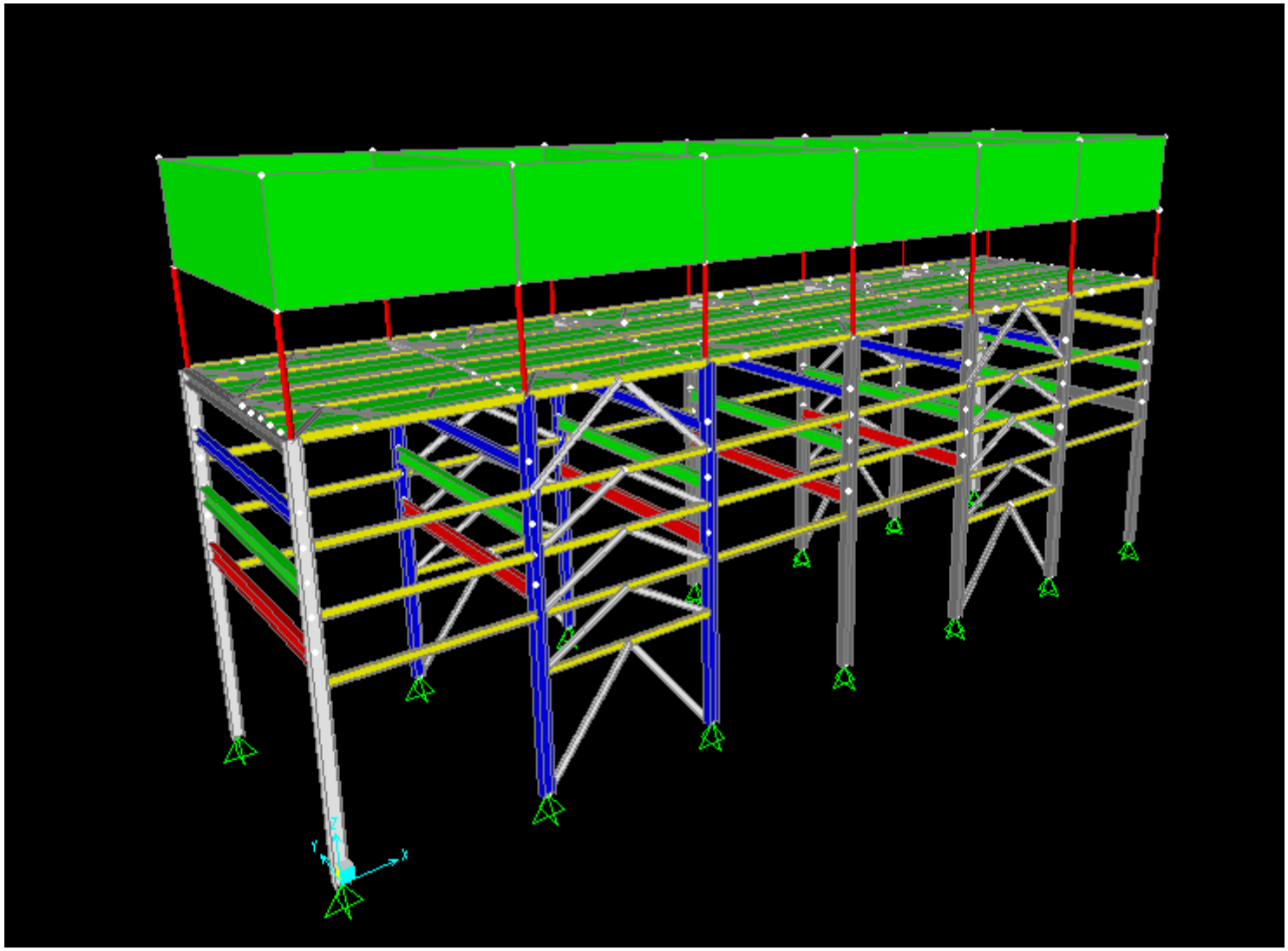
Training Guide

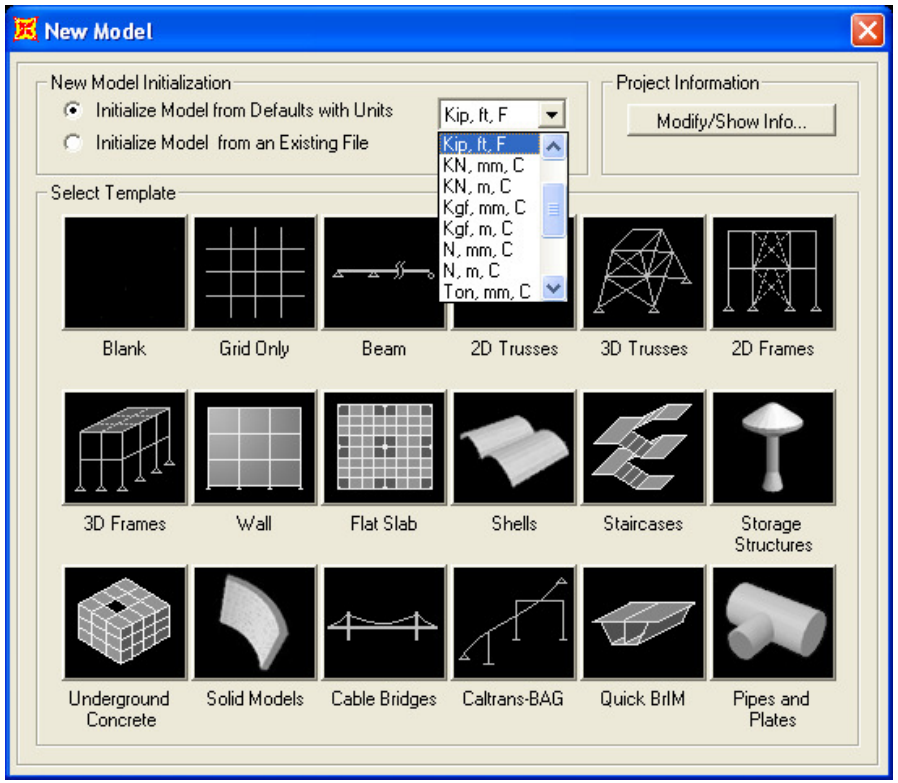
Modeling, Analysis, and Design of Pipe Rack Structures Using SAP2000

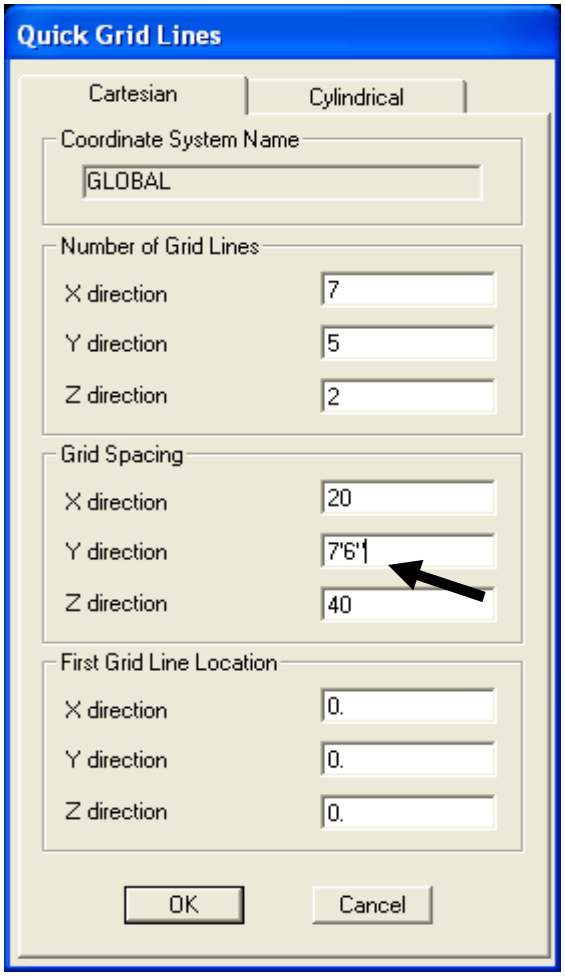
SAP2000

Using the attached pdf drawings (PR1.pdf and PR2.pdf), this tutorial will show you how to model, analyze and design the 6 bay pipe rack structure shown below supporting air cooled heat exchangers. This is an extruded view of the structure, color coded by sections.

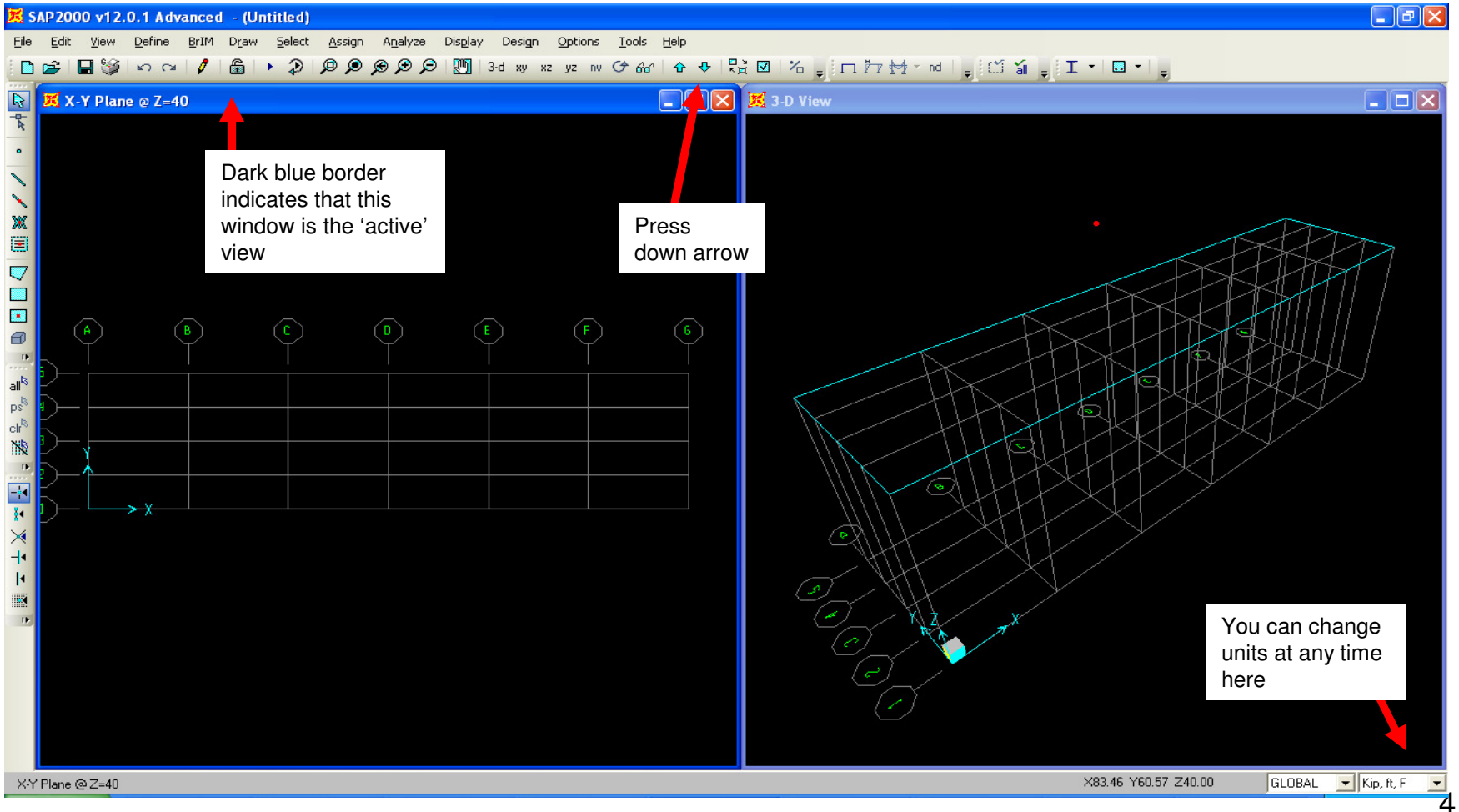


Click new model button or **File>New** to begin. The first screen that appears enables you to select units and offers a number of template options. In most cases you will want to define grids. Select Kip, ft, F units and click 'Grid only'. Input uniform spaced grid lines and spacing shown below right, including the 7'6" architectural units in Y direction grid spacing then press OK. You can input using architectural units and tab or press OK for SAP2000 to convert to current units

