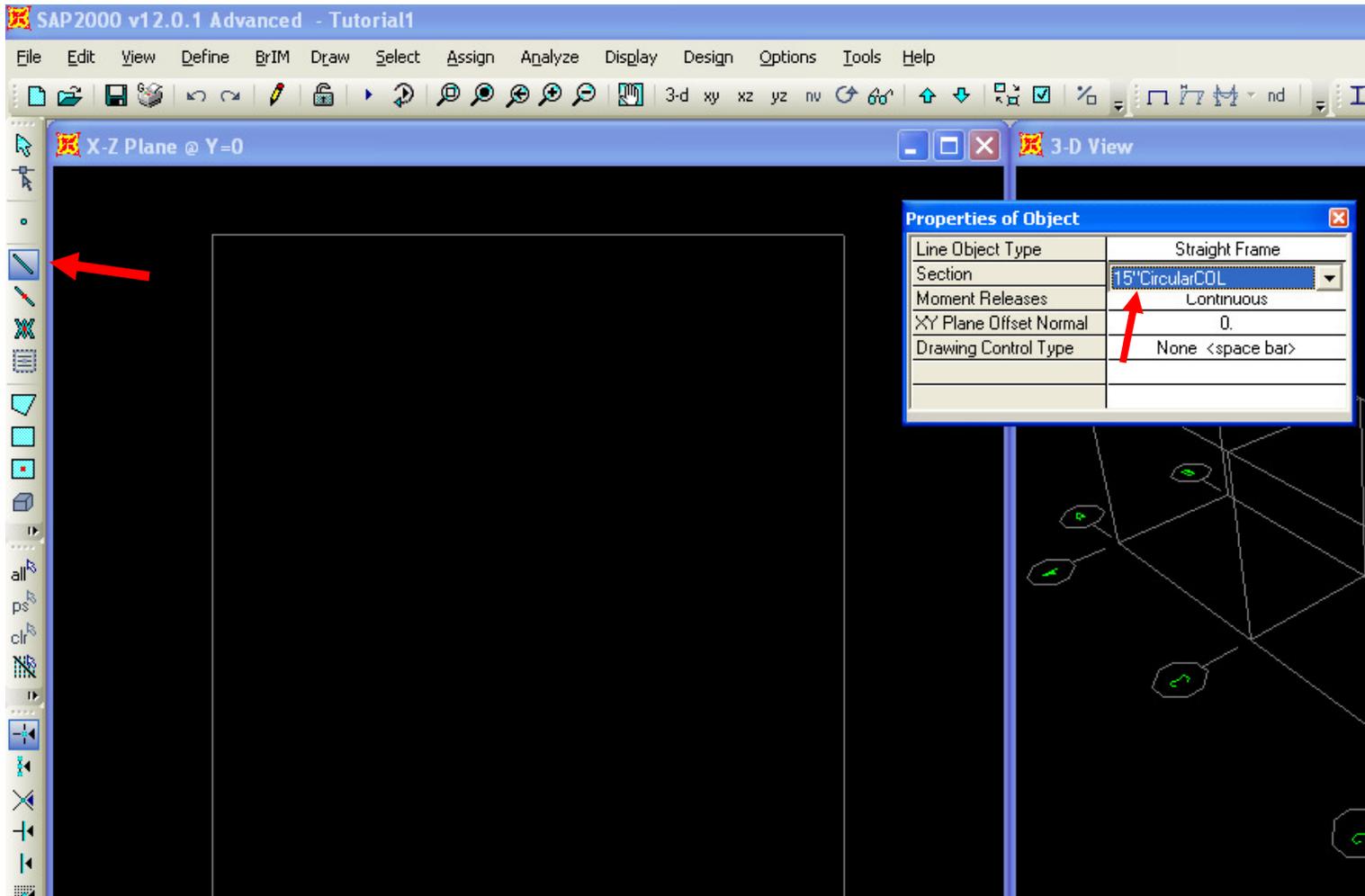
Details... Contour...  
 Refresh Done

Curves  
 New Curve  
 Selected Curve Color: [Black]  
 Click to: Add Curve Delete Curve

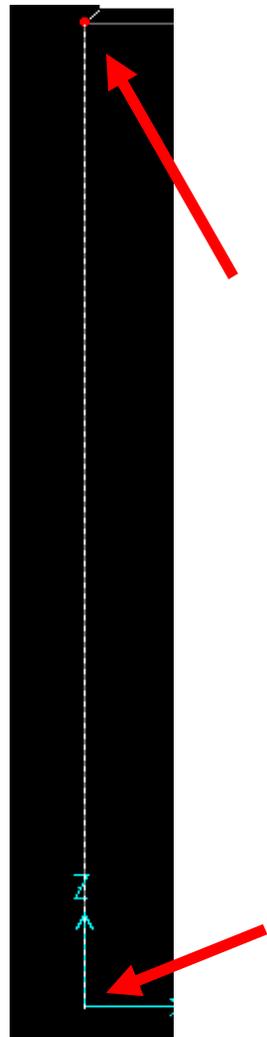
Click 'Add new property' button once more, but this time click Concrete 'Circular' type, name it 15"CircularCOL, type 15" and then press tab to have SAP2000 convert to current ft. units. Click OK twice to accept and save defined sections. Click the save button  and name the model *Tutorial1*



You should still have a split screen view as shown below with X-Z planar view on the left. If not, click your left window to make it active, then click the xz button to change view and up/down arrow keys   if you need to adjust further. Next, click 'Draw frame/cable element' button as shown and click the section to change the section property to *15"CircularCOL* in the pop-up properties box.



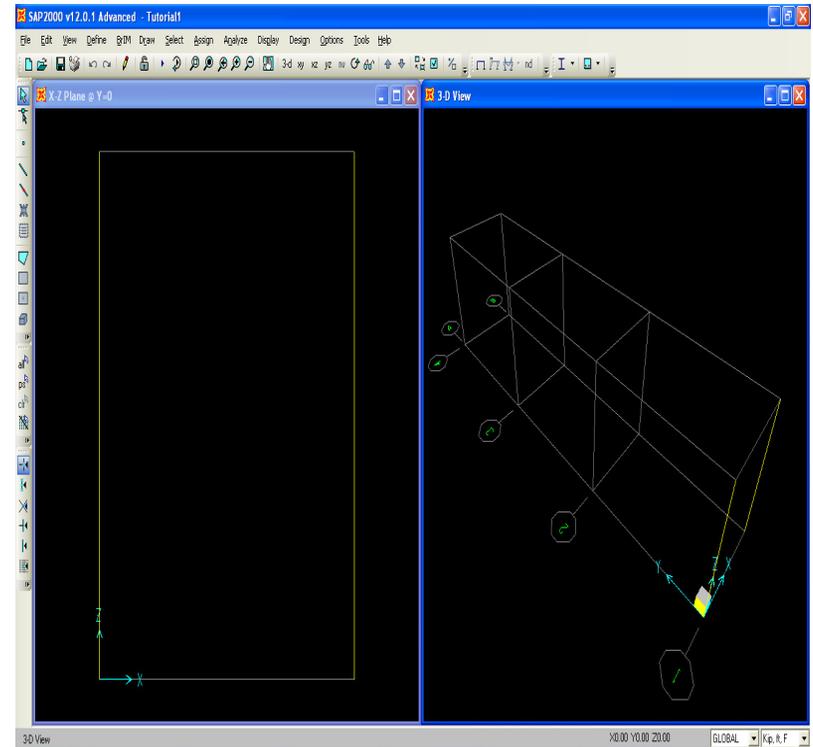
In X-Z view window, click once on the bottom left grid intersection and then click up to the top left grid intersection in order to draw the column. Next, right mouse click to lift your cursor to draw another column on the right side. Click once on the bottom right intersection and complete the column by clicking on the top right grid intersection. Next, press Esc key on your keyboard or the select button/icon  to switch to select mode