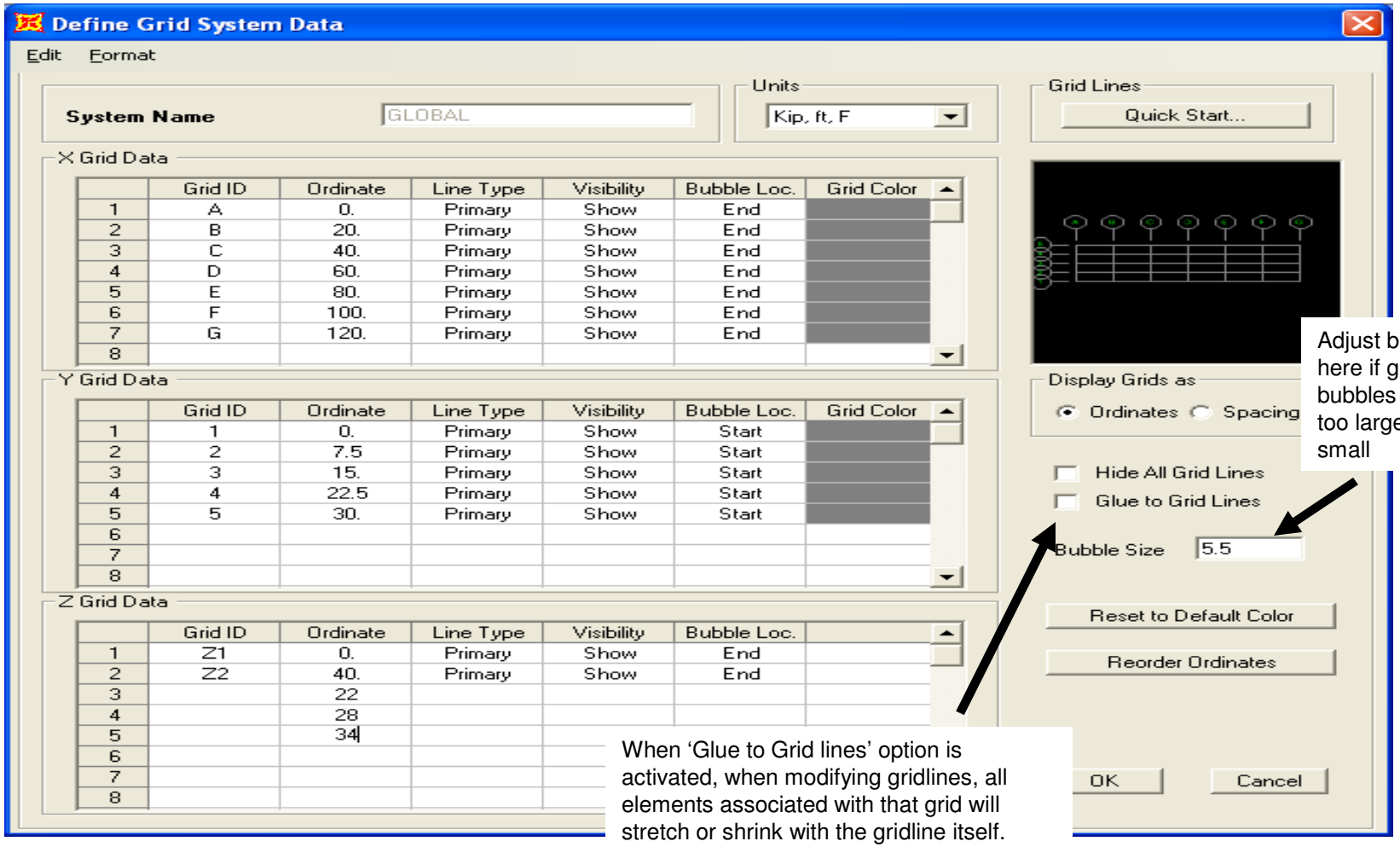




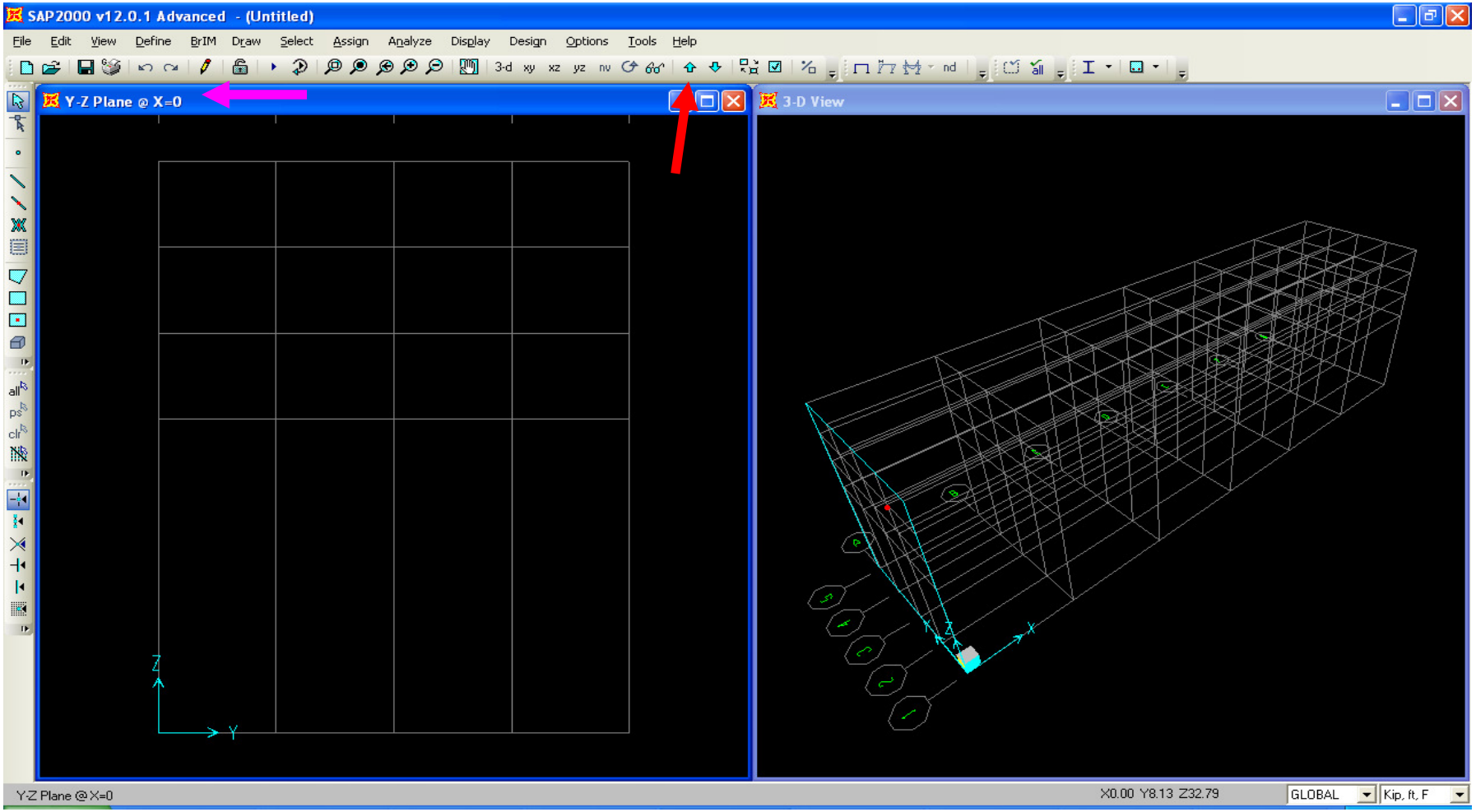
By default, you will see a split-screen view. If you click the planar view on the left, you will see a blue outline on the 3D model highlighting the planar view location. Press the down arrow to move to plan view Z = 0



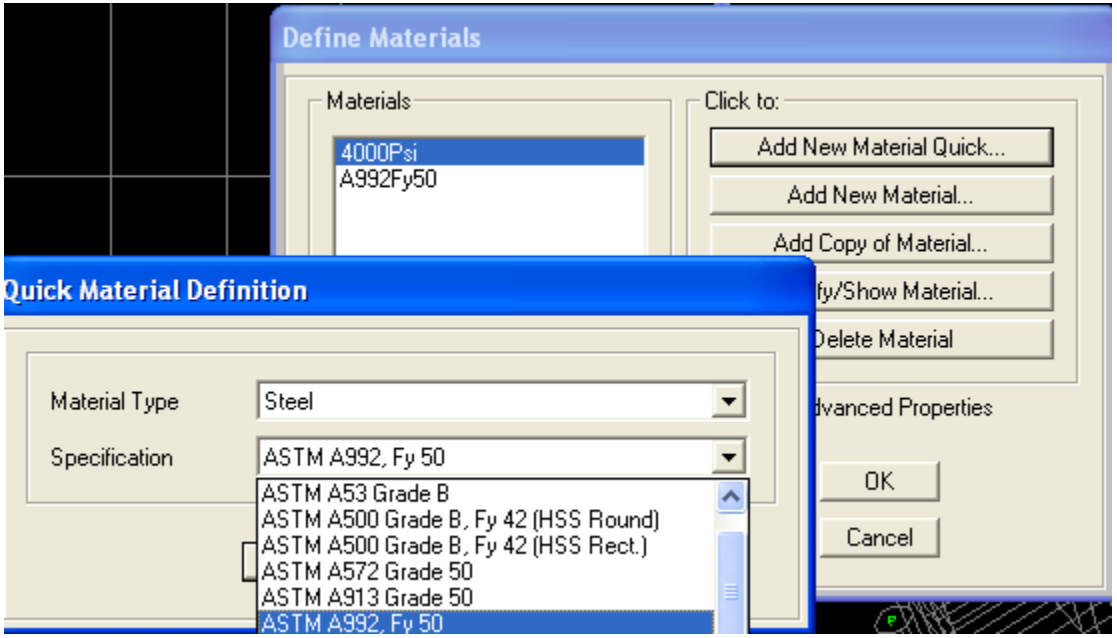
Next, double click any gridline with your mouse to display the Define grid screen below. Type in values in Z grid direction as shown below for main beam locations shown on the PR2.pdf drawing and press OK



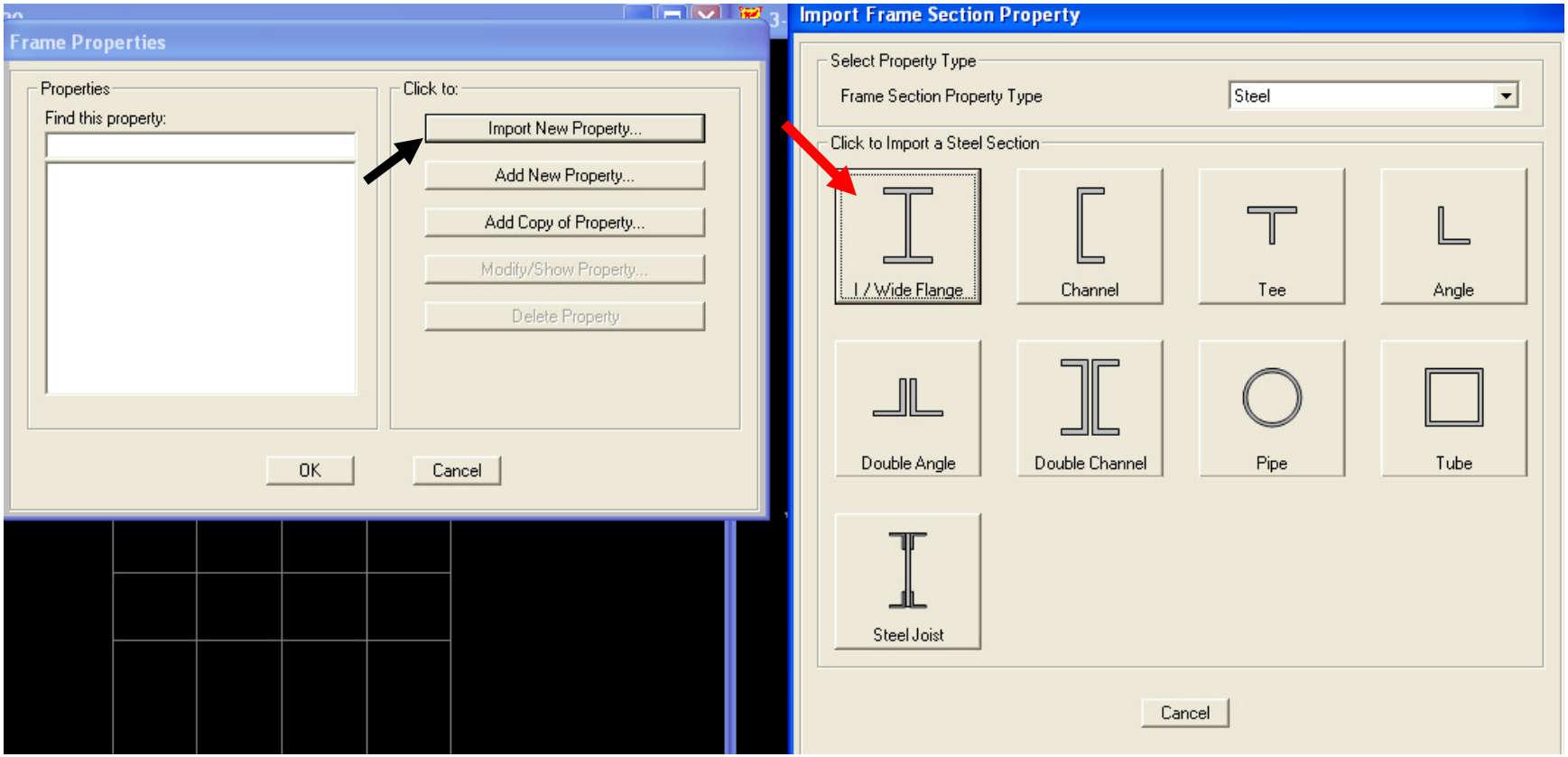
Click yz button to switch planar view to elevation and press arrows to move to X = 0 Y-Z plane as shown below



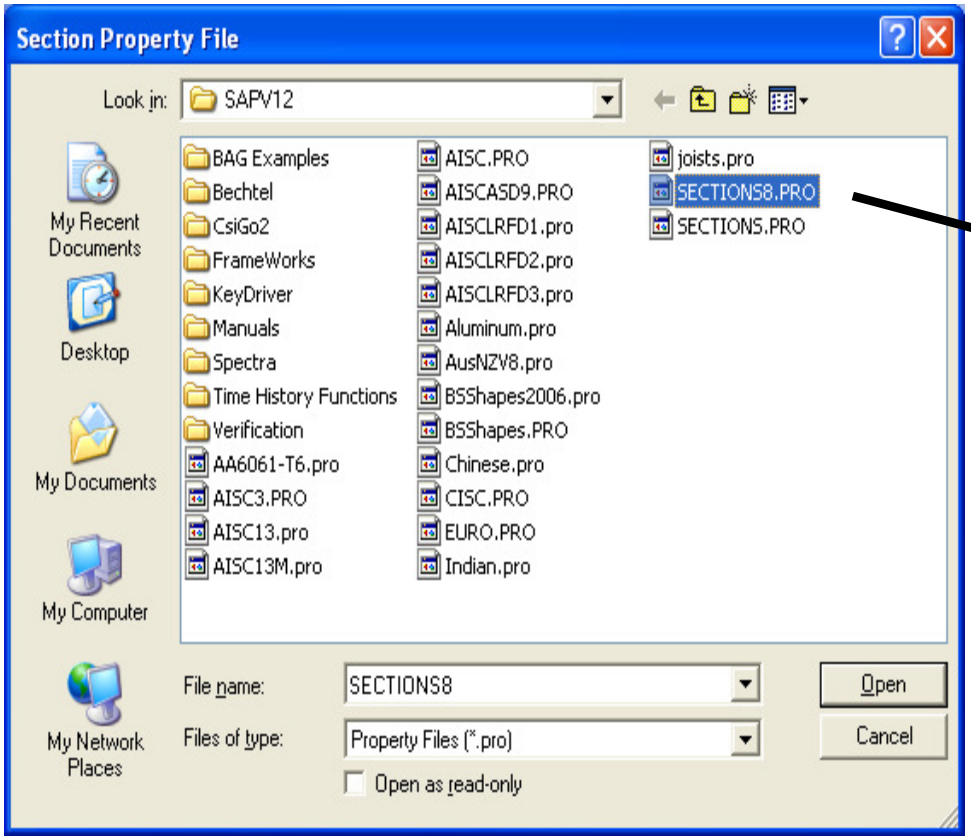
Before we import our sections, use the main menu to review the **Define>Materials** where we see two default materials, 4000 psi concrete material and A992 steel material. You can add new materials using the 'Add new material quick' button to select from a material library, or use the 'Add New Material' button to add a material not found in the libraries. In a concrete structure, typically multiple concrete materials need to be used for different strengths and weights.

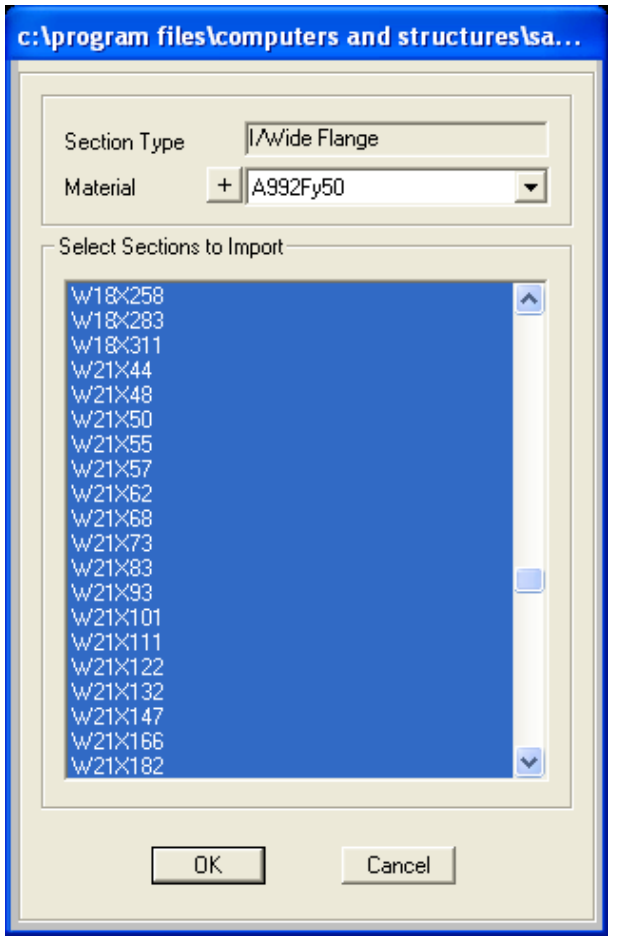


Cancel out of the Define Materials dialogue to accept the default materials, then go to **Define>Section properties>Frame sections** where you will see the screen on the bottom left. For standard steel sections and joists, you use the 'Import New Property' button. First, click *I/Wide Flange* button to import those sections.

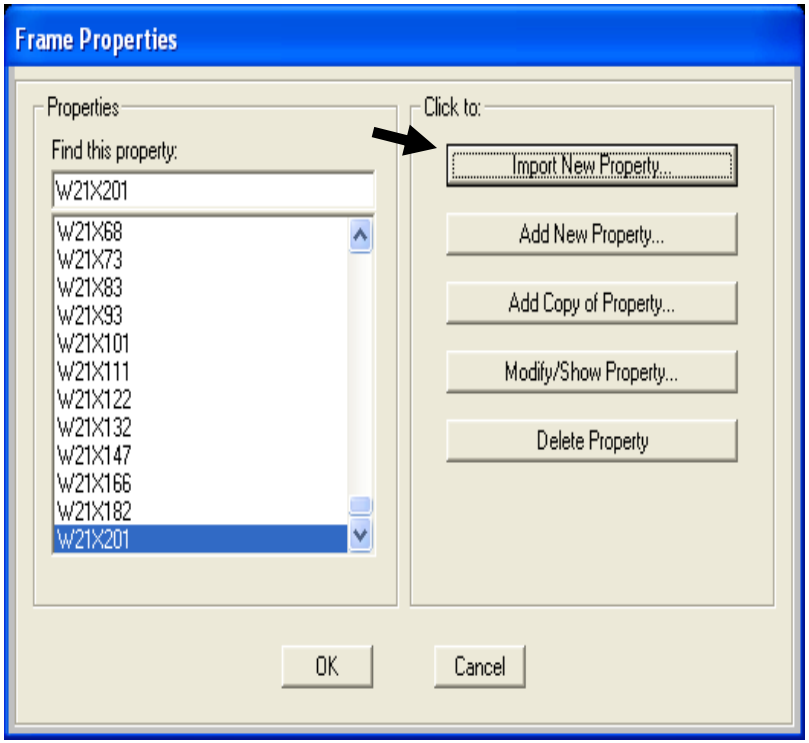


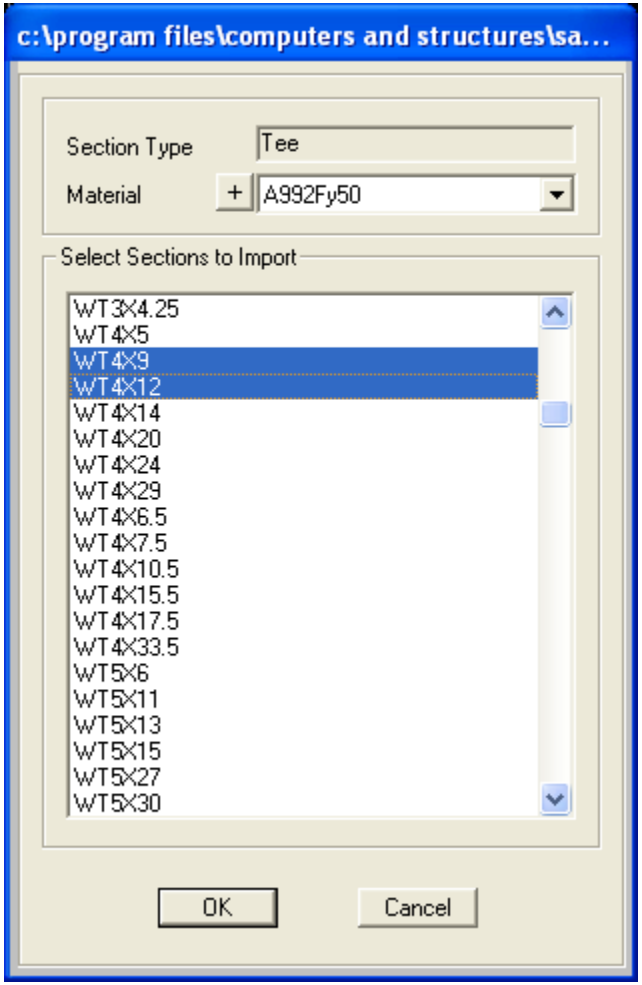
In order to utilize the latest AISC sections, go to the directory where SAP2000 is installed and click the SECTIONS8.PRO file. As you can see, SAP2000 offers libraries for several international steel shapes, as well as aluminum shapes and a joist library. After clicking to open this file, you will see a dialogue like the one on the bottom right. Use your mouse and drag to select all W sections from W8 through W21's and press OK



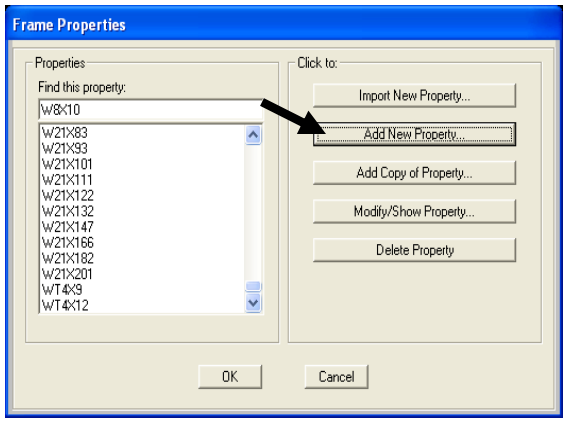


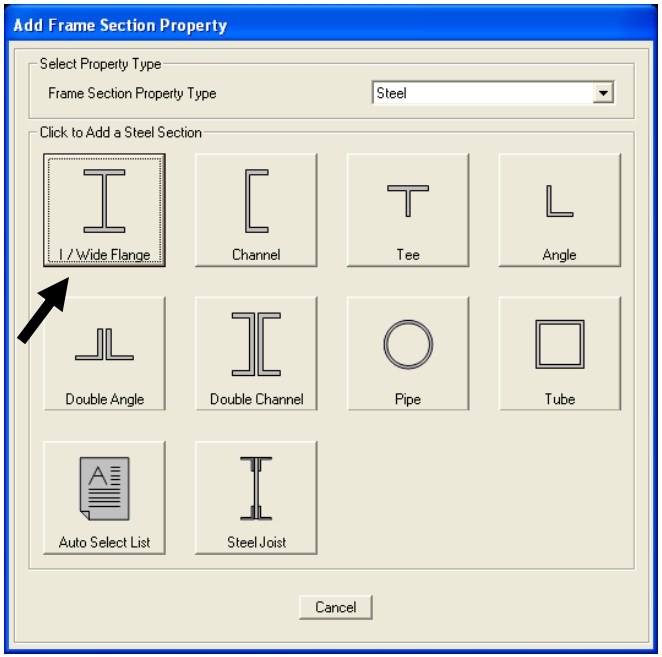
Press OK again to add these sections to the working list. Next, click ‘Import New Property’ button and click the ‘Tee’ button to view the Tee sections, select WT4X9 and WT4X12 and press OK twice to add these sections to the working list.

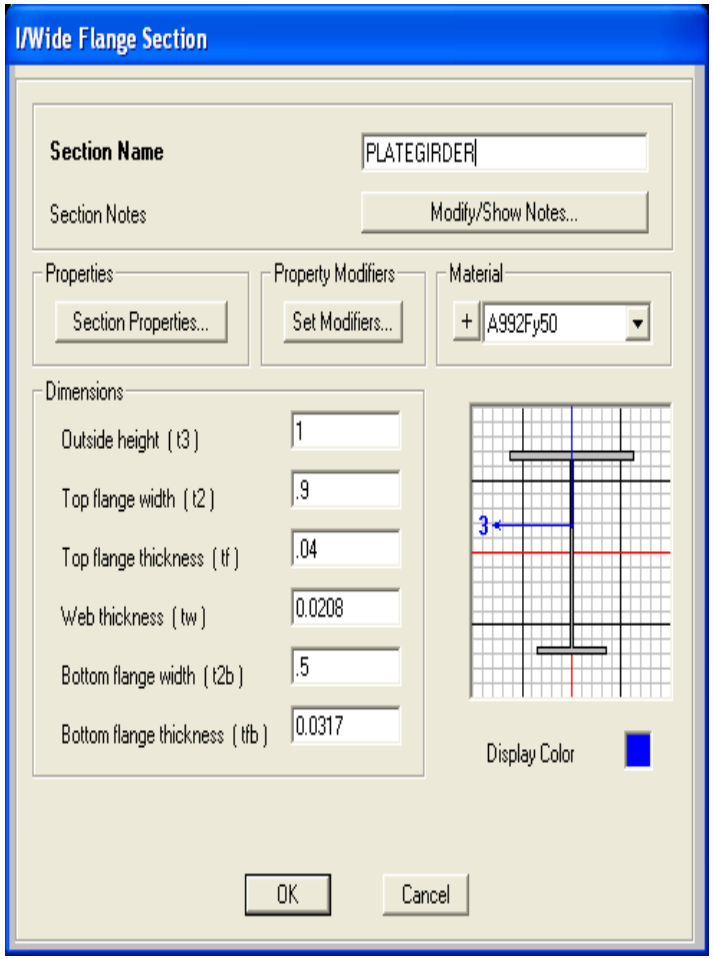




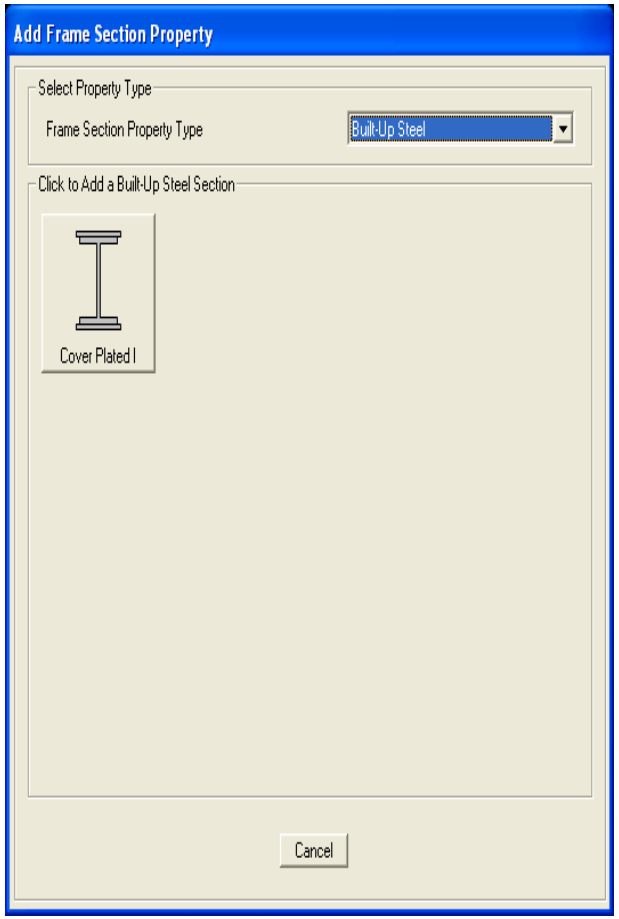
As a learning exercise, click ‘Add New Property’ button, then click I/Wide Flange. As you can see on the right, you can change dimensions to define a nonstandard steel section and even change material to concrete. Use of the Add options are needed to add/define concrete sections, nonstandard steel sections, nonprismatic steel shapes, and auto-select lists. Cancel out of this Add I/Wide Flange screen