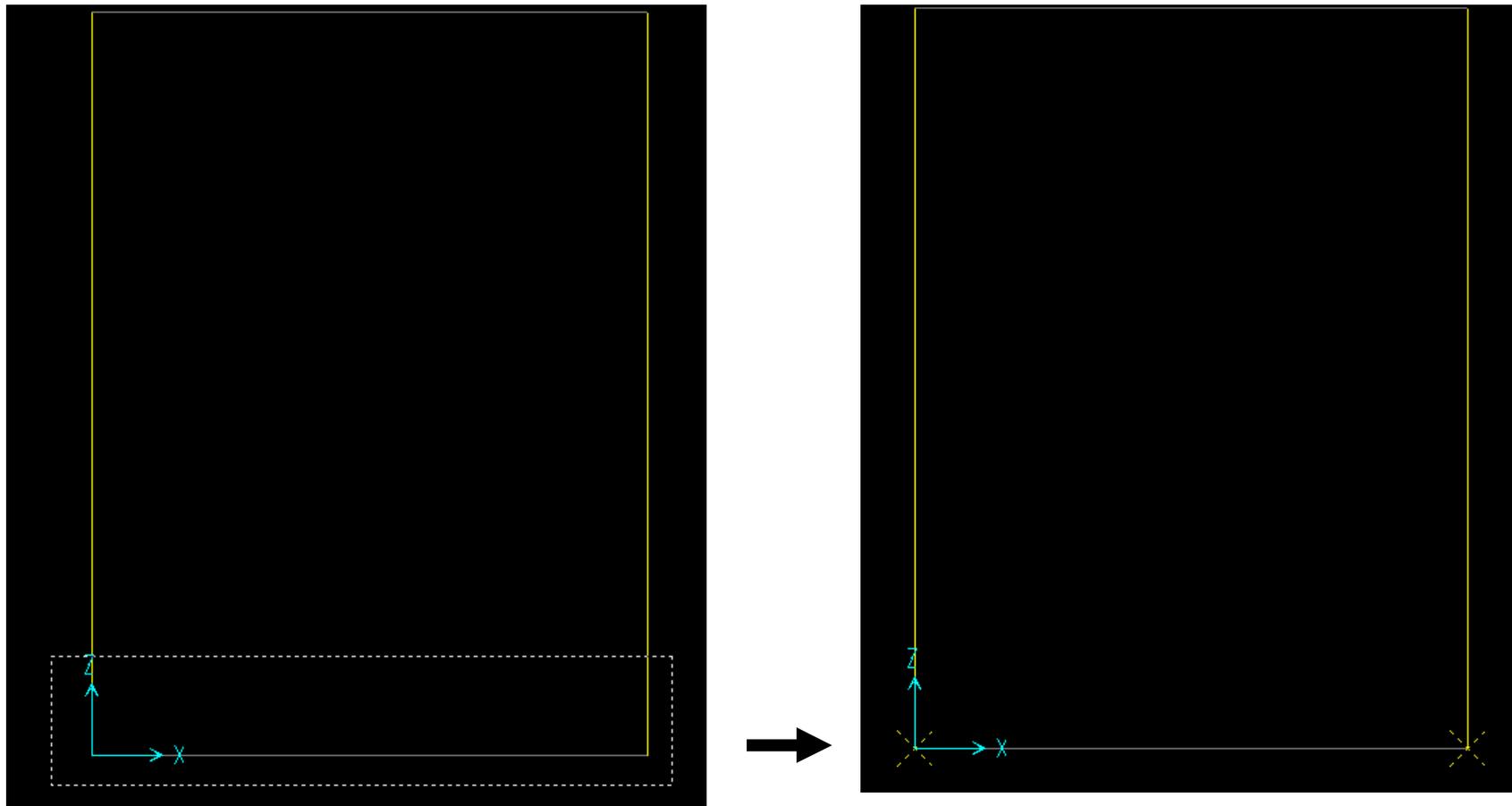
Draw column with two mouse clicks



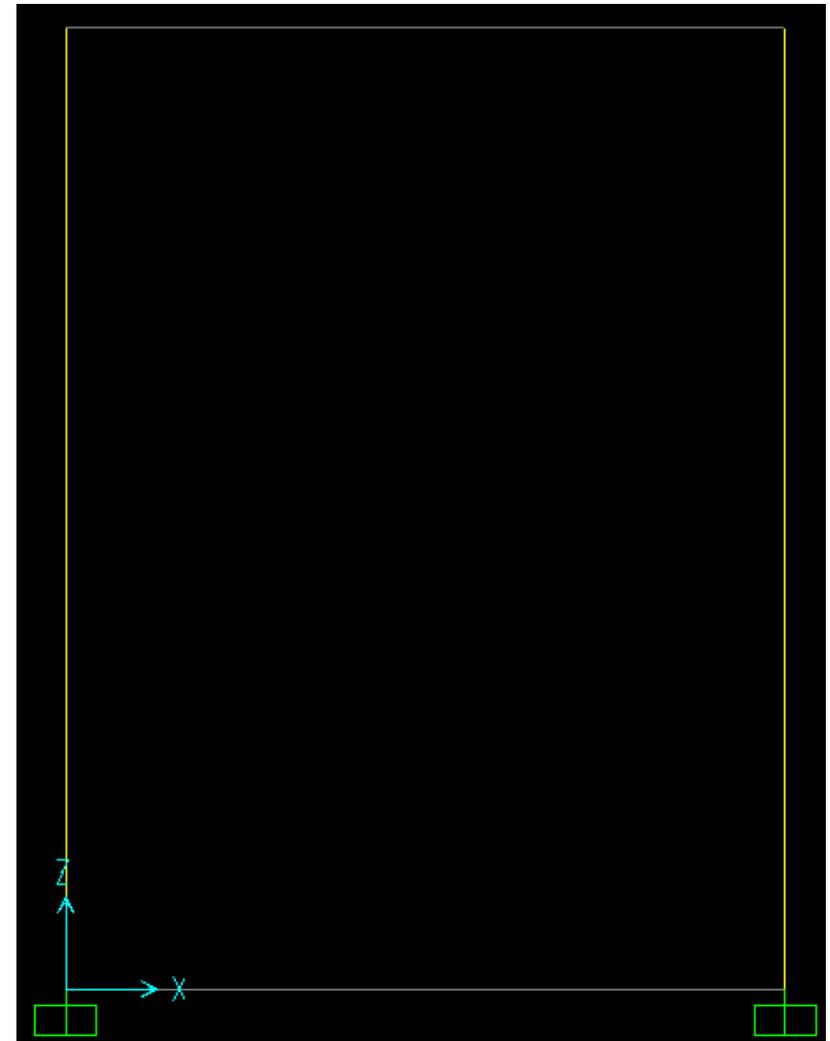
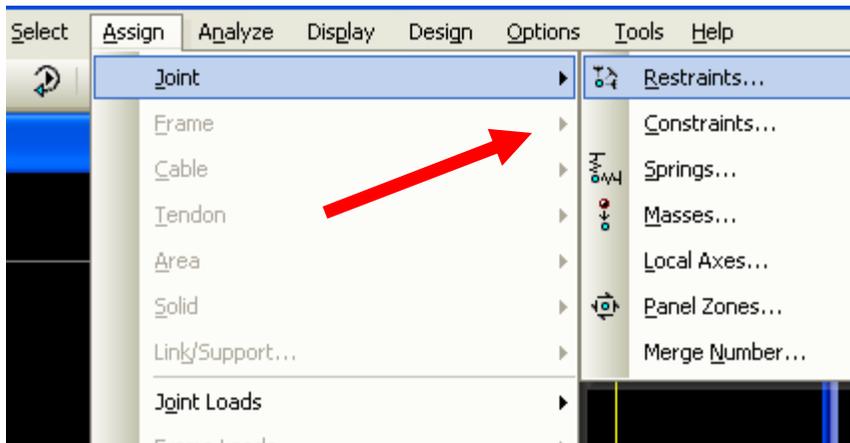
Elevation view



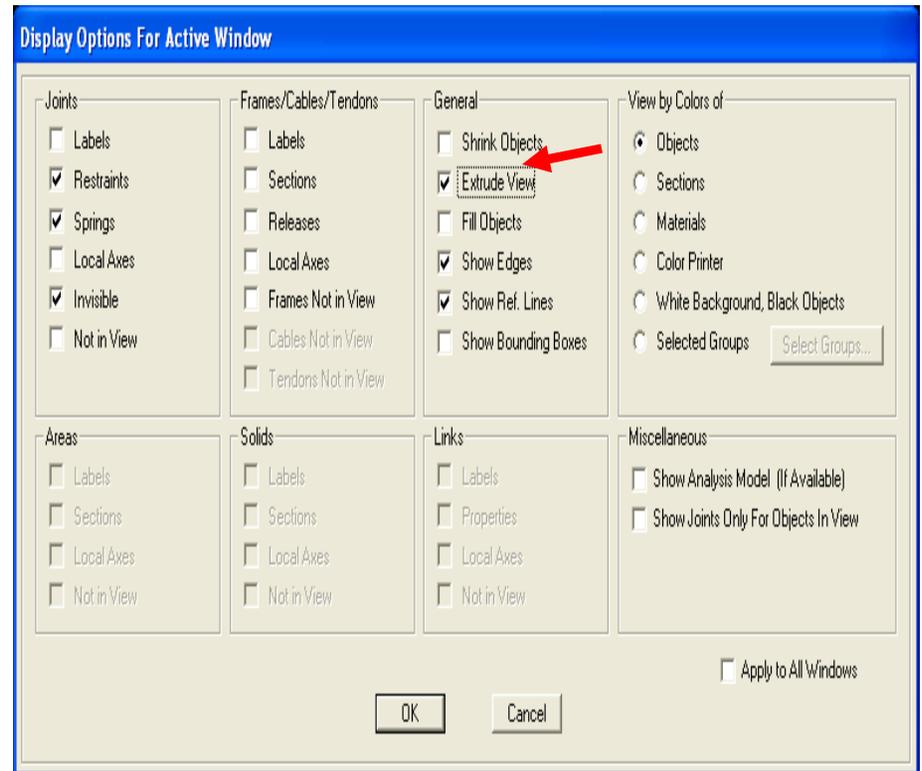
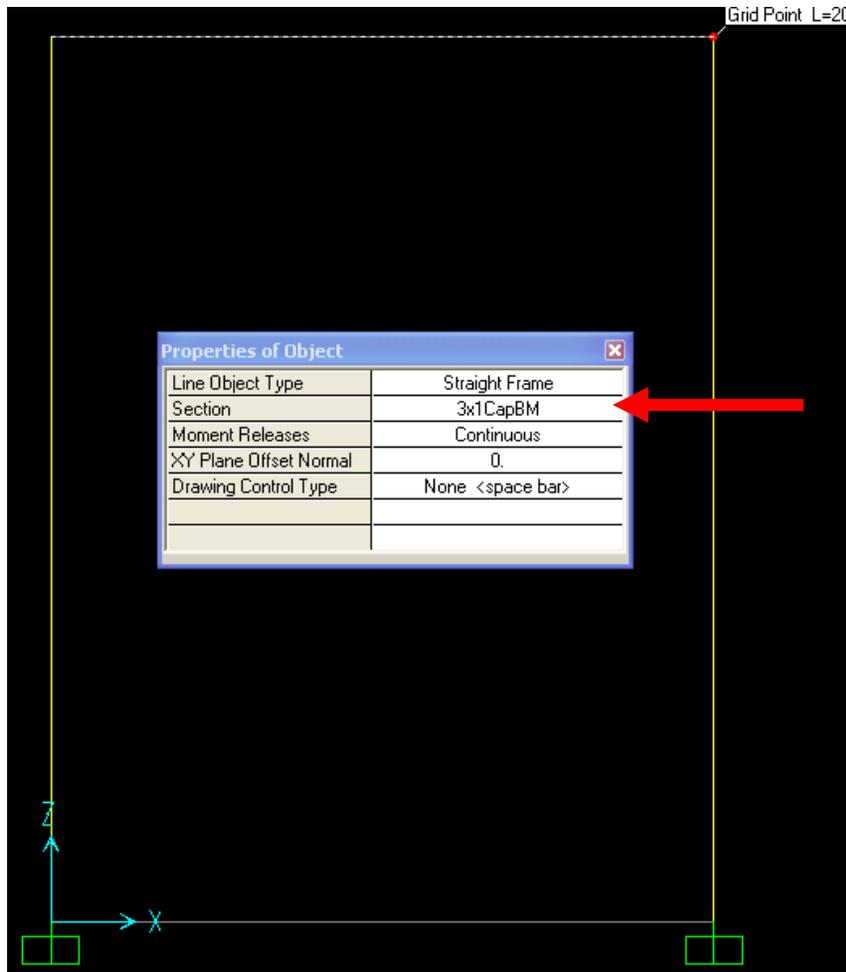
Holding your left mouse key down, drag your mouse to window around the base of the two columns in order to select bottom joints as shown



Next, use the Assign menu>Joint>Restrains to add fixed restraints as shown. Press OK

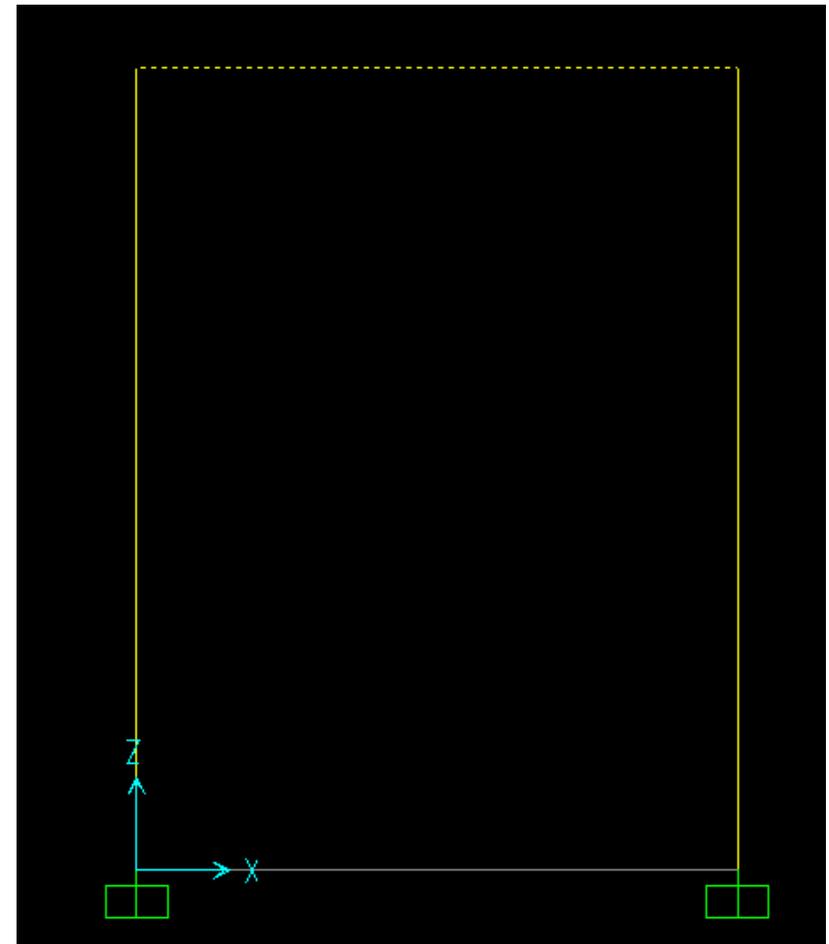
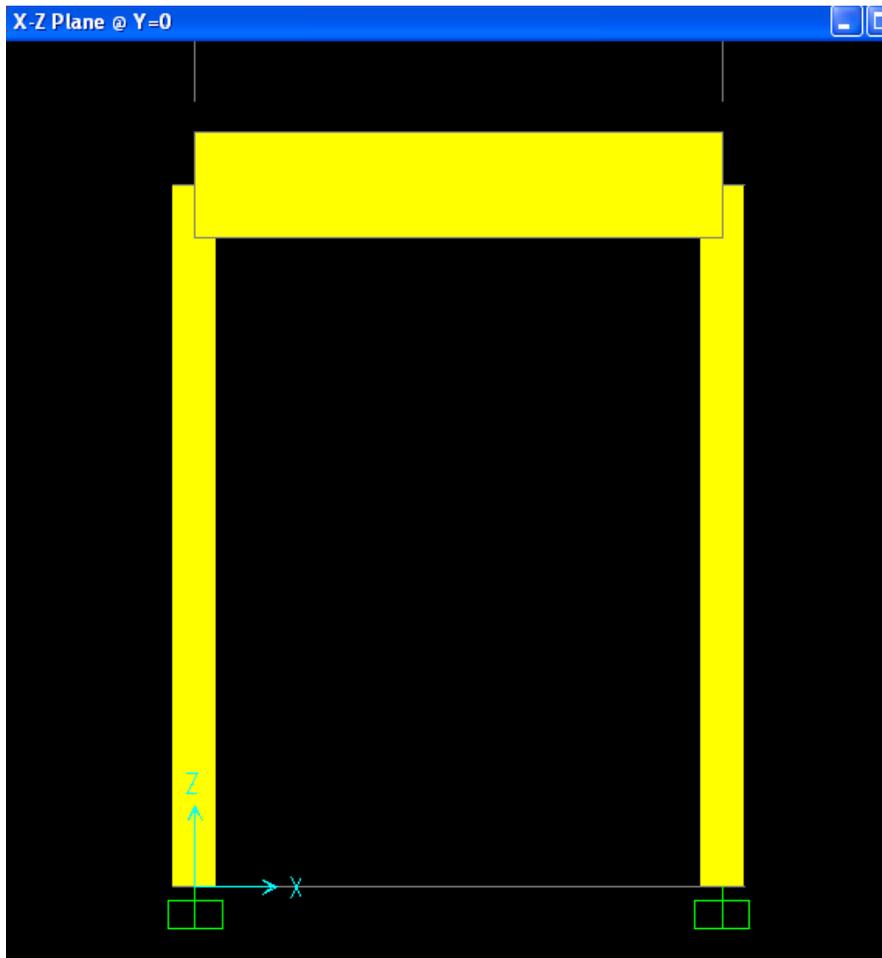