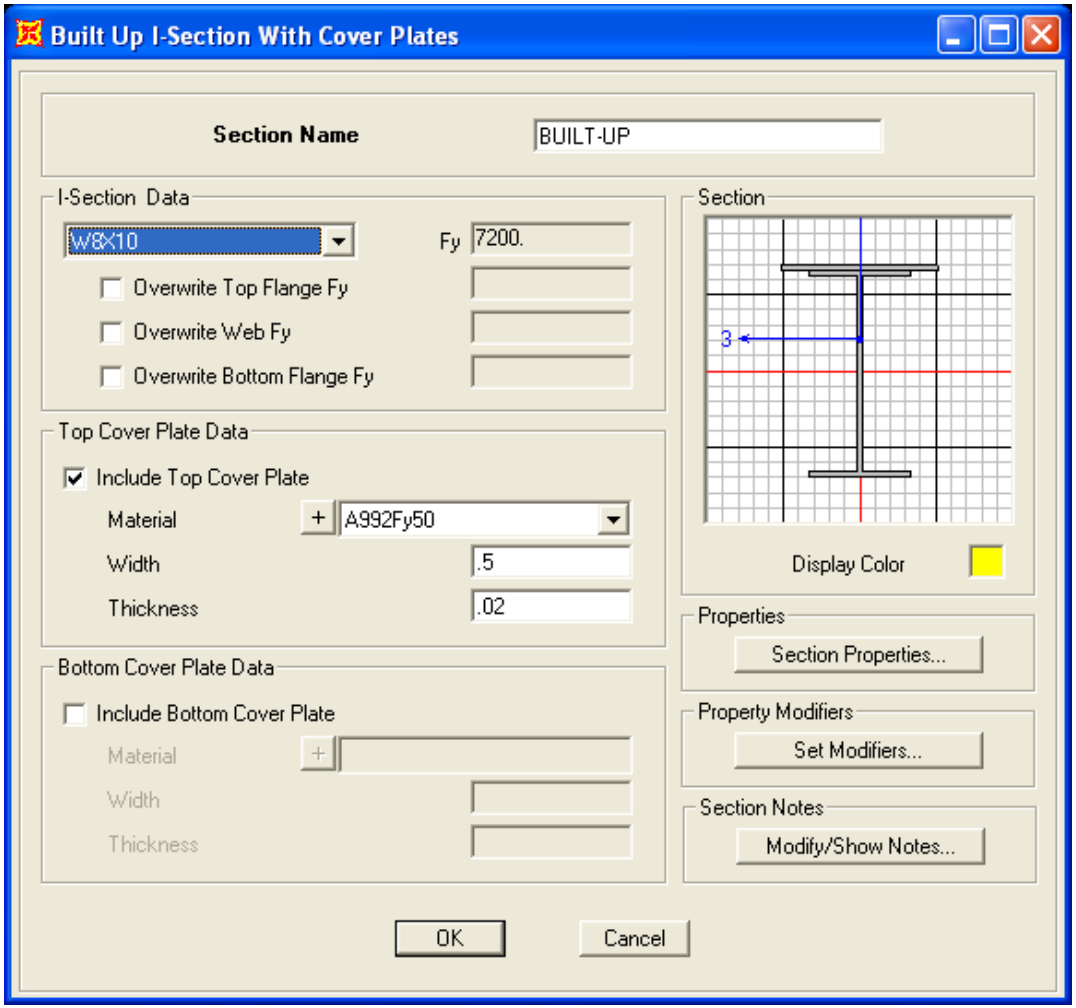




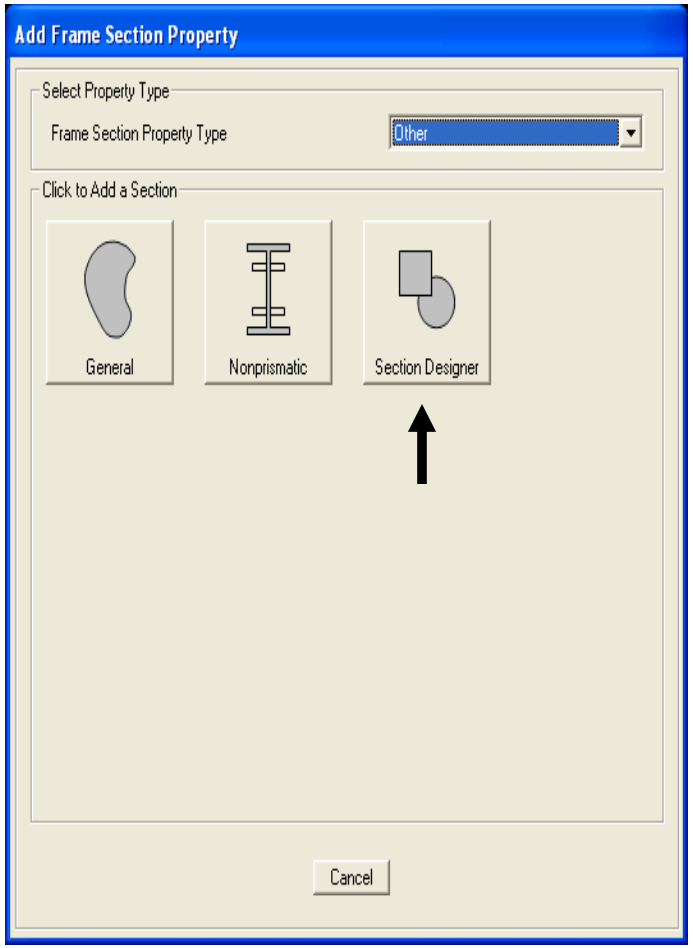


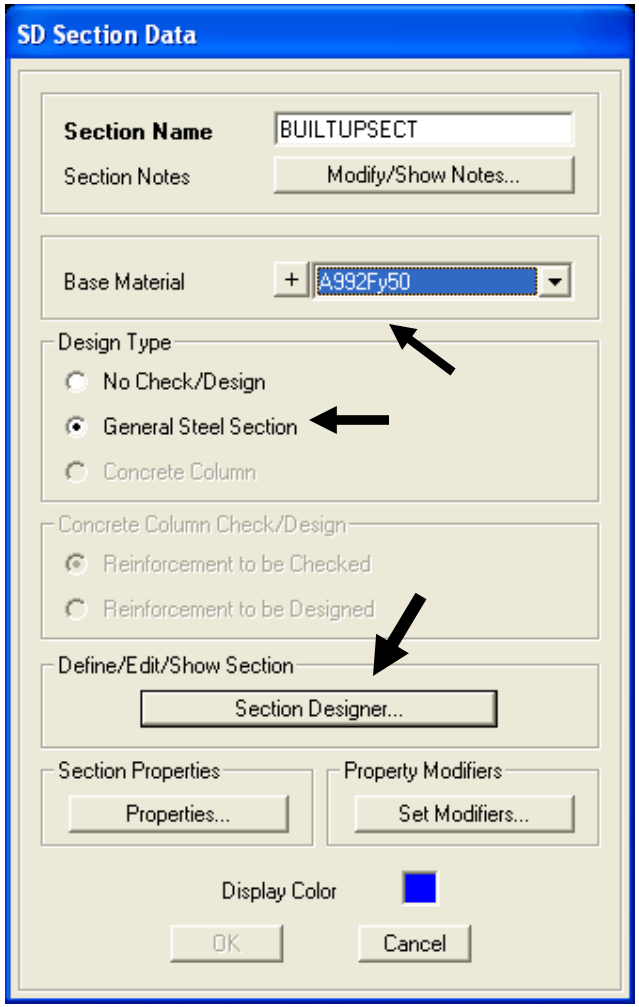
As another learning exercise, click ‘Add new property’ again, change Frame Section Property type to ‘Built-Up Steel’, can click Cover Plated I as shown below. Here you can quickly add top and/or bottom cover plates to Wide flange sections which are used for design as well as analysis. Cancel this as we are using only standard sections for this exercise, although it’s useful to know this tool is available for design.



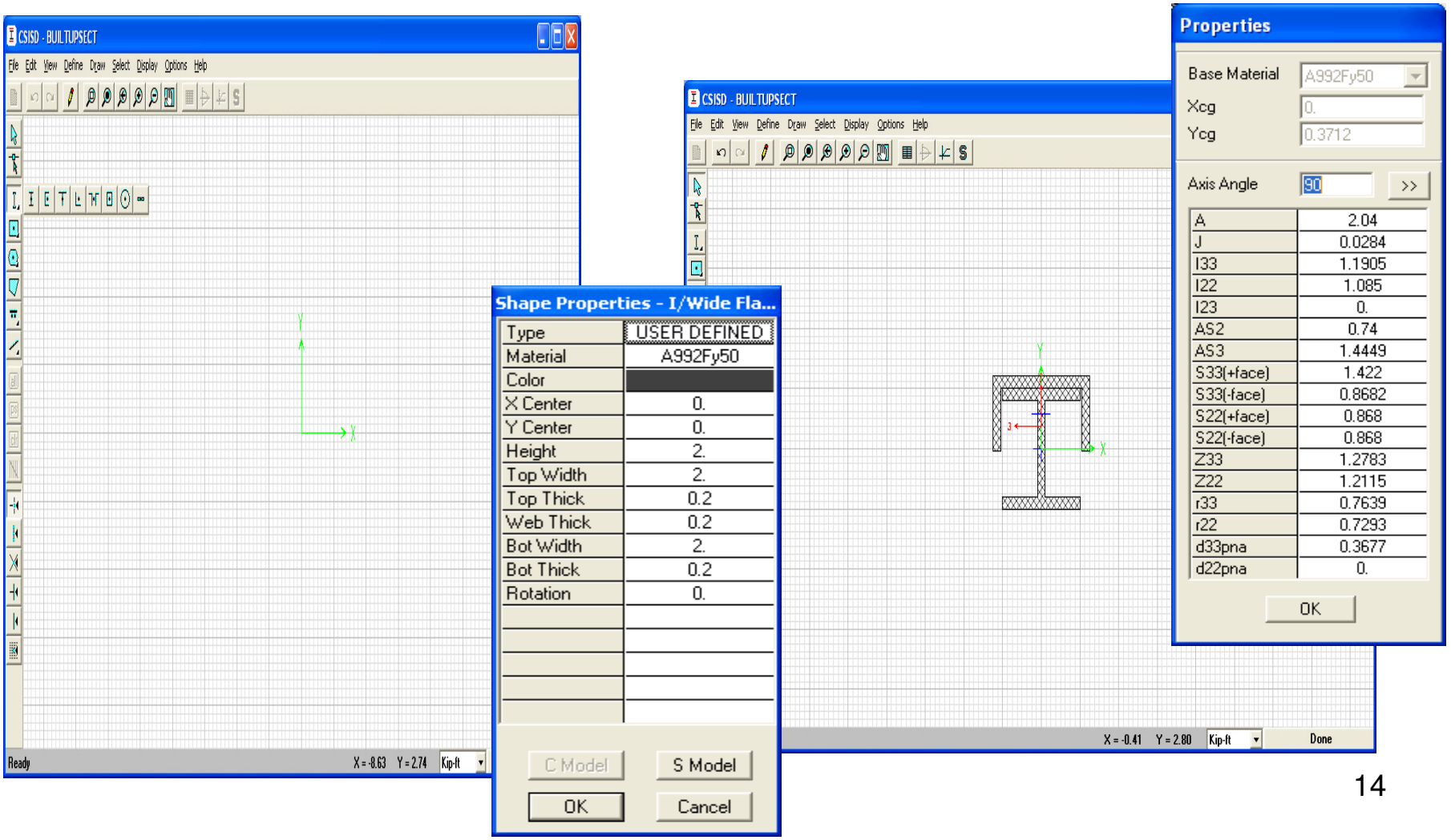


Click ‘Add new property’ again and change Frame section property type to ‘Other’. Here is where you can define general sections, nonprismatic sections, or access the section designer. Click ‘Section Designer’ button, and when you select a ‘Base material’, you have the option to include it in the design. Click ‘Section designer’ button shown in 2nd screen shot below

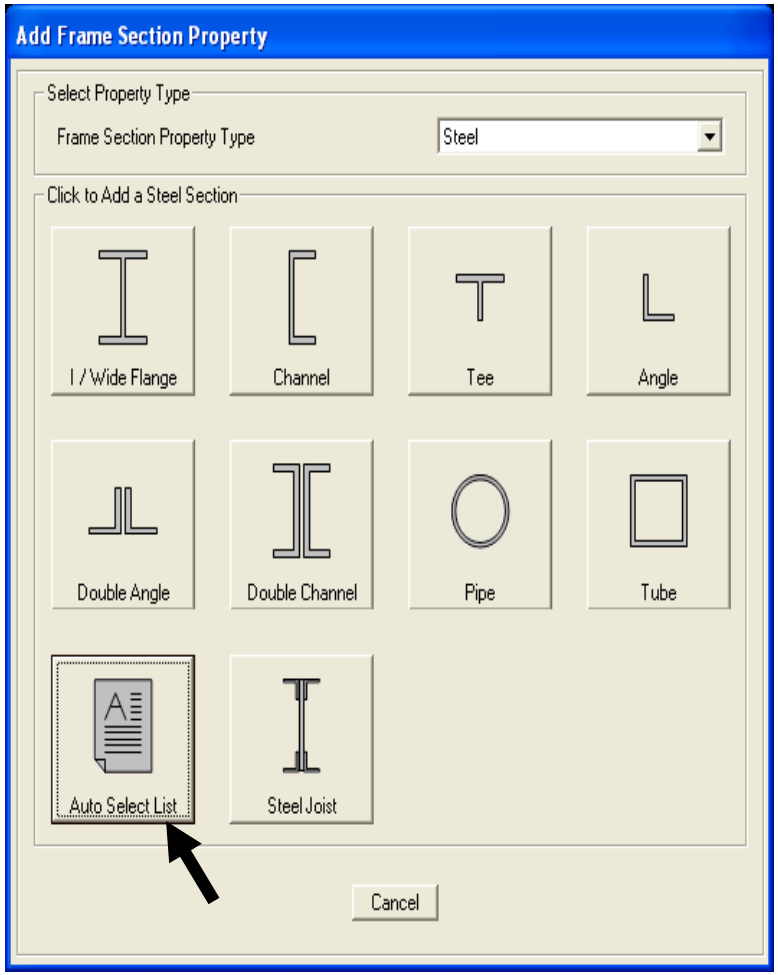


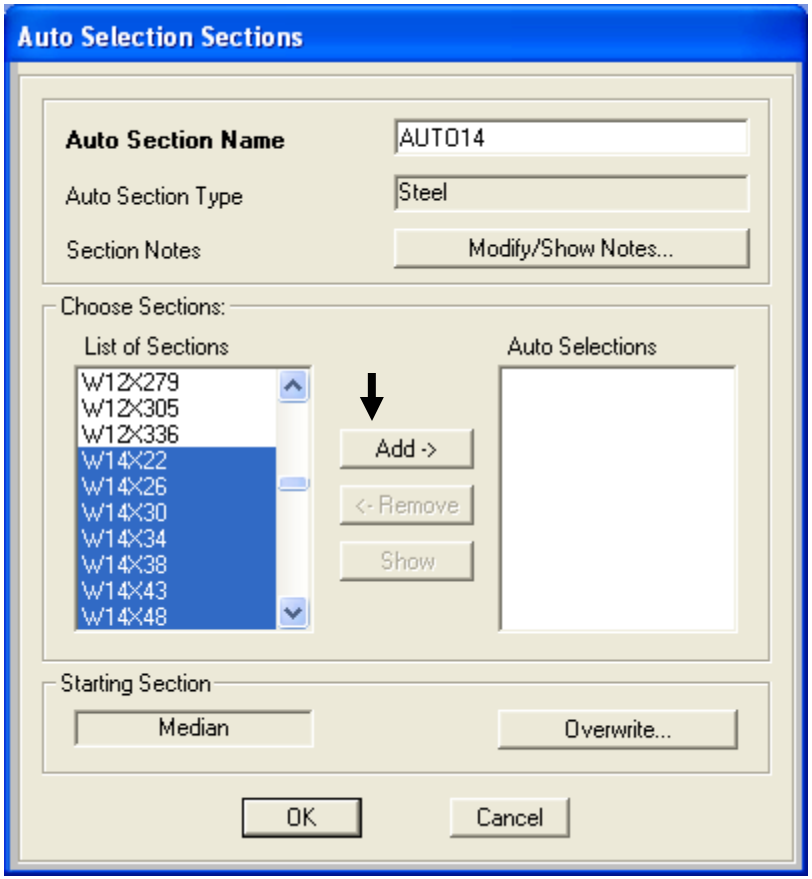


Use the shape options below to create built-up combined sections. Use reshaper tool and right click the individual sections to move them, rotate, or modify dimensions to help you combine them. SAP2000 automatically calculates combined areas, moments of inertia, etc. and uses this combined section not only for analysis, but also for design. Cancel out and return to Define frame section property dialogue to continue.

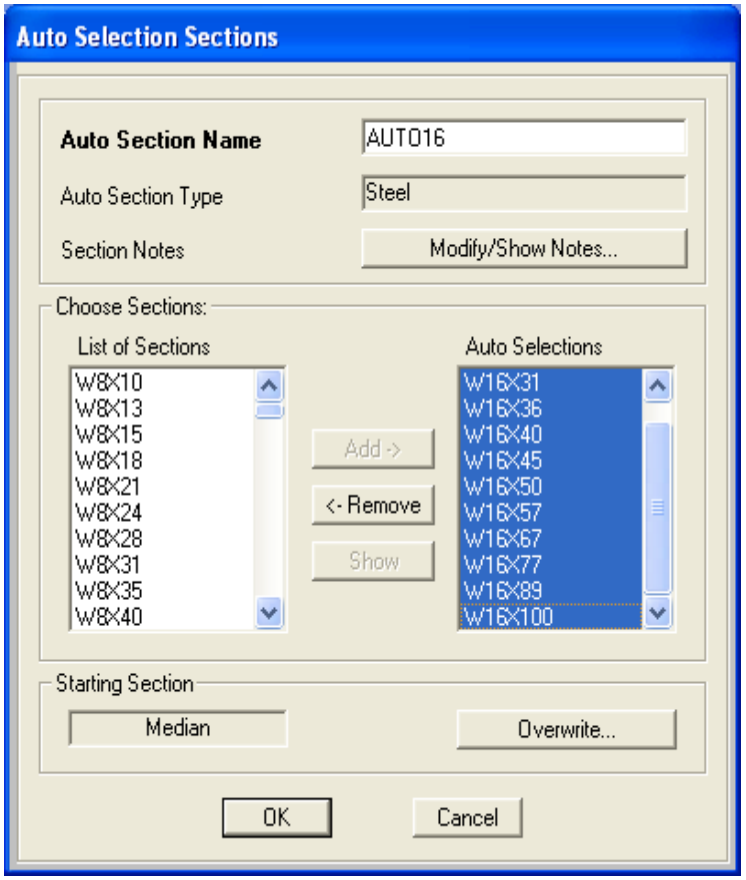


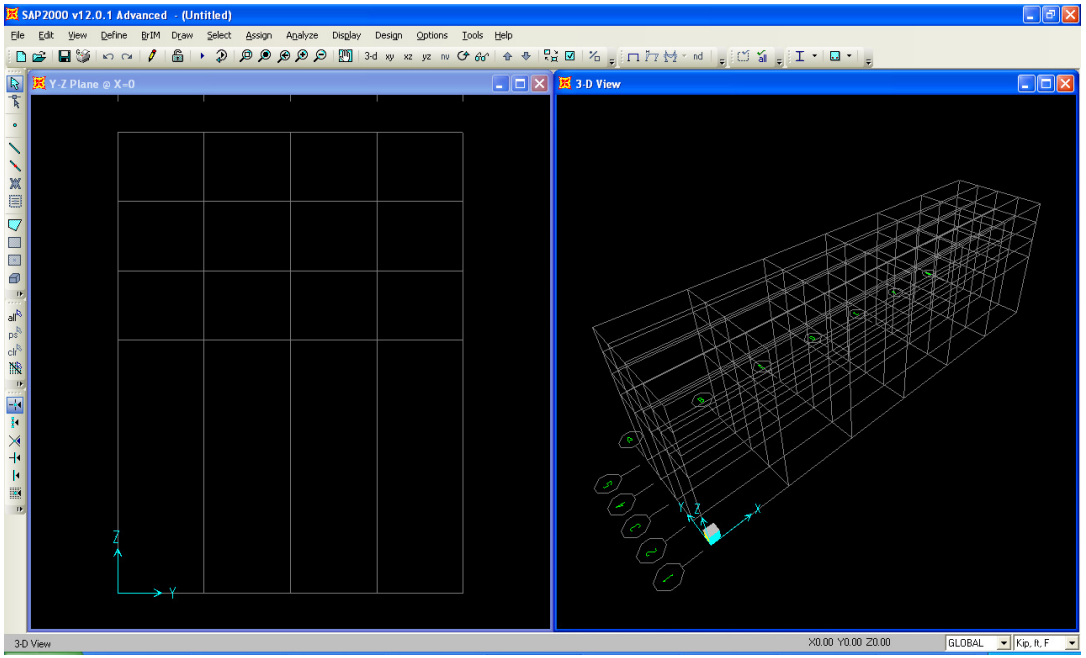
The drawings call for W14 columns. The exact section is not specified which means that the engineer needs to size the most economical W14 section which meets design requirements. We will do this by defining and assigning Auto select lists which SAP2000 will use to optimize for lightest section during design. We will define separate Auto select lists for W14's, W16's, W18's and W21's. Click 'Add New Property' button, and using frame section type Steel, click Auto Select List. Change the Auto Section Name to AUTO14 in order to help us remember, then select all W14 sections as shown below, then click Add-> to add them to the Auto Selections list



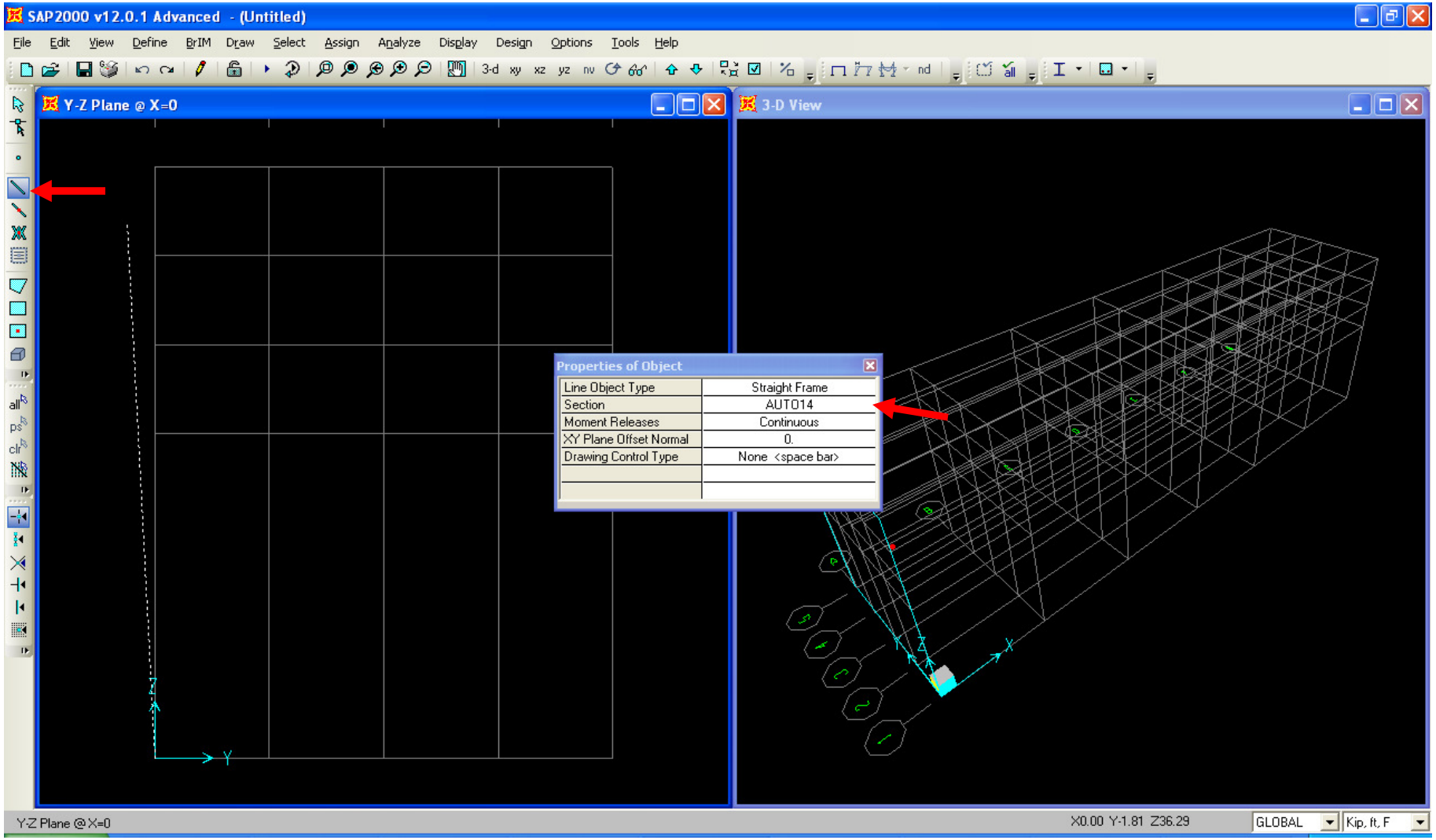


Change the Auto Section Name as shown below, select all the W16 sections on the left, then press Add-> to place them on the right, the press OK to accept. Repeat this process to create an auto select list for the W18's, then one more for the W21's. Don't forget to name each Auto Section list differently (AUTO14, AUTO16, AUTO18, AUTO21), then press OK until you return to the main menu screen shown below right