Click 'Draw frame/cable element' button, change Section property to 3x1CapBM and click twice to draw the cap beam as shown. Click 'Set Display options' button  and checkbox 'Extrude view' option and press OK

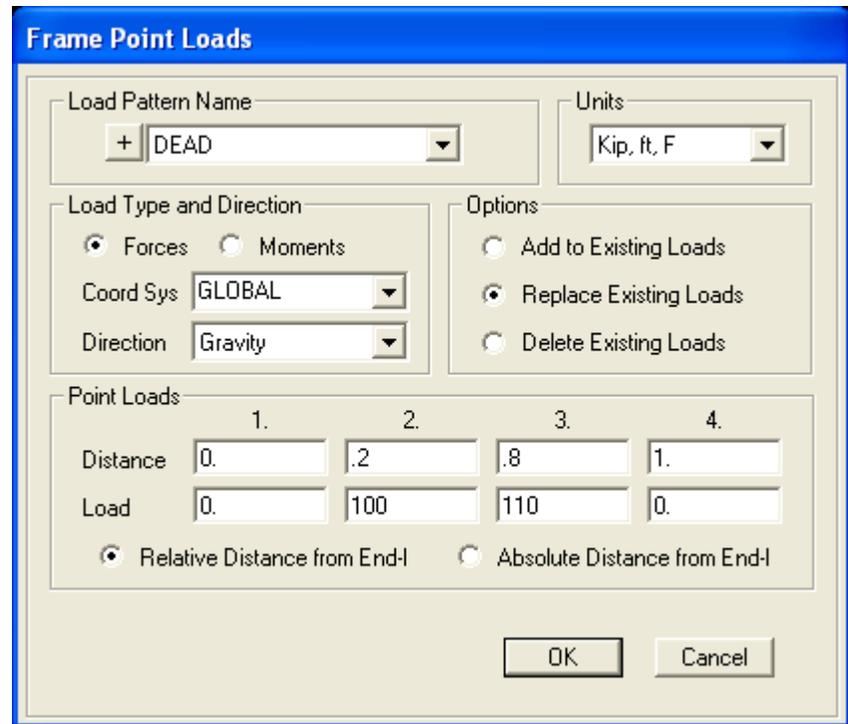
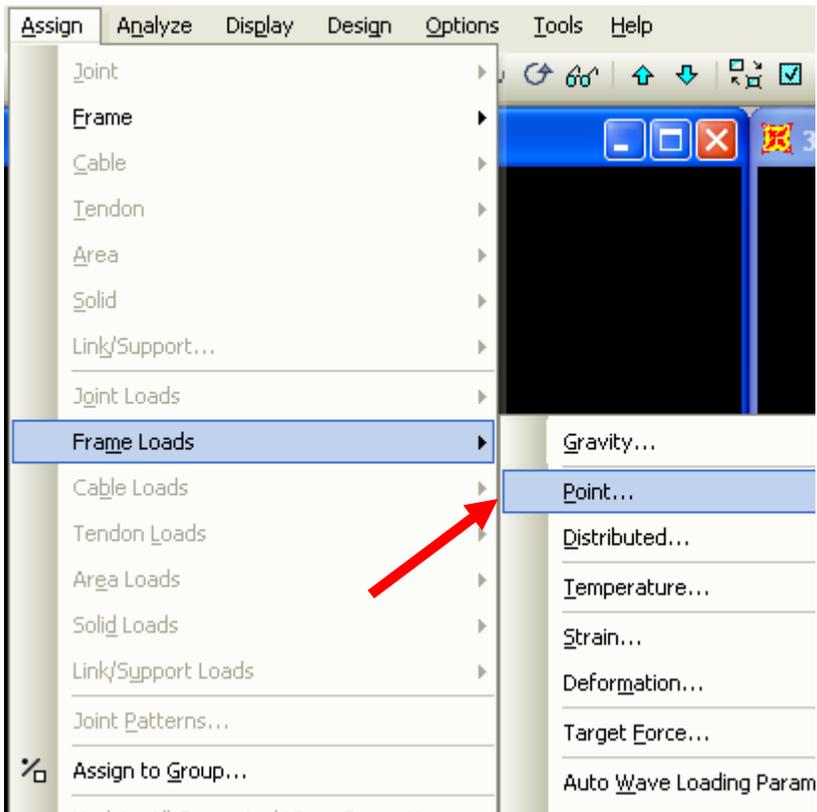


The extruded view enables you to see a proportional rendered view of the model. You can see that we have basic centerline connection. By clicking in a window to make it 'active', you can display extruded view in 1 window or all windows. Click 'Set display options' box  once more, uncheck the 'extruded view' option and select the beam by clicking on it as shown.

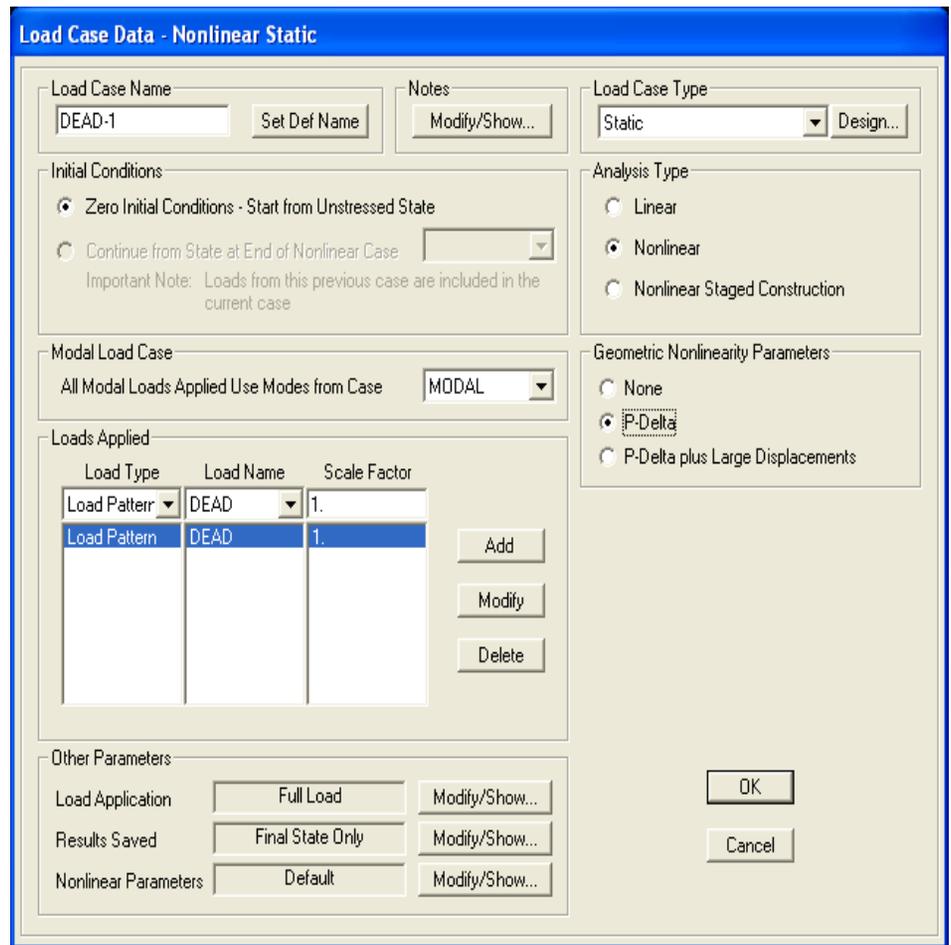
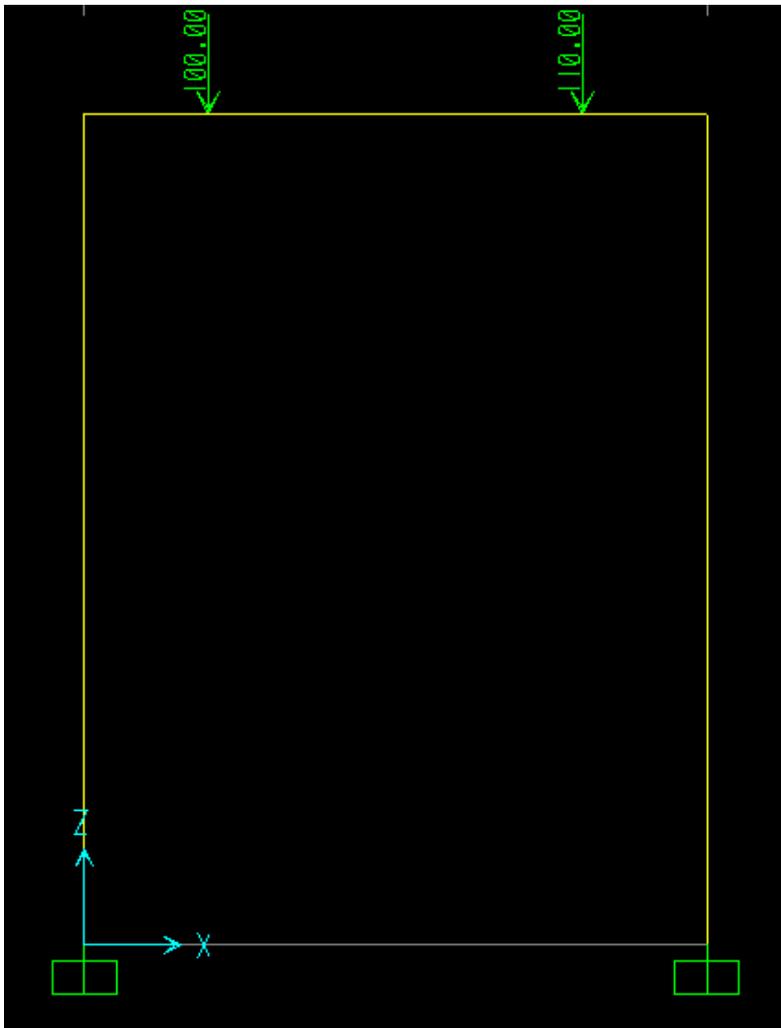
Selected beam displays as dotted line



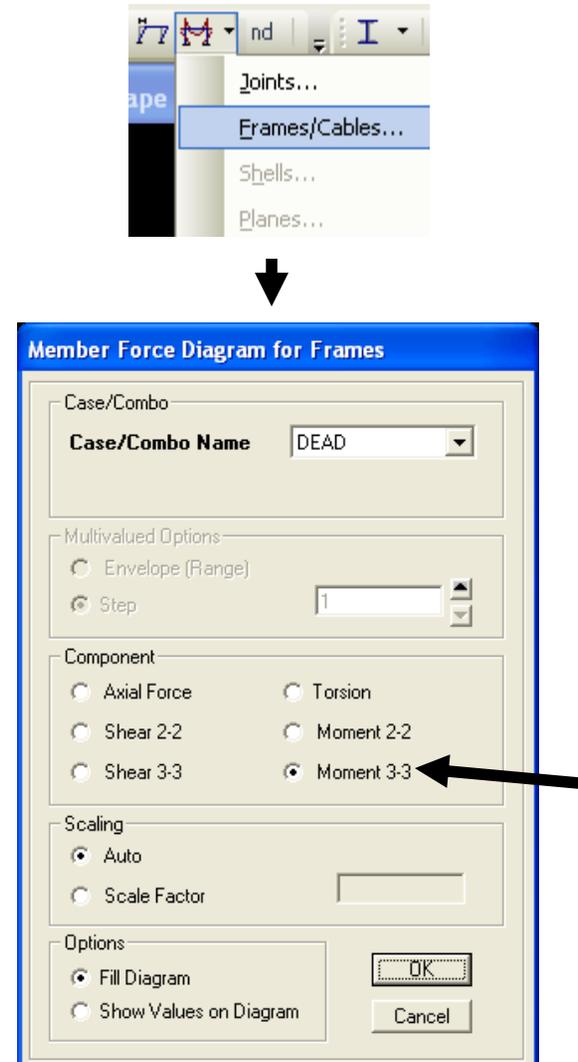
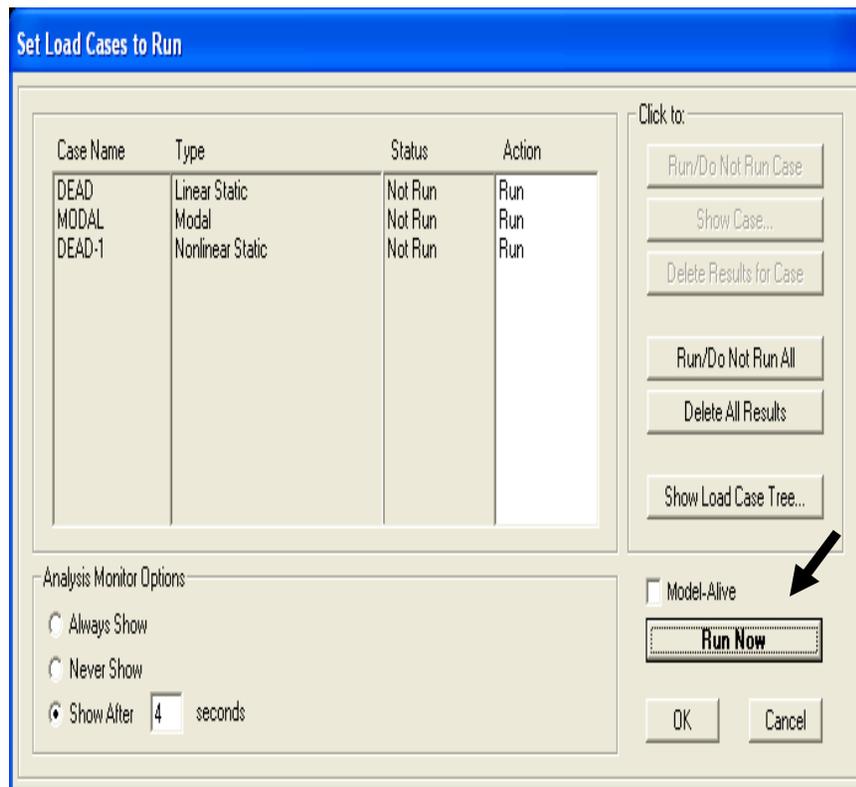
Go to Assign>Frame loads>Point to assign point loads along the length of the beam. Using Relative percentage distance, change the 2<sup>nd</sup> distance to .2 and 3<sup>rd</sup> distance to .8 and input a 100 Kip load for 2 and 110 Kip load for 3 and press OK to assign these loads in the DEAD load pattern. By clicking the + button next to the load pattern name, you could define additional load pattern cases to assign the load to. Alternatively, you could Define load patterns and combos under the Define menu before assigning loads



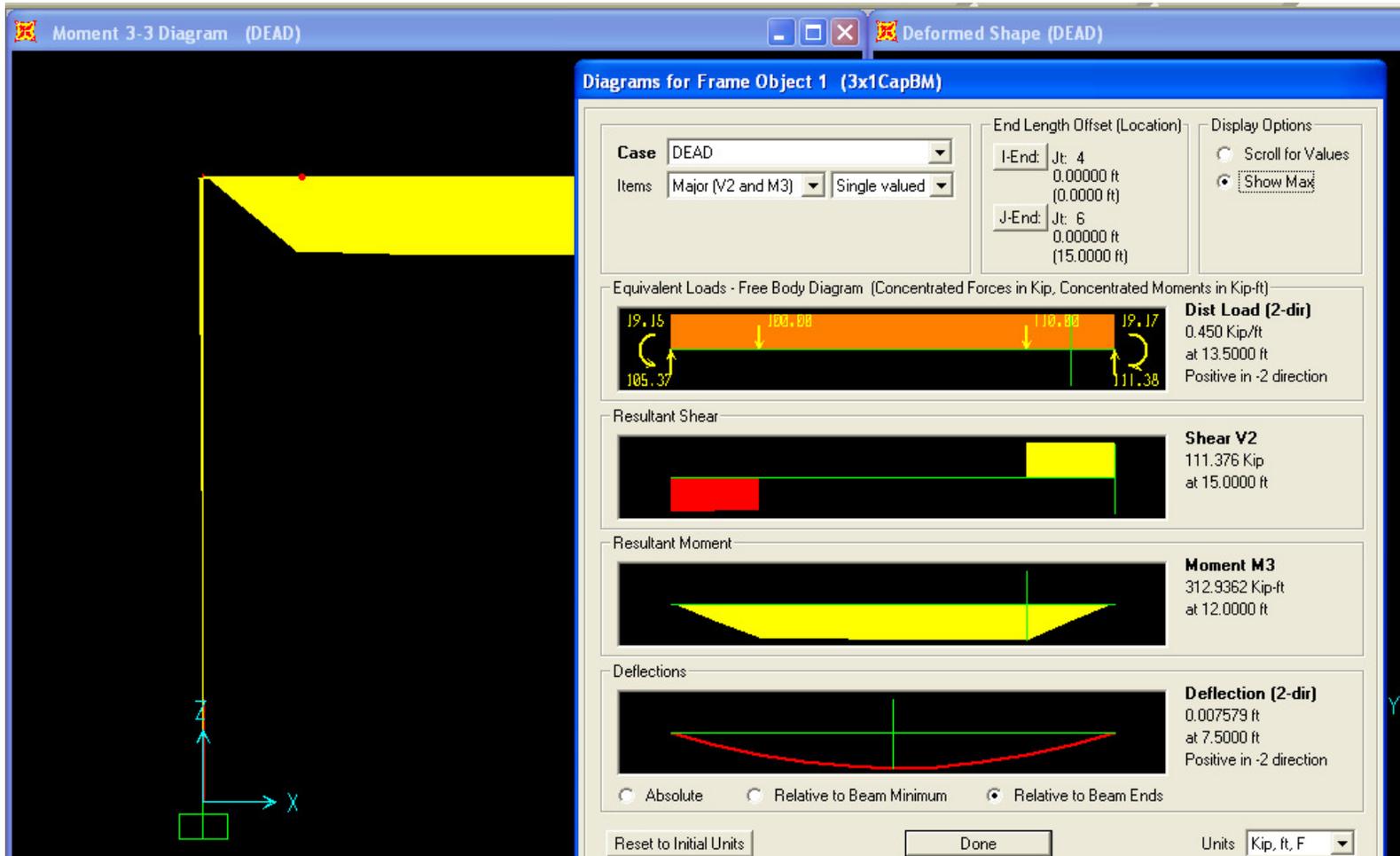
In order to consider p-delta effects, go to Define>Load cases, click DEAD and press the 'Add copy of load case' button and check the checkbox Nonlinear and P-delta as shown below and press OK twice to accept. Alternatively, we could have modified the DEAD case, but by making a copy of it, we can compare results between the P-delta DEAD case and the DEAD case without P-delta in the same analysis



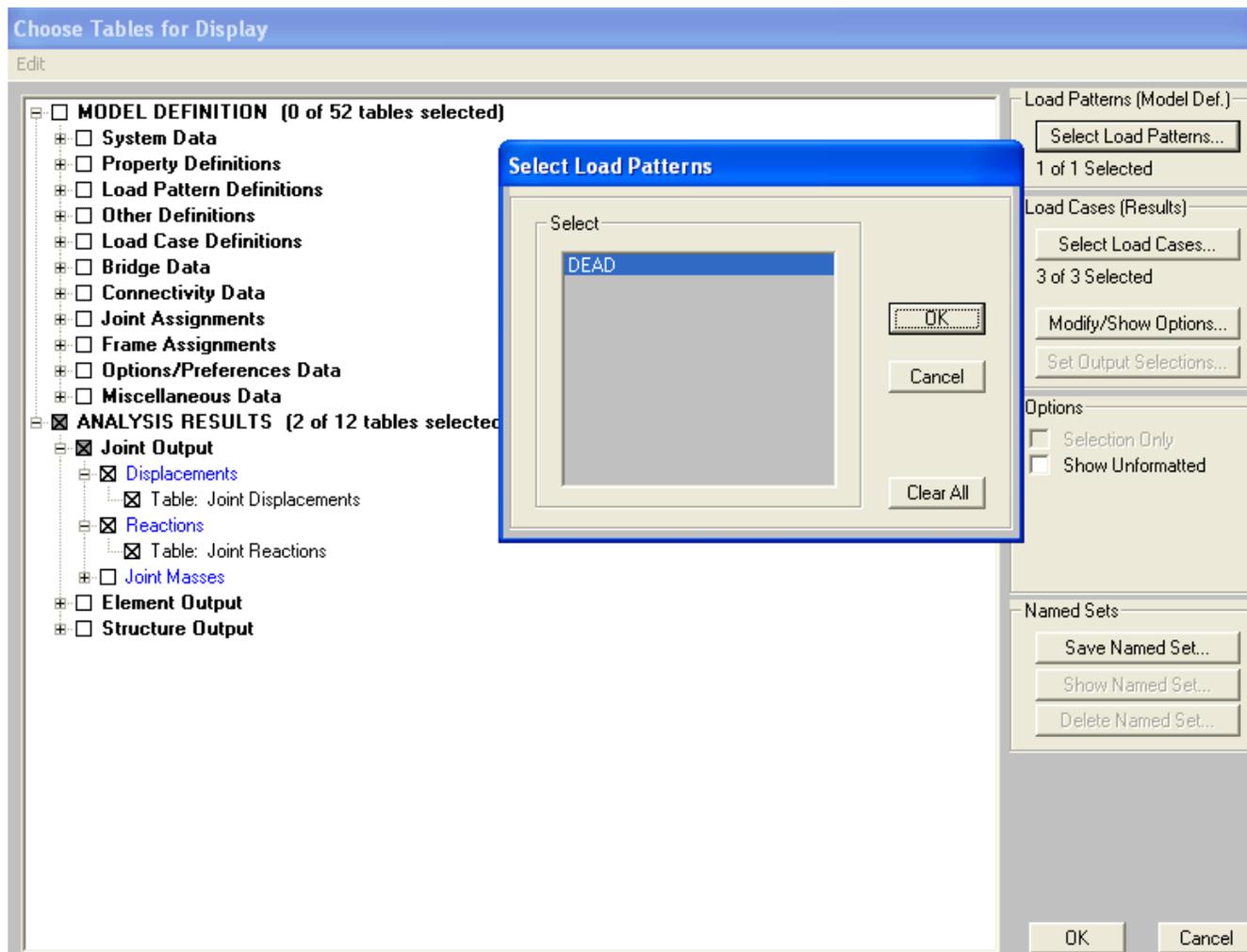
Press F5 on your keyboard to run the analysis and press 'Run now' button as shown. After the analysis has completed, click the arrow next to the 'Show forces/stresses' button as shown and select Moment 3-3 to interactively display major moment.



You can then right click individual frames for results as shown, toggling between cases and results options. Press Done.