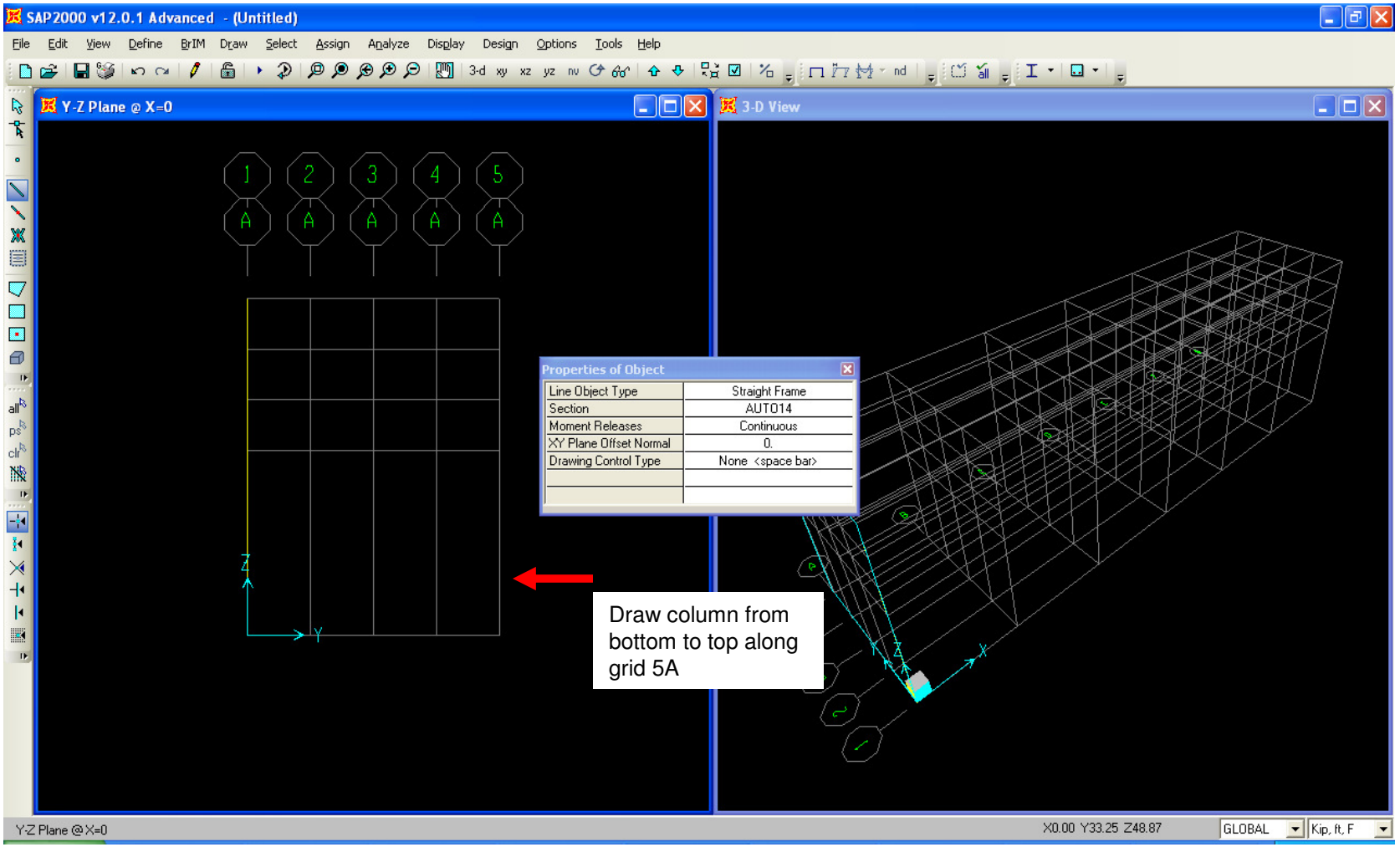




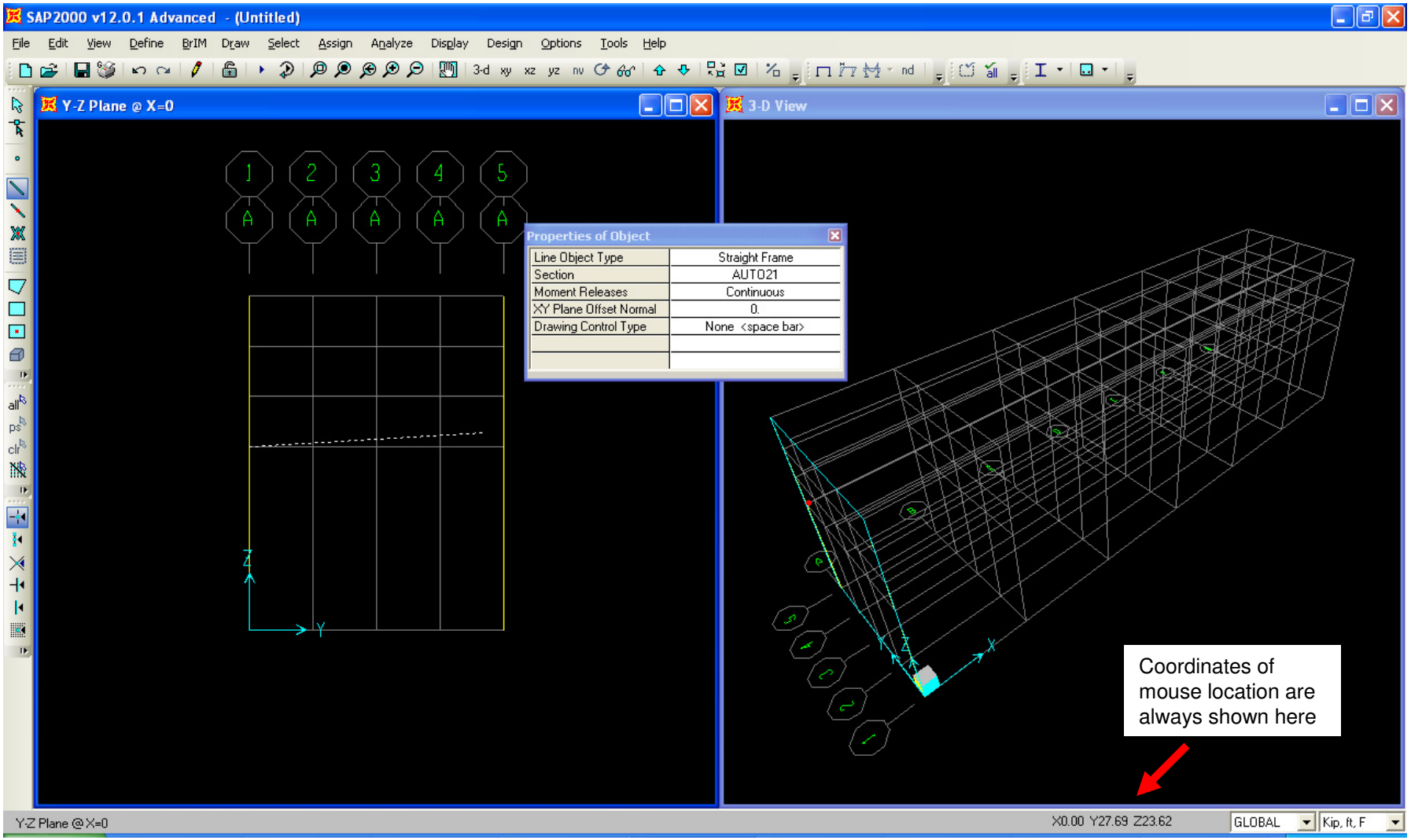
Click the ‘Draw Frame/Cable object’ button as shown, click Section in the floating properties box to select AUTO14, moment release continuous, then in Y-Z window X= 0 ft, click once on the bottom left as shown and then click once more at the top. Instead of assigning an individual section, for example W14X90 or another section, the W14 Auto Select list enables SAP2000 to optimize to choose the lightest section in that list which meets design code and user specified drift requirements.



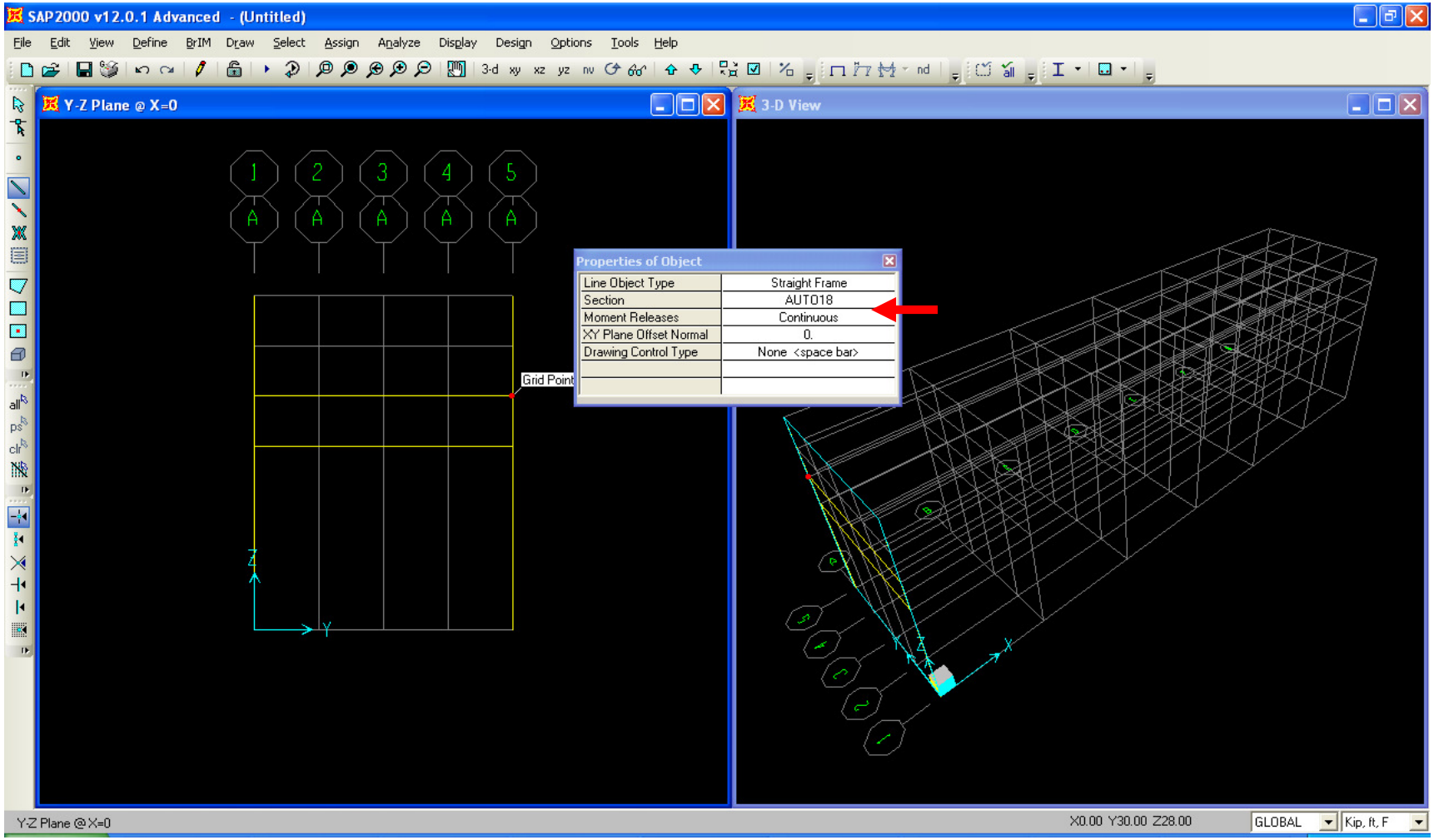
After clicking the top to complete drawing the column, right-click to move your cursor and draw another column along 5A. Click zoom-out button to view grid labels



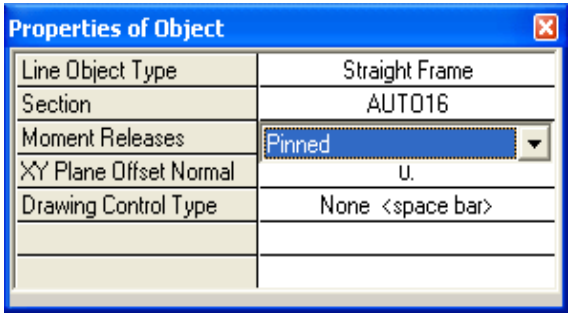
This is what your model should look like. Next, we will draw the W21 moment connected continuous beam as per the drawing. Click the Section and select AUTO21 and draw a beam left to right as shown. SAP2000 will internally create a joint at the beam/column intersection in order to transfer shear and moments, but SAP will then reformulate the results to report and design the columns using the length which was used to draw them (40 ft. tall columns)

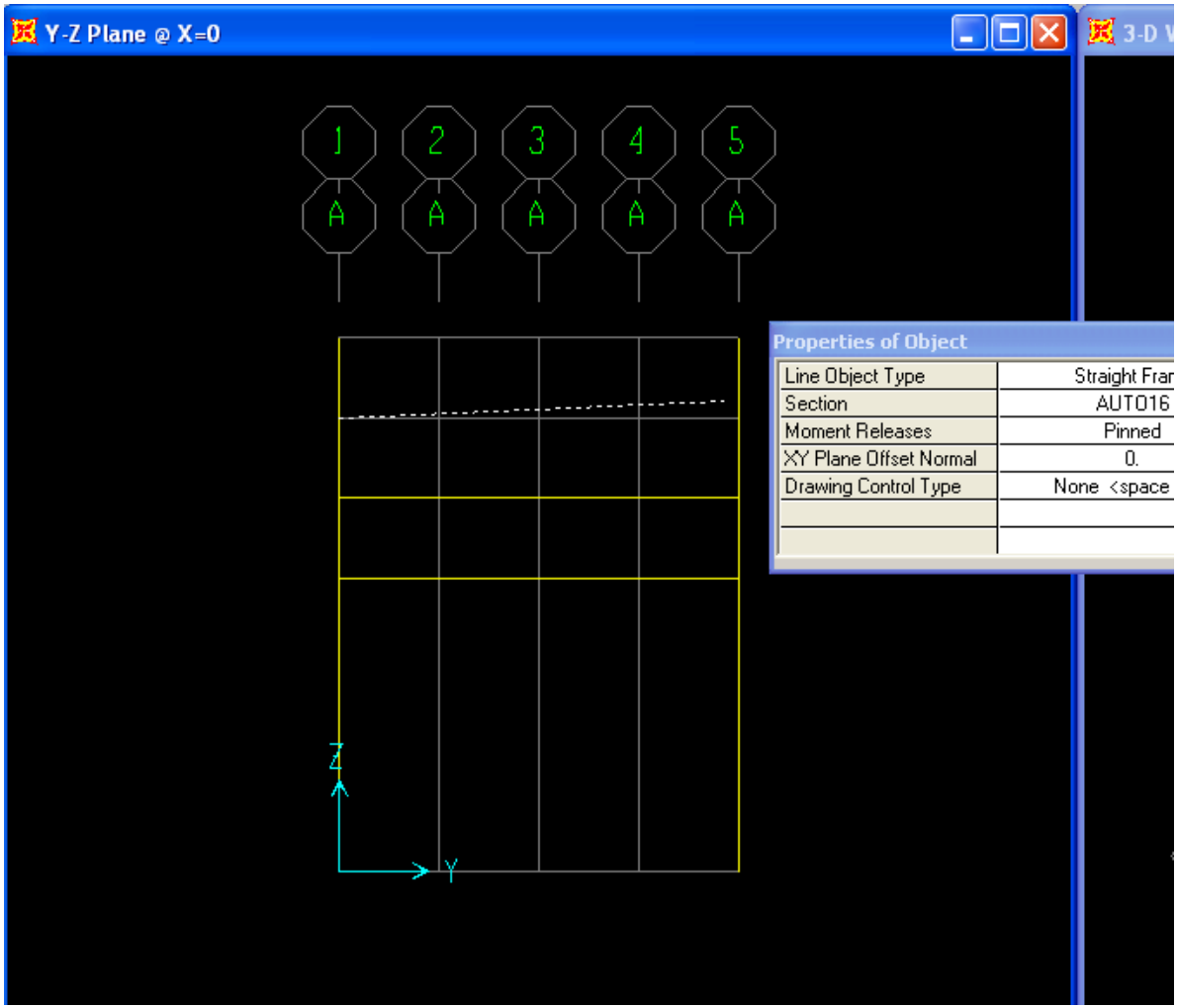


Right click to move your cursor, change section to AUTO18 as shown below, and draw another beam as shown. As you can see, the grids help you snap to draw