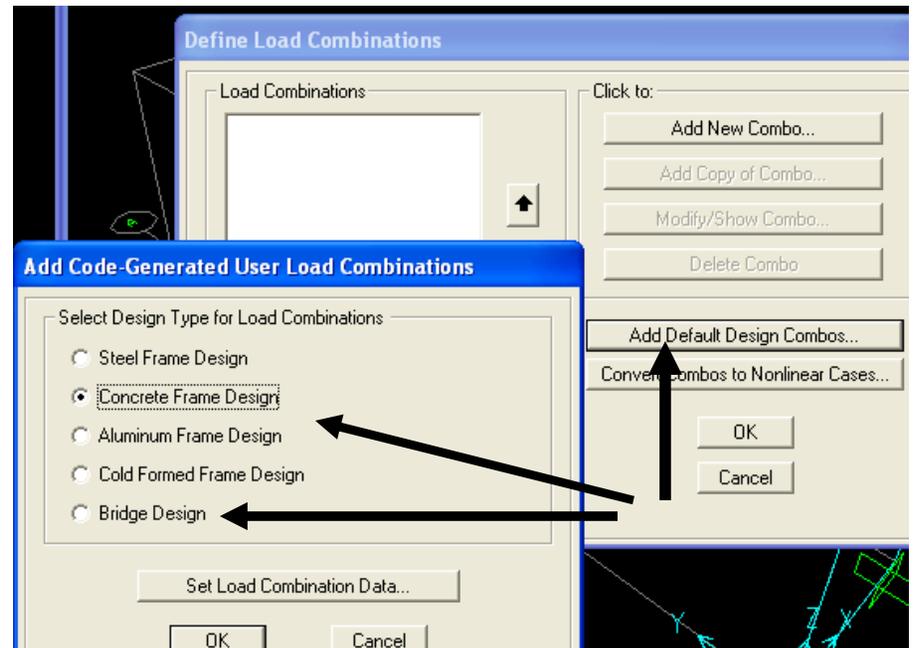
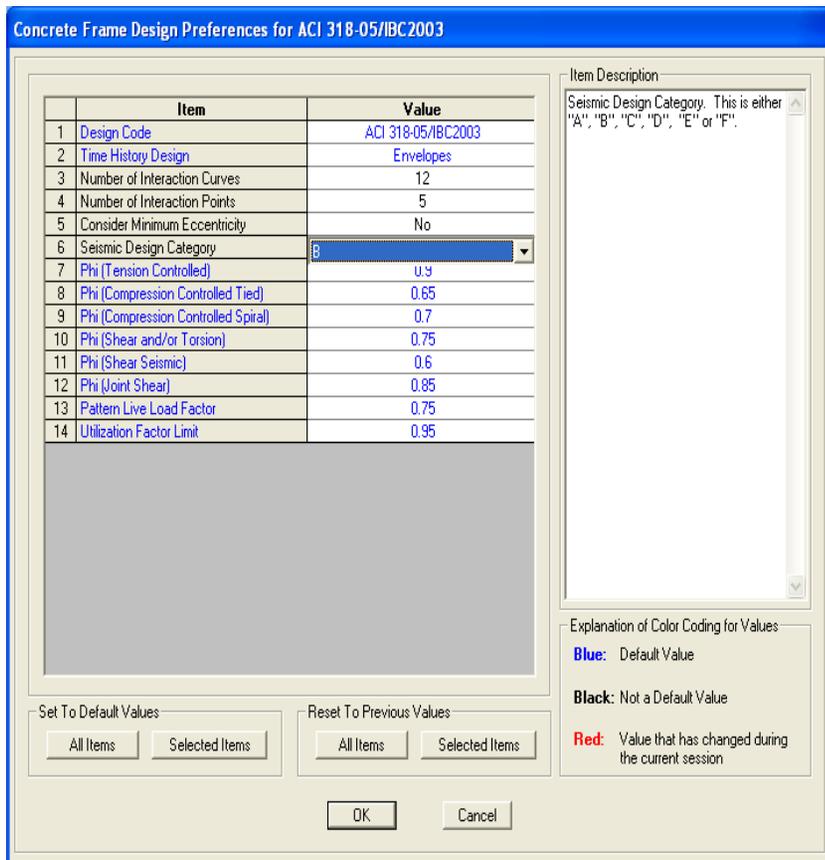


Use Display menu>Show tables to generate output tables and input summaries. You can select which load patterns and cases to include, which joints and elements you want to report and which reports you want to generate

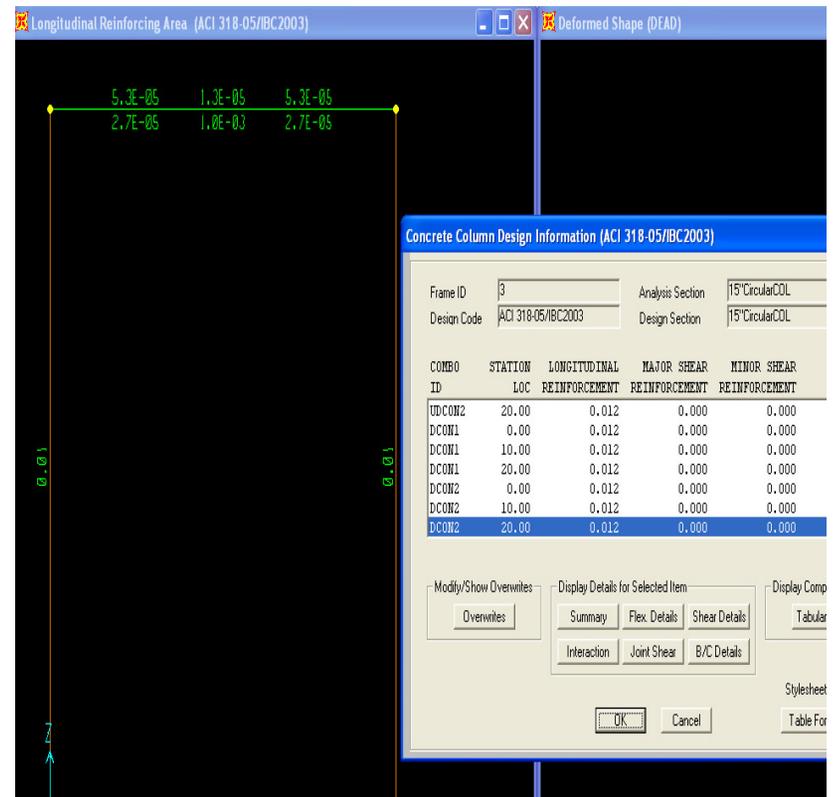
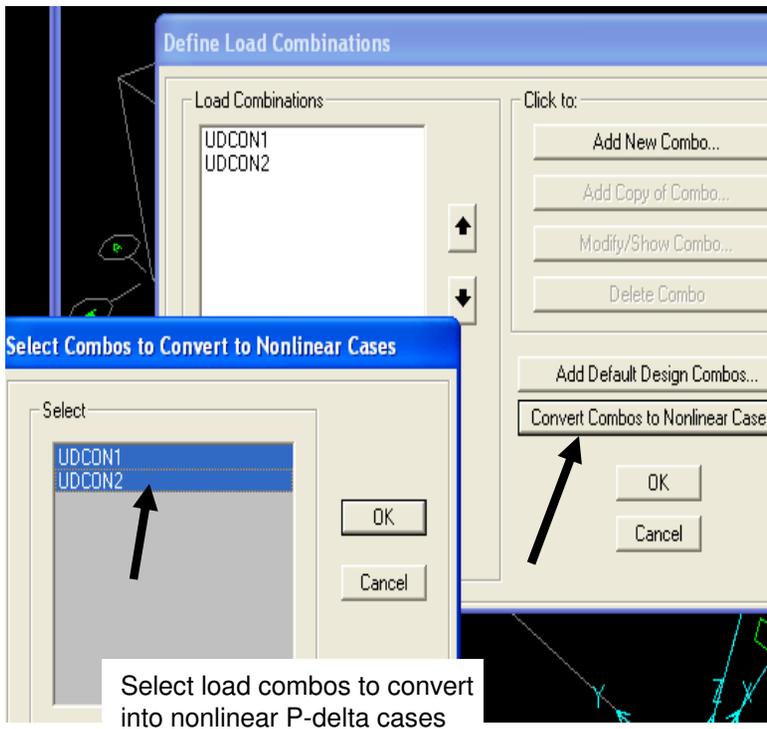




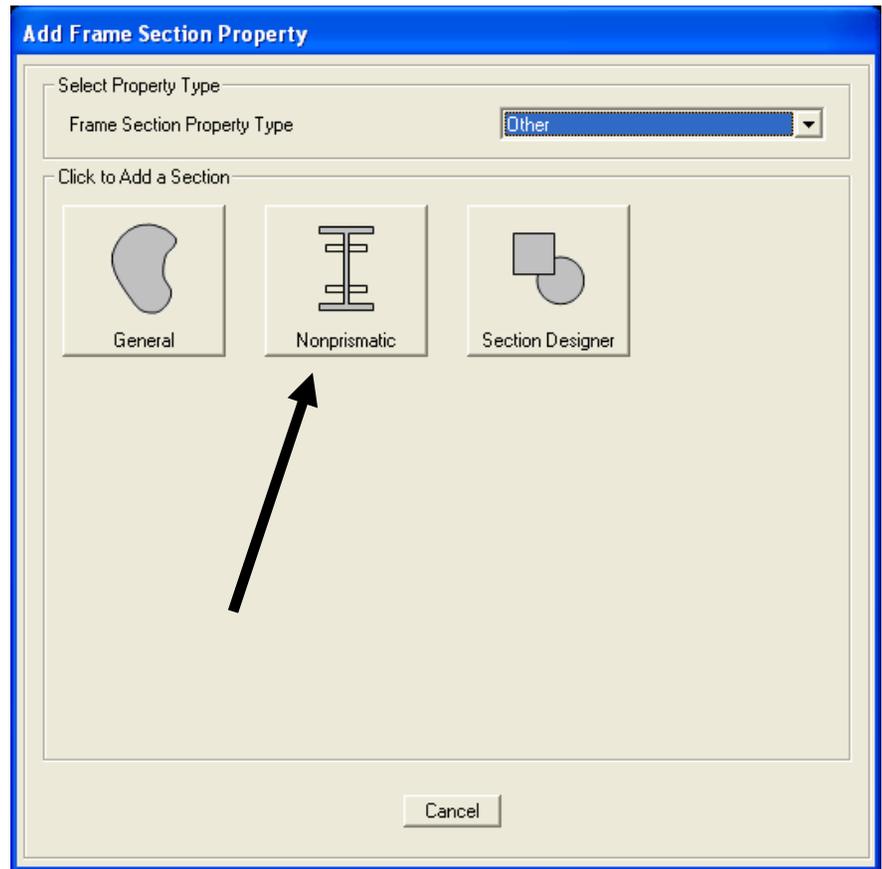
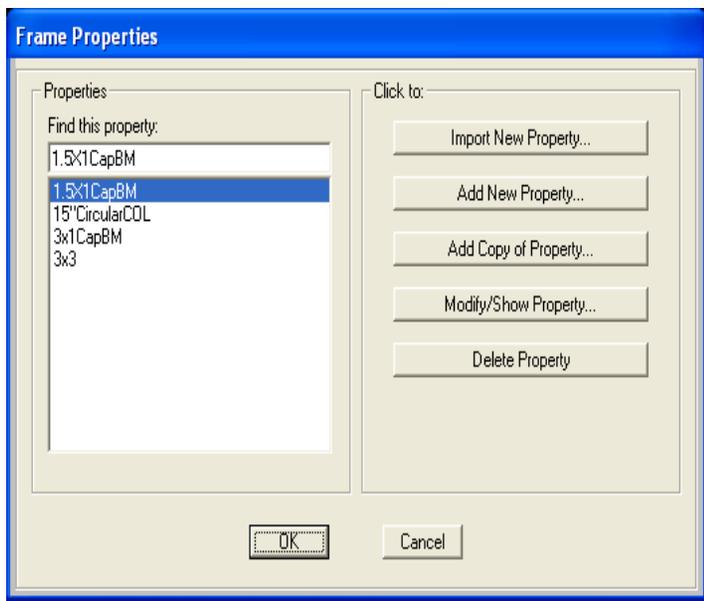
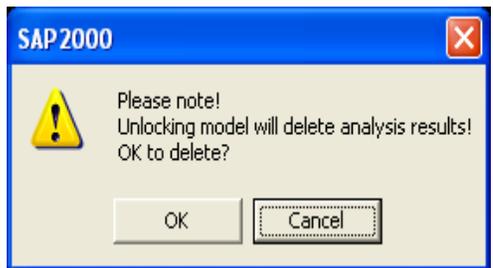
To perform an ACI 318-05 design check, go to Design menu>Concrete Frame design>View/Revise preferences to specify design parameters. Then go to Define menu>Load Combinations. Here you can define your own factored load combos, or let SAP2000 generate them automatically based on your selected design code.



As an alternative to defining P-delta cases one-at-a-time as we did before, SAP2000 offers an option to automatically convert load combos into nonlinear P-delta cases by clicking the 'Convert Combos to Nonlinear cases' button on the load combo screen, then select/highlight the combos to be converted. To do a concrete design, first run an analysis. After the analysis completes, then run a concrete design by clicking the 'Start concrete design/check' button  - After the design completes, you can right click individual frames for design results, or use the Design menu>Concrete frame design>Display design info, or use Display menu>Show tables to print and sort results



Close the output forms, unlock the model, by clicking the unlock button , say OK to the unlock warning. Next we're going to define a nonprismatic cap beam, so go to Define>Section properties>Frame sections, click 'Add new property', 'Other' frame section property type and click 'Nonprismatic'



Accept the default VAR1 section name and type the values shown below for sections, lengths and length type. SAP2000 lets you define the length type by variable percentage or absolute length using current units (ft in this example). Press OK twice to return to the model

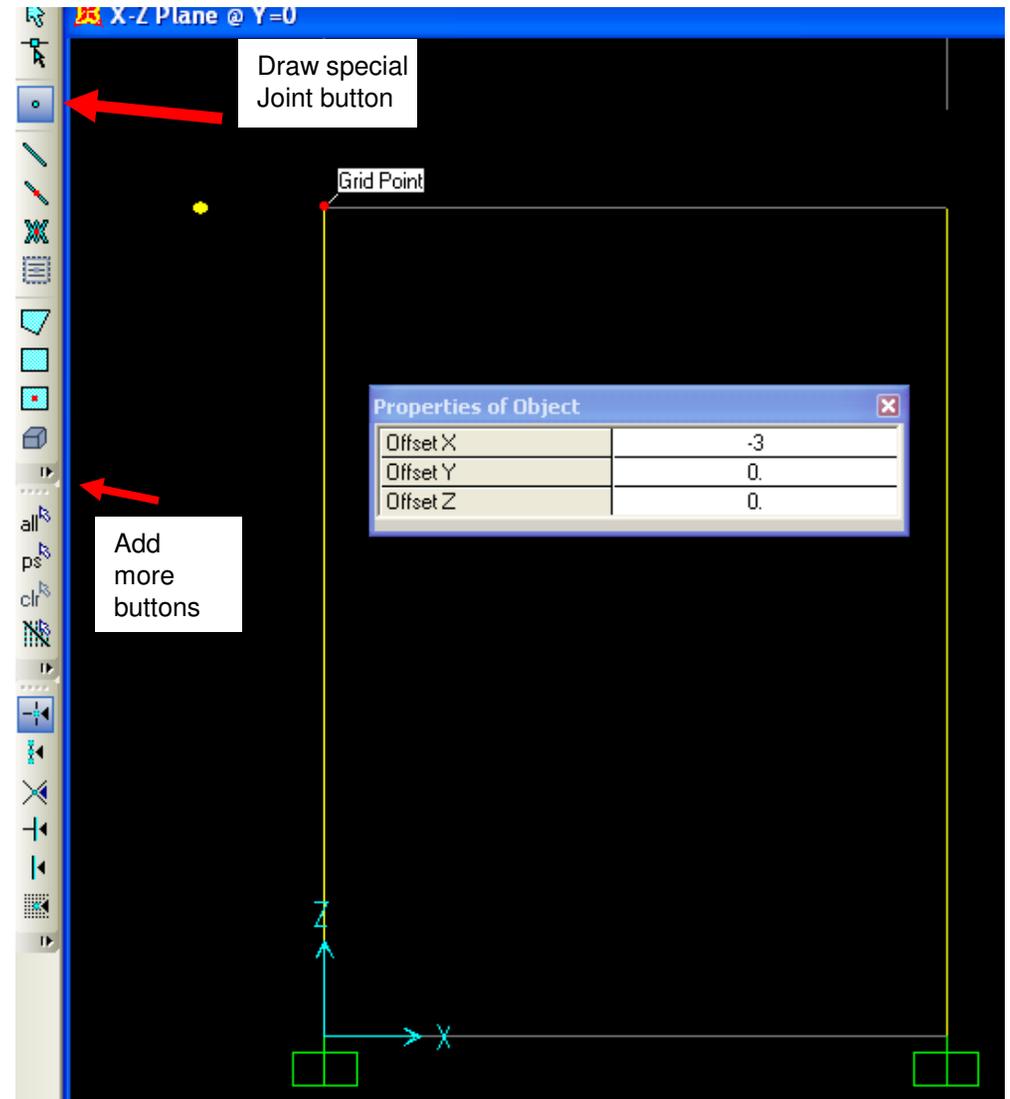
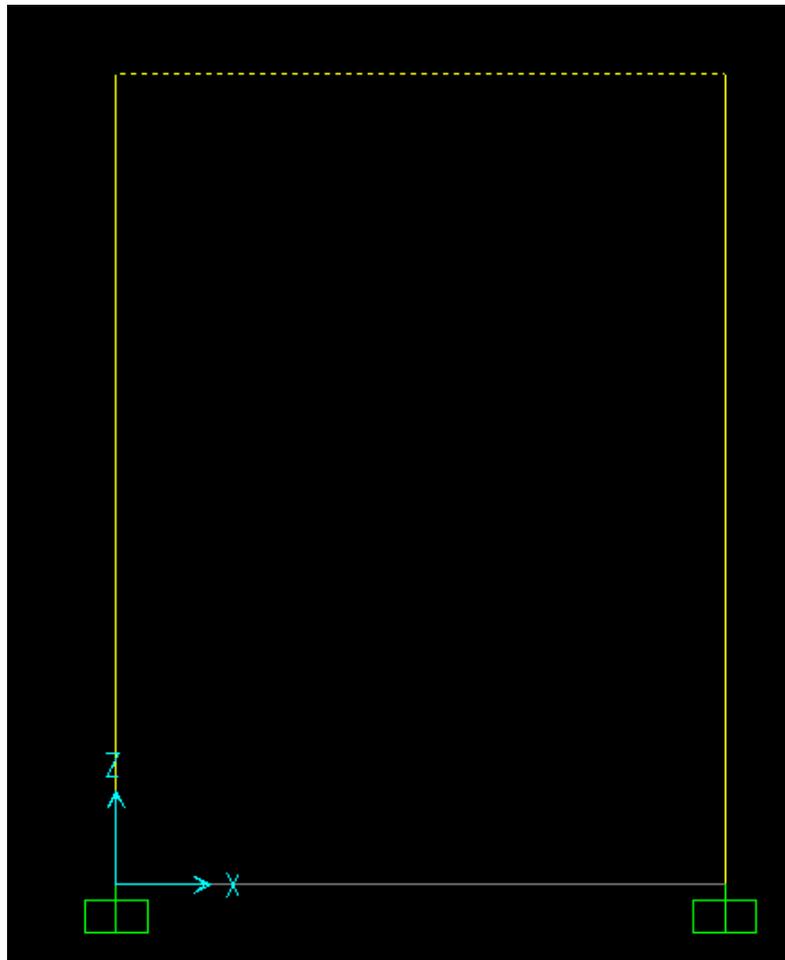
**Nonprismatic Section Definition**

**Nonprismatic Section Name**  **Display Color**

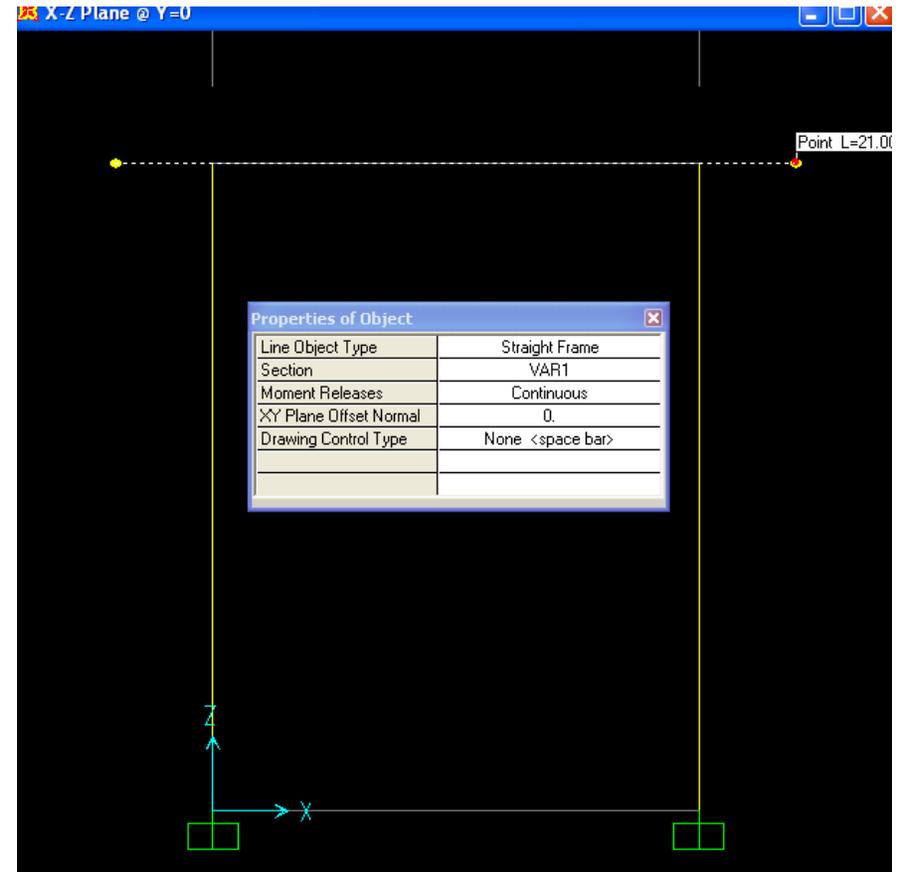
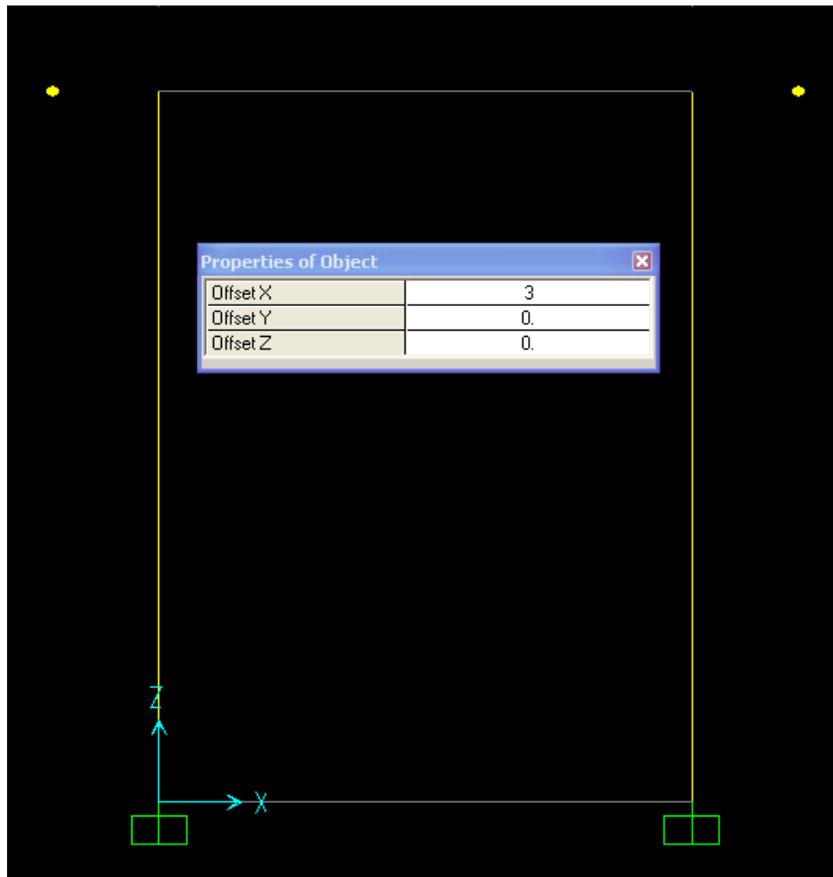
**Section Notes**

Start Section	End Section	Length	Length Type	EI33 Variation	EI22 Variation
3x1CapBM	1.5x1CapBM	3	Absolute	Parabolic	Linear
1.5x1CapBM	3x1CapBM	3	Absolute	Parabolic	Linear
3x1CapBM	3x1CapBM	15	Absolute	Parabolic	Linear
3x1CapBM	1.5x1CapBM	3	Absolute	Parabolic	Linear