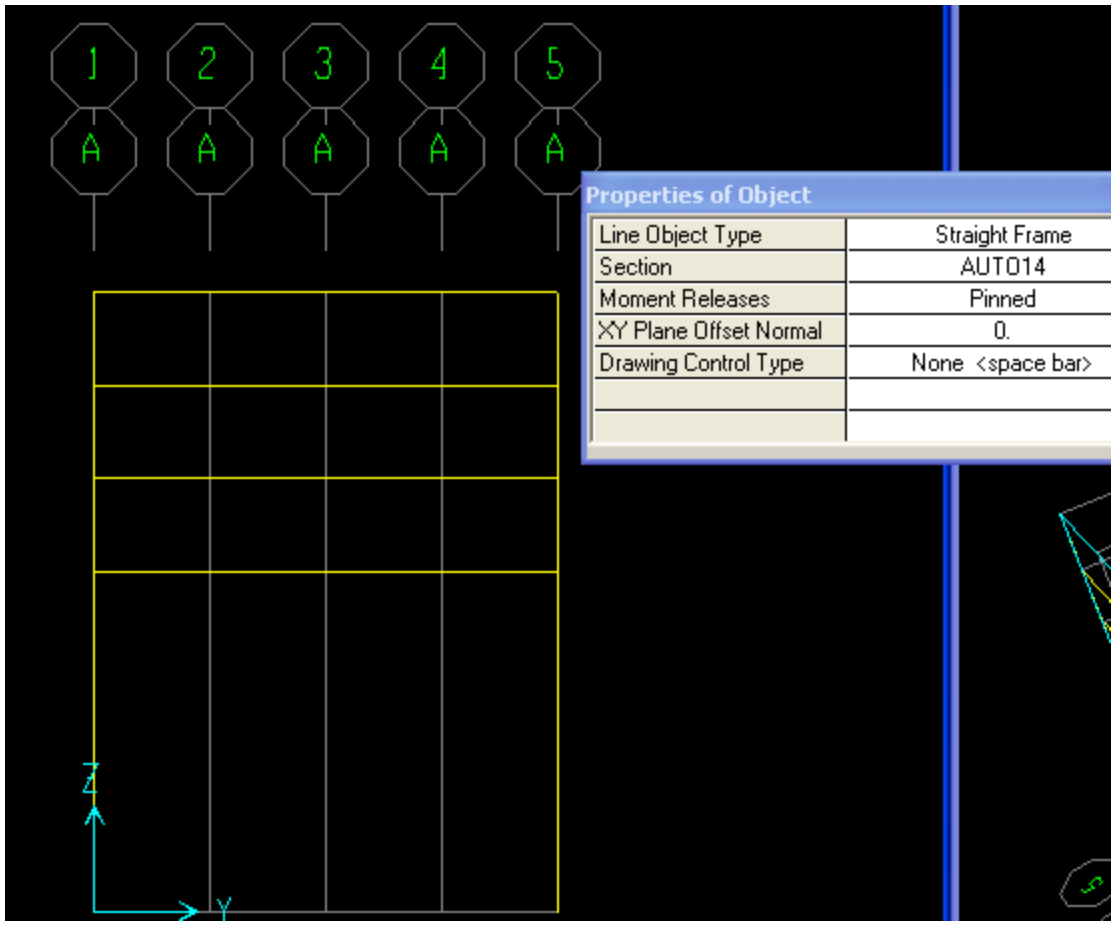


Change the Section to AUTO16, and since this beam is not designated M.C., the moment release needs to be changed to Pinned. Draw the third beam as shown.

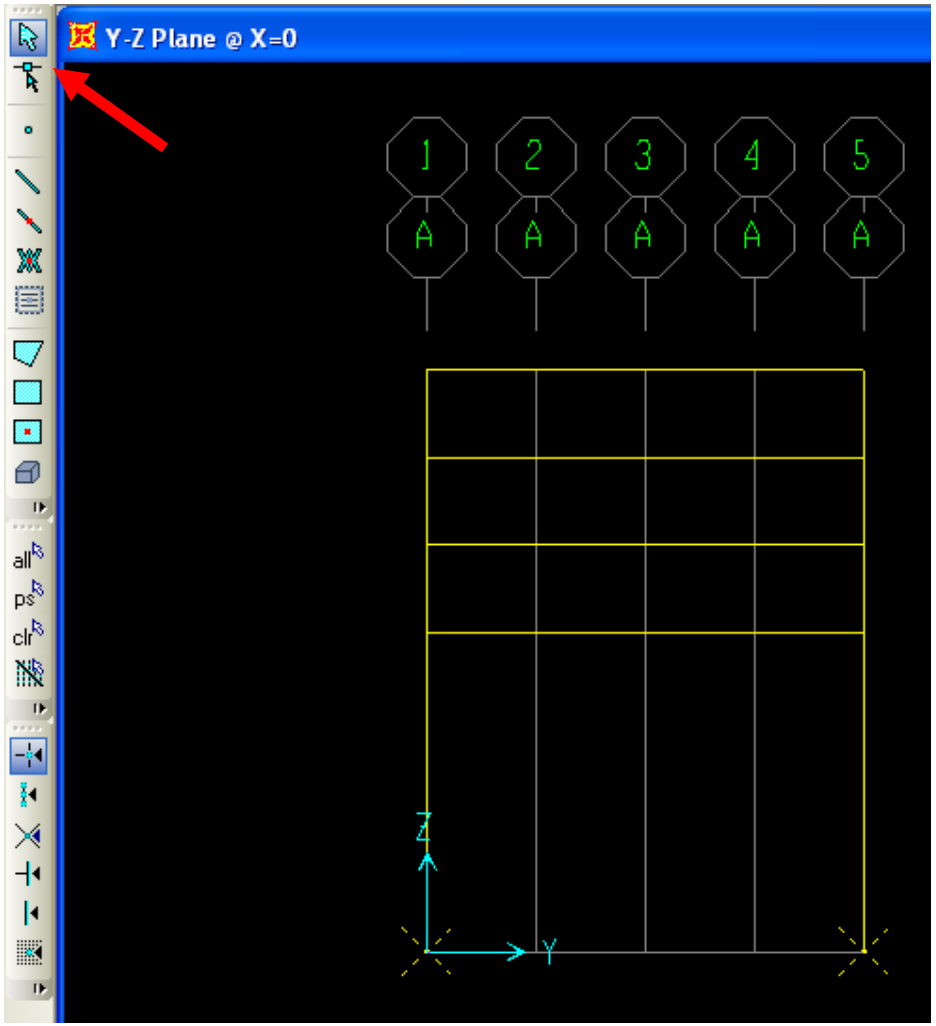




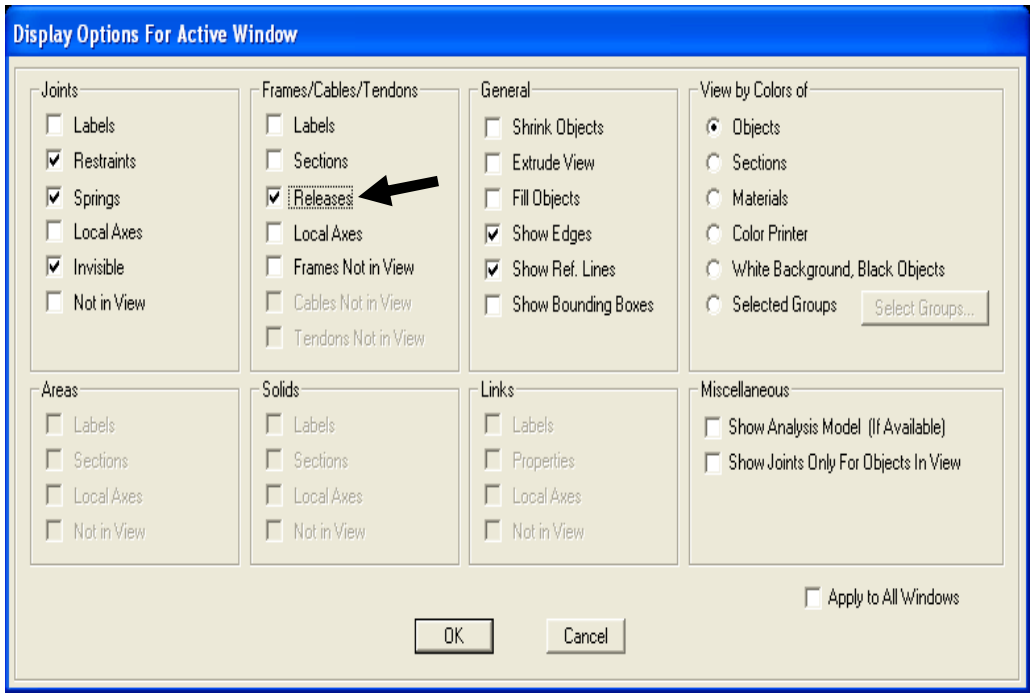
Change Section to AUTO14, keep the Moment releases as Pinned, and draw the 4th beam as shown along the top gridline just above the beam which was previously drawn

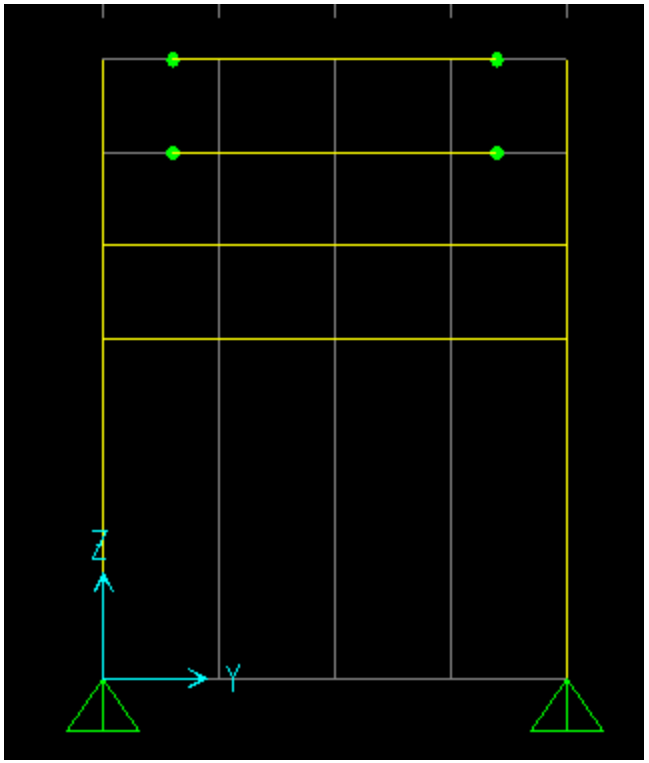


Press the Esc key on your keyboard or click the select button as shown below left in order to go into Select mode. While holding down the mouse key, drag to window a box around the bottom two joints to select them. Next, go to **Assign menu>Joint>Restraints** and add pin restraints as shown below. Press OK

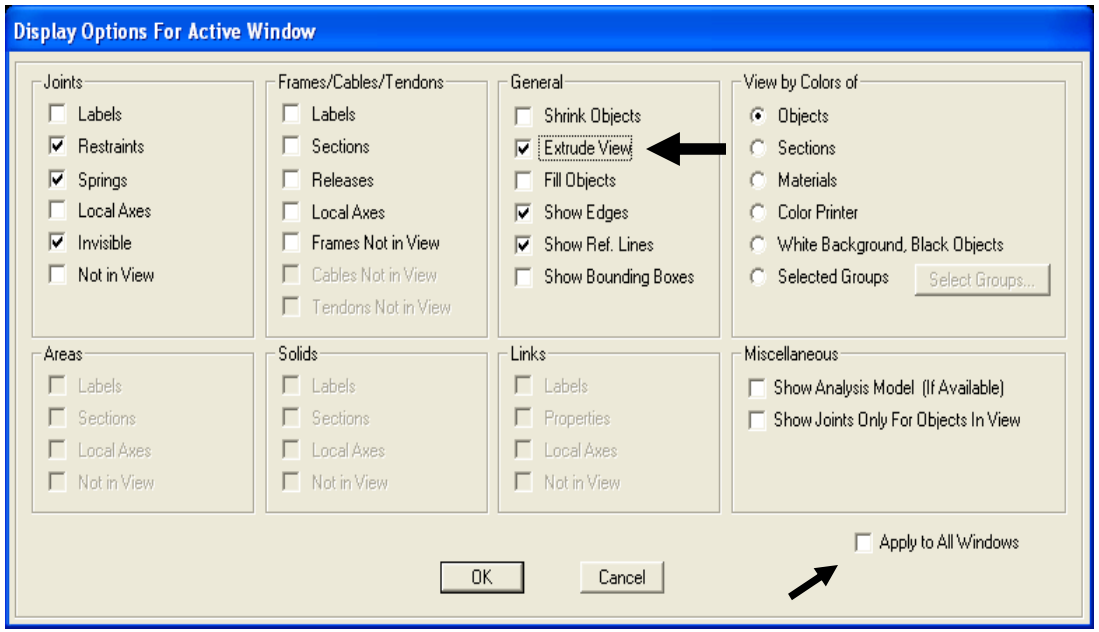


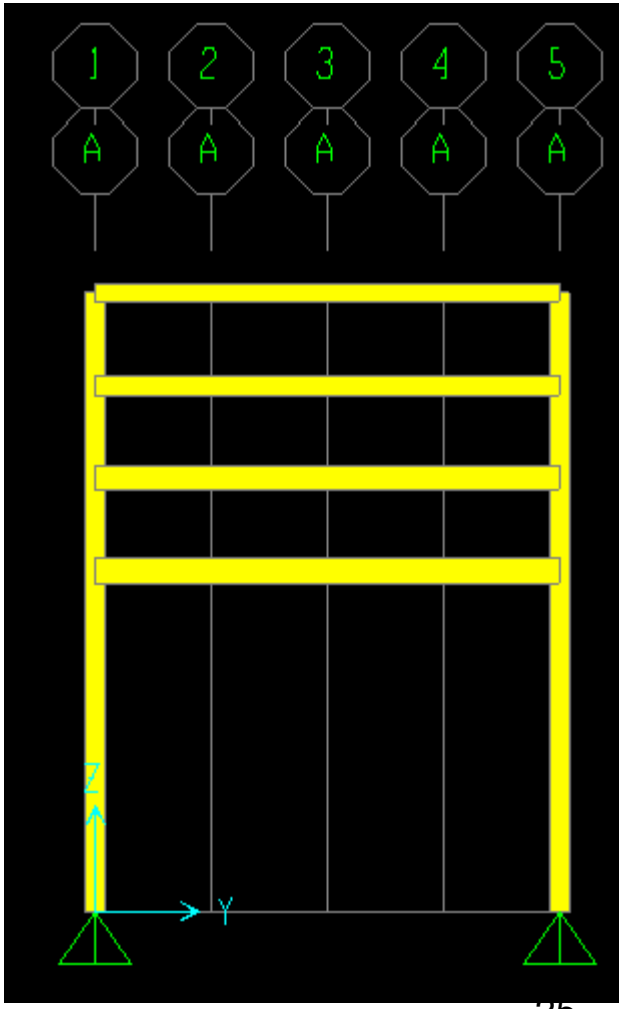
Click anywhere inside the 2D planar view window on the left to make sure it's the 'active' window, then click 'set display options' button  at the top, checkbox/activate the Releases option as shown and press OK in order to graphically view moment releases