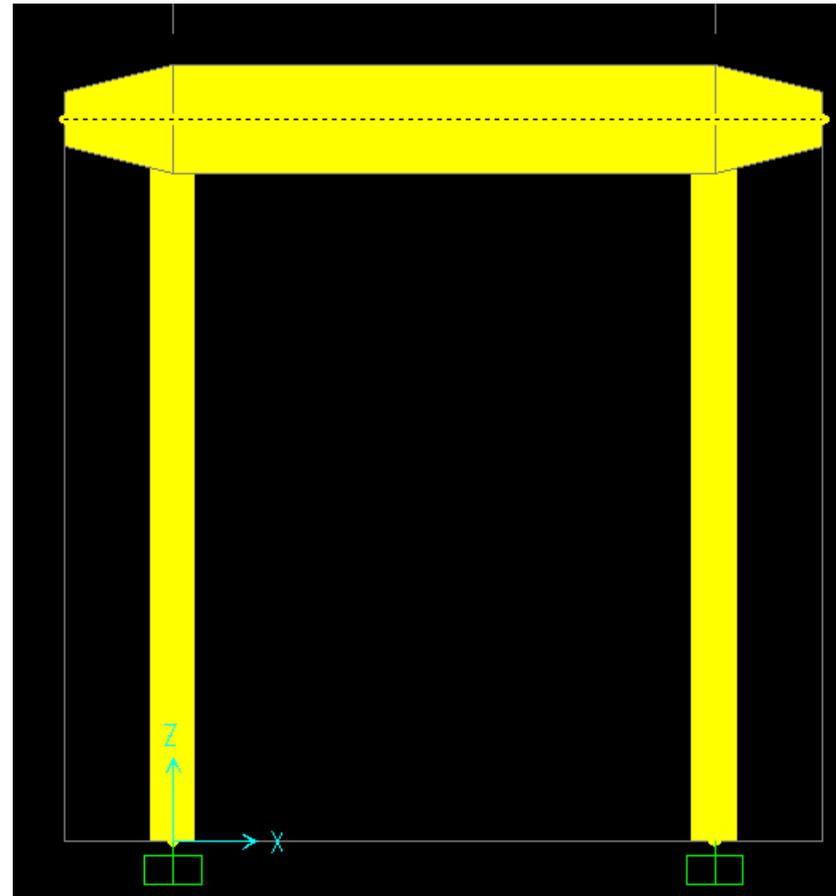
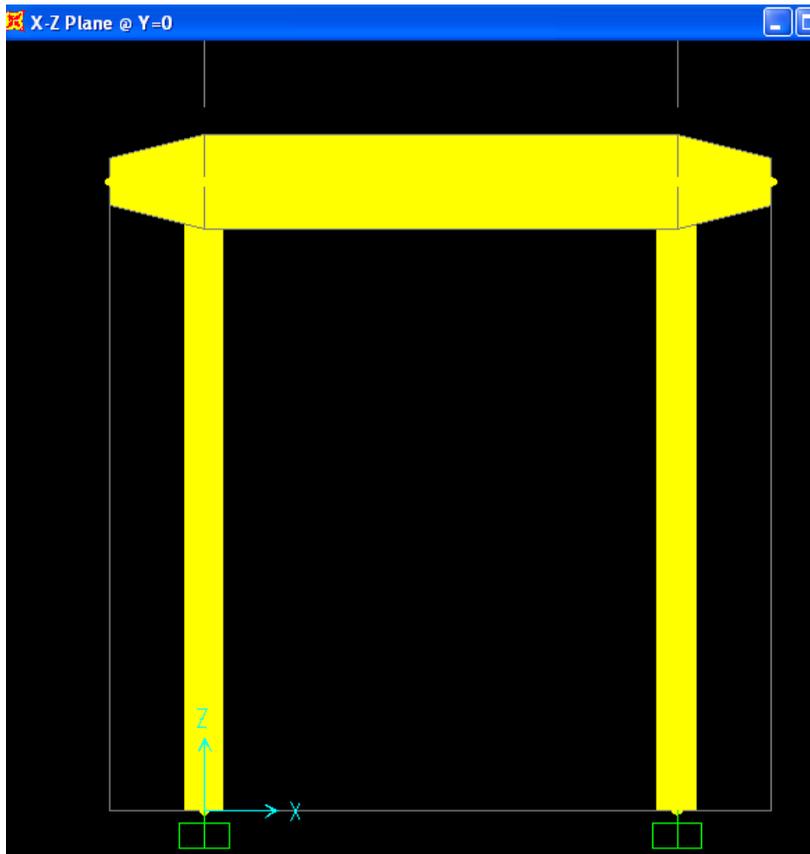
Select the existing beam and press the Del (delete) key on your keyboard. Next, click the 'Draw special joint' button, type -3(feet) in the X direction and click the top of the left side column to draw a joint 3 feet away as shown. You may or may not have to add the Draw special joint button to your palette using the 'More buttons' arrow



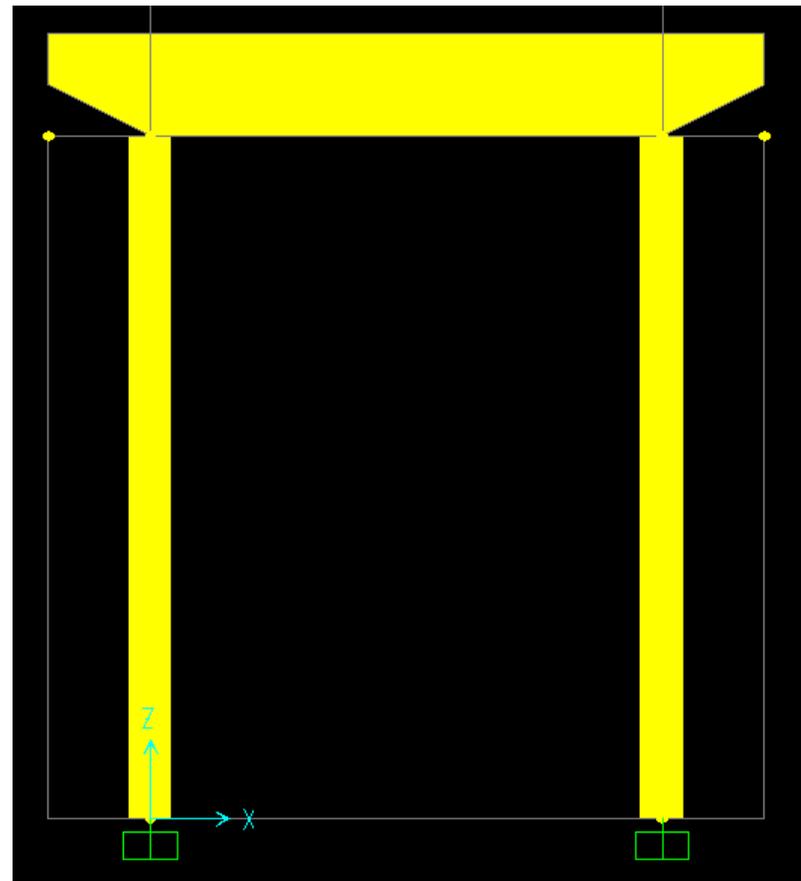
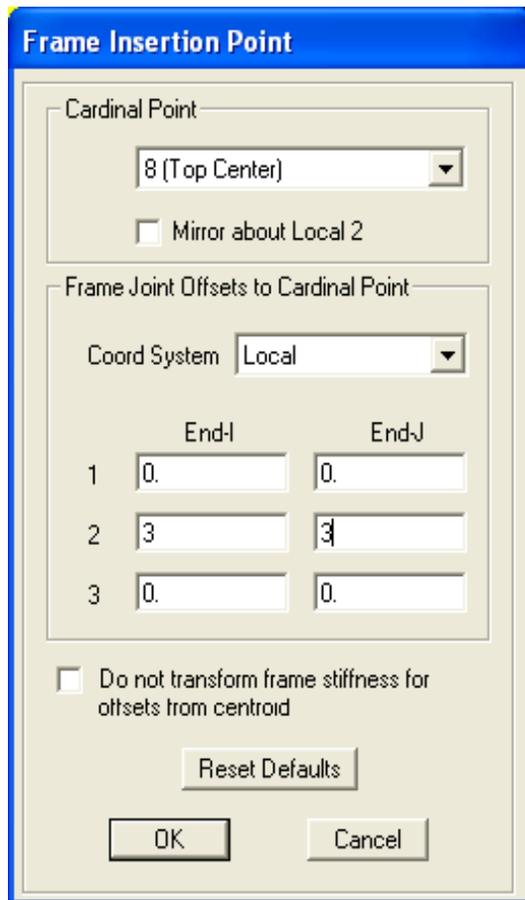
Next, change offset dimension to (+) 3 feet in the X direction and click the top of the right column to add another joint. Next, click 'Draw frame/cable element' button , change section to VAR1 with continuous connection and connect the dots by clicking once on the special joint on the left and then complete the beam by clicking on the special joint on the right as shown. SAP2000's object based modeling will automatically connect the columns to the beam with no further meshing required. Special joints with snap tools can also be used to add joints on frames to assign joint loads without manually dividing the frame



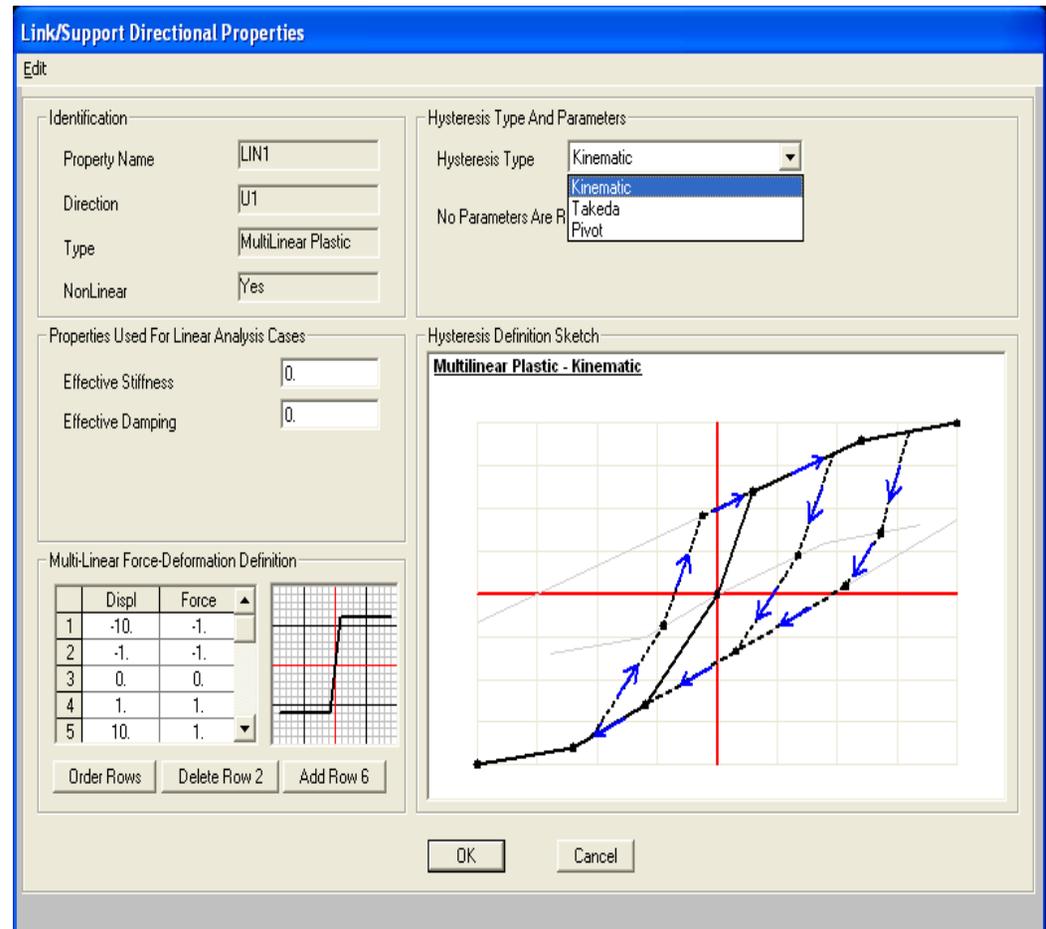
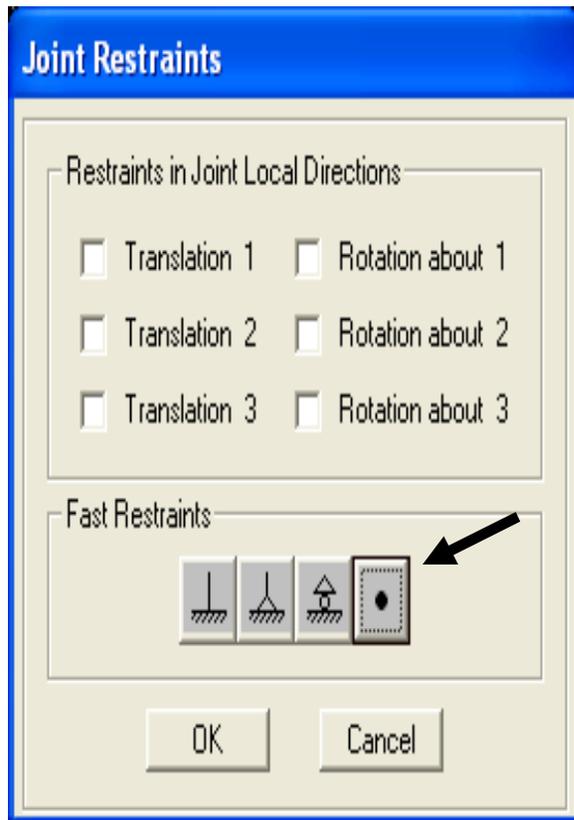
Press Esc key or click select arrow button  to exit draw mode. Click the 'Set display options' menu and checkbox extrude view for the planar view. Hold down your left mouse key and drag right to left while intersecting the beam in order to select



Next, use the main menu and go to Assign>Frame>Insertion point. Then press the F1 key on your keyboard to read more about this feature, which lets you assign cardinal point offsets + additional offsets. Please select cardinal point 8 (top center) with additional 3 ft. joint offsets in the local 2 direction as shown. Local 2 direction = Z in global coordinate system for this beam. Press OK



To delete restraints, you need to select the joints where restraints are assigned by windowing around the base of the bent structure, then Assign>Joint>Restrains and click the black dot to assign a null which deletes the restraints. Next, go to Define menu>Section properties>Link/support properties where you can define other supports known as “links”, which can be 1 point or 2 point links. Click ‘Add new property’ button to review options. Below right is the multi-linear plastic spring which is commonly used for nonlinear pile/soil interaction to define P-Y and T-Z curves



There are also nonlinear link property options for gap springs, base isolators (friction bearing and elastomer), nonlinear viscous damper with velocity exponent with linear options to assign dashpots C and linear springs K. Under define menu you will also see the option to define frequency dependent K and C for dynamic analysis, as geotechnical data may report soil properties as a function of frequency. Links can be assigned as point supports, or on a per lineal unit basis on frame elements, or per area basis for shells and solid finite elements. In this manner, you can easily model piles explicitly using circular or rectangular frame elements, then assign the link as a frame spring/link assignment.

### Link/Support Directional Properties

Identification

Property Name: LIN1

Direction: U1

Type: T/C Friction Isolator

NonLinear: Yes

Properties Used For Linear Analysis Cases

Effective Stiffness: 0.

Effective Damping: 0.

Properties Used For Nonlinear Analysis Cases

Stiffness for Compression: 0.

Stiffness for Tension: 0.

Gap Opening for Compression: 0.

Gap Opening for Tension: 0.

Damping Coefficient: 0.

OK Cancel

### Frequency Dependent Link Property Data

Edit

Identification

Property Name: FREQD1

Section Notes:

Display Color:

Degrees of Freedom with Nonzero Properties Shown Checked

U1  U1U2  U1U3  U1R1  U1R2  U1R3

U2  U2U3  U2R1  U2R2  U2R3

U3  U3R1  U3R2  U3R3

R1  R1R2  R1R3

R2  R2R3

R3

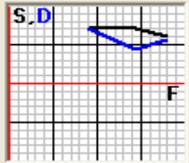
Symmetrical

Show Properties for this Degree of Freedom

U1

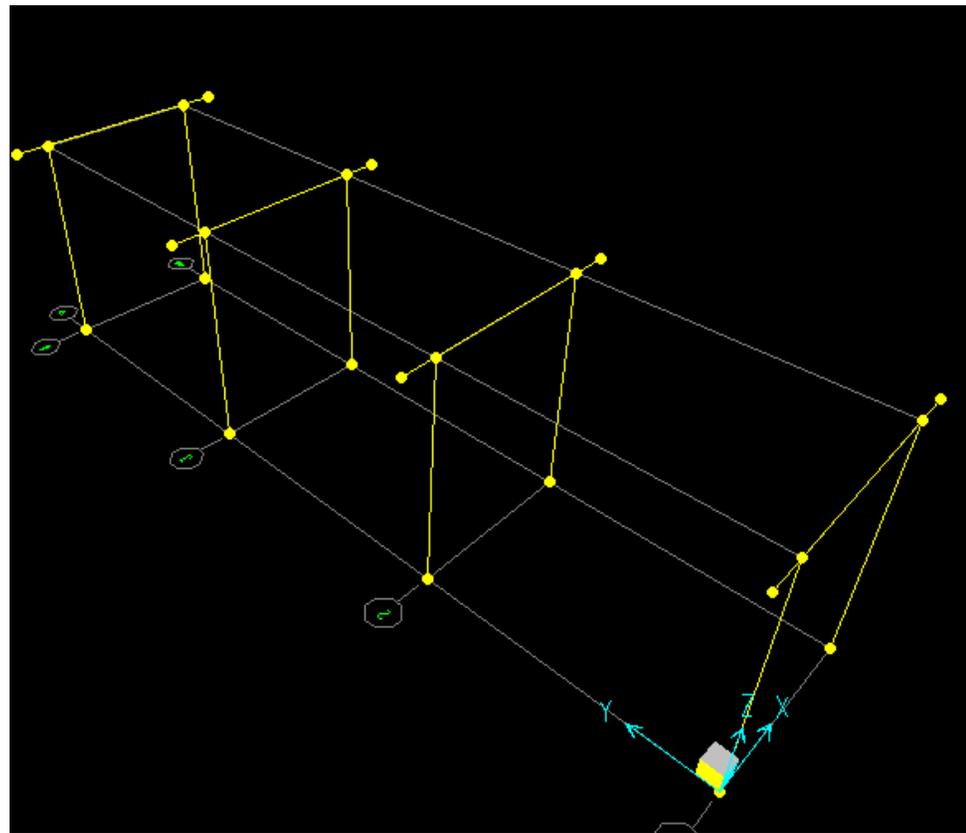
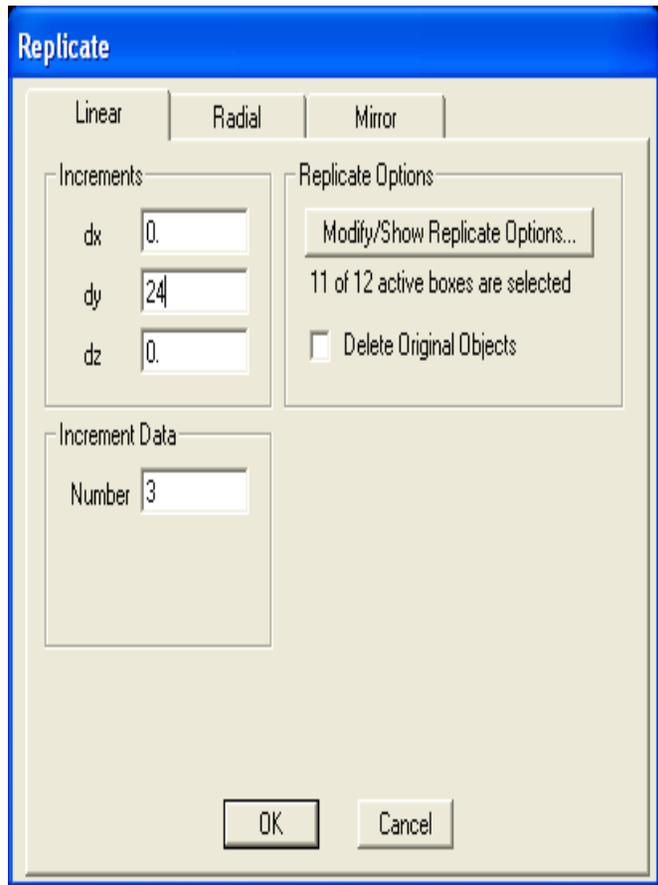
Properties for Degree of Freedom U1

	Frequency	Stiffness (Re)	Damping (Im)
	Hz	Kip/in	Kip/in
1	25.	0.15	0.19
2	40.	0.15	0.12
3	50.	0.13	0.15

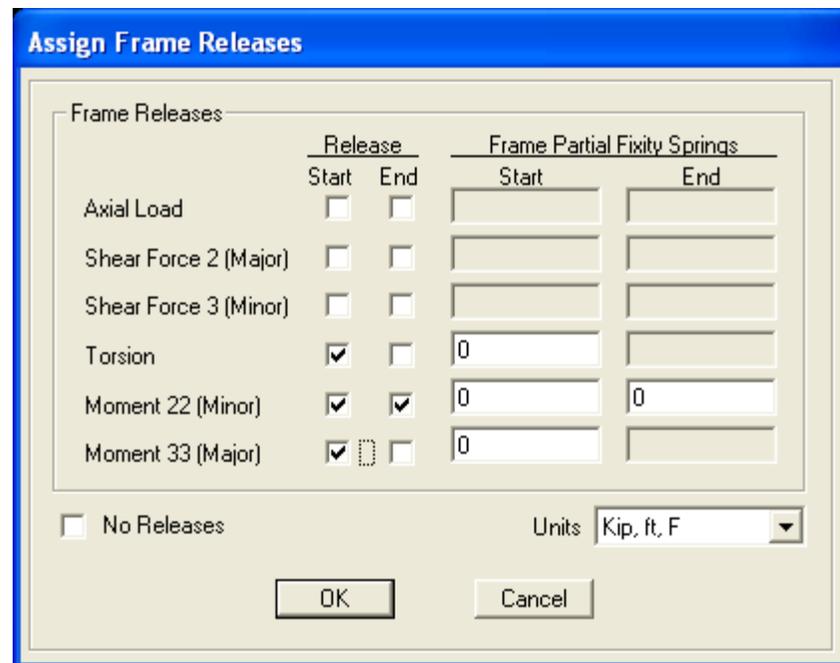
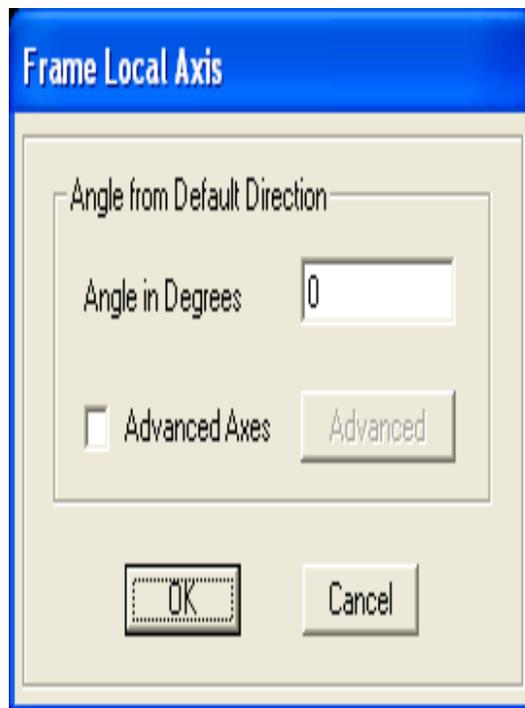


OK Cancel

We'll end the tutorial here by demonstrating two more options: Replicate commands, rotating frame local axis, and assignment of shear and moment releases. Check the bottom right portion of your screen to make sure you're in Kip-ft units, click the 'Select all' button <sup>all</sup> to select and then Edit>Replicate. Here you can copy and paste in linear, radial or mirror fashion. If you want to rotate all or a portion of the entire structure, you do it with Replicate, just checkbox the 'Delete Original Object' box



To rotate local axis of frames or to assign moment and/or shear releases, first select frames by clicking them or using one of the Select menu options, then Assign>Frame>Local axes and type an angle in order to rotate local axes. If you want to assign releases, choose the Assign>Frame>Releases/Partial fixity option shown below on the right



There are several good Watch & Learn video tutorials on our website here [http://www.csiberkeley.com/Support\\_WL\\_SAP.html](http://www.csiberkeley.com/Support_WL_SAP.html) which cover bridge modeling and other features of SAP2000. These videos are good tools to help you get started while minimizing the learning curve. Also, the SAP2000 Help menu enables you to access user manuals and other documentation.

Please feel free to contact me if you have feedback or suggestions regarding this tutorial or if you have SAP2000 modeling questions. I'm here to help new users get started:

**Darrell Foster**

CSI Houston, TX office

Tel: 713-533-4420

**Computers & Structures, Inc.**

TECHNOLOGY FOR A BETTER WORLD

[darrell@csiberkeley.com](mailto:darrell@csiberkeley.com)

[www.csiberkeley.com](http://www.csiberkeley.com)