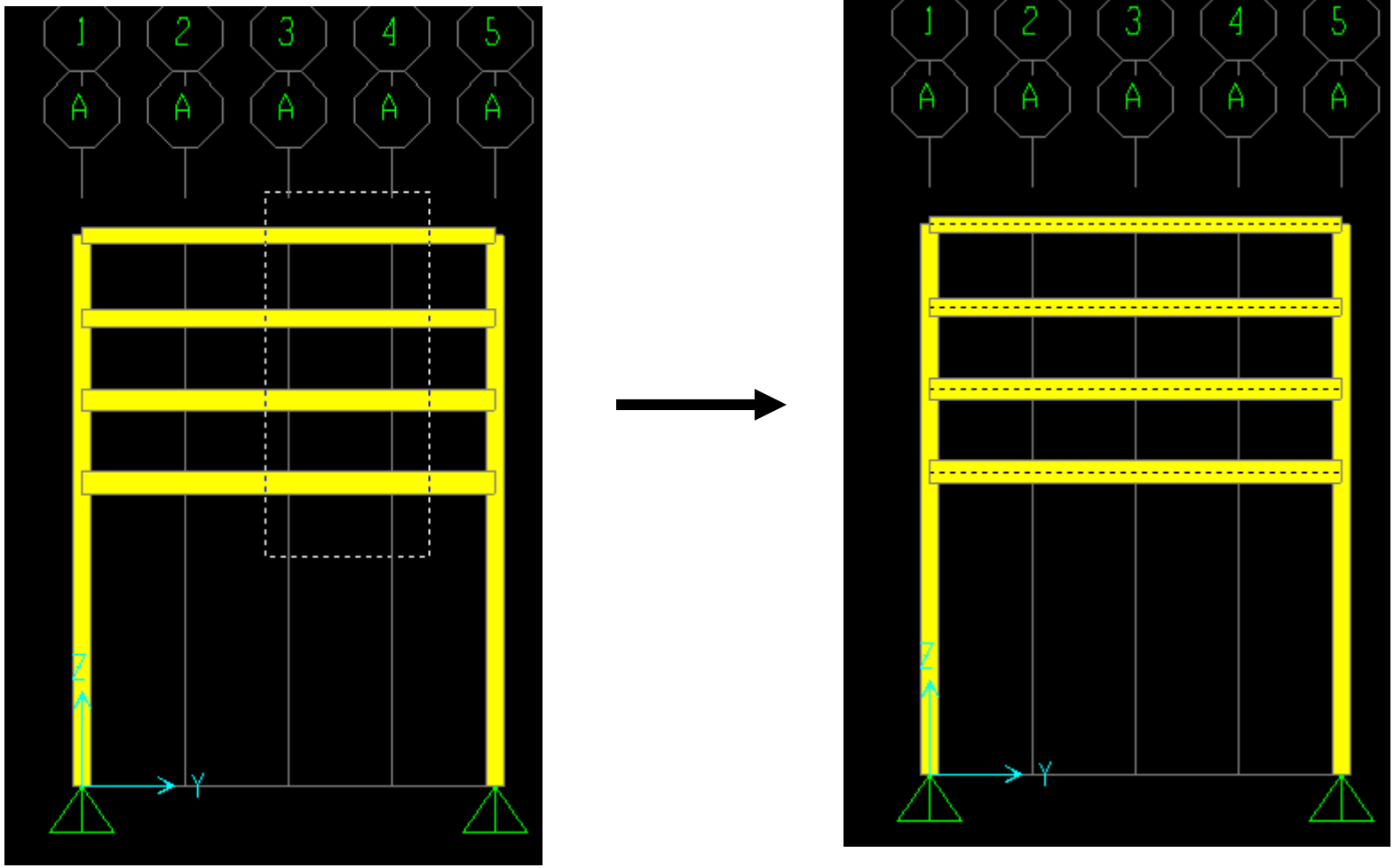


Next, click set display options button again, but this time uncheck the Releases and activate the Extrude view option as shown to create a rendered view. Note the option below to ‘Apply to All Windows’. Each display option can apply to the active window only or to all windows. In this example, we will apply only to our active planar view window.

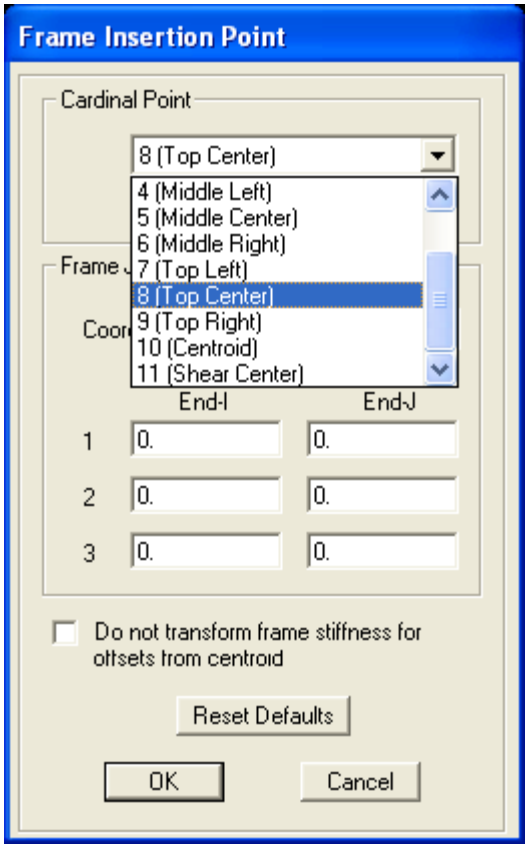




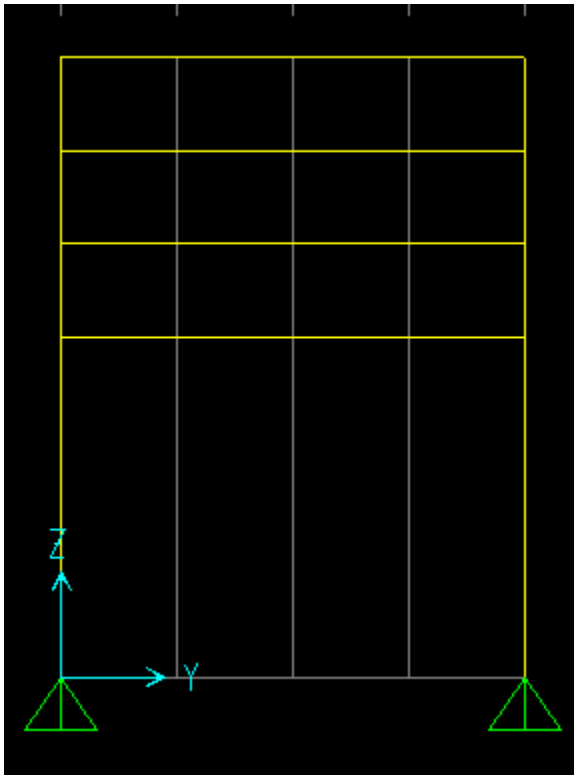
We will next adjust the beams to be dimensioned to top of steel (T.O.S. in drawing). Holding down the mouse key, drag your mouse right to left to window intersect the 4 beams. Dragging right to left is a convenient method to select elements and joints which are intersected.



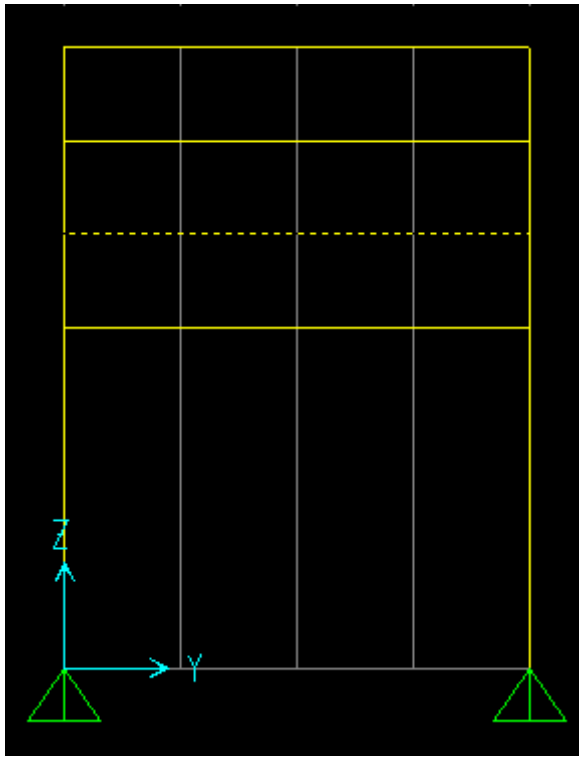
Next, use **Assign menu>Frame>Insertion Point** to assign the 'cardinal point' type of connection. You can press F1 with this dialogue open to read the Help menu for more information. In this case, we want top of steel, so select #8, top center and press OK. Based on frame section size and type of insertion point, SAP2000 moves the frame and adds a rigid link to account for the offset/eccentricity which will change analysis results compared to centerline connection. You can toggle back and forth with undo/redo in extruded view to visualize what SAP2000 is doing. Next, use the 'set display options' to deactivate the extruded view for the time being. You may have to press the F4 key on your keyboard to clear the screen if the insertion point assignments are in view after deactivating extruded view



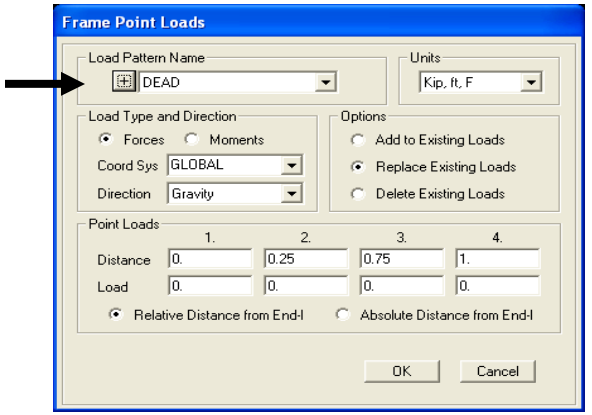
After assigning insertion point, use Set display options to uncheck Extruded view

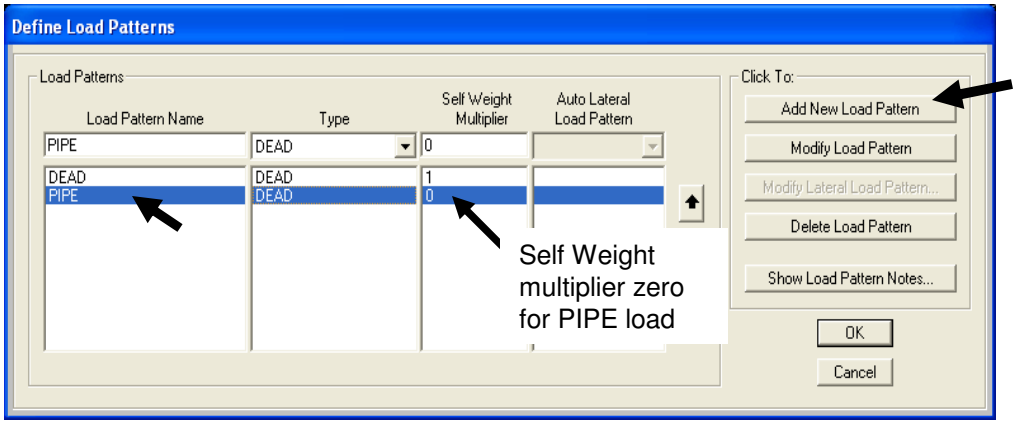


Per the drawings, there are two 24” diameter pipe loads dimensioned from the left side beam column intersection on the W18 beam, one 15’ feet from the left beam/column intersection and the other 3’ away in the Y direction from the centerline of the first pipe. Left click the AUTO18 beam and use **Assign>Frame loads>Point**. With this option, you can assign one or more point loads along selected frames based on percentage or actual distances from I to J end using current units. Click + sign to add additional load pattern for pipe loads as shown. Name the Load Pattern “PIPE”, type DEAD, Self weight multiplier of 0 to avoid double counting self-weight of the structure, then press ‘Add new load pattern’ to add it to the list and press OK. Alternatively, we could have used **Define>Load Patterns** beforehand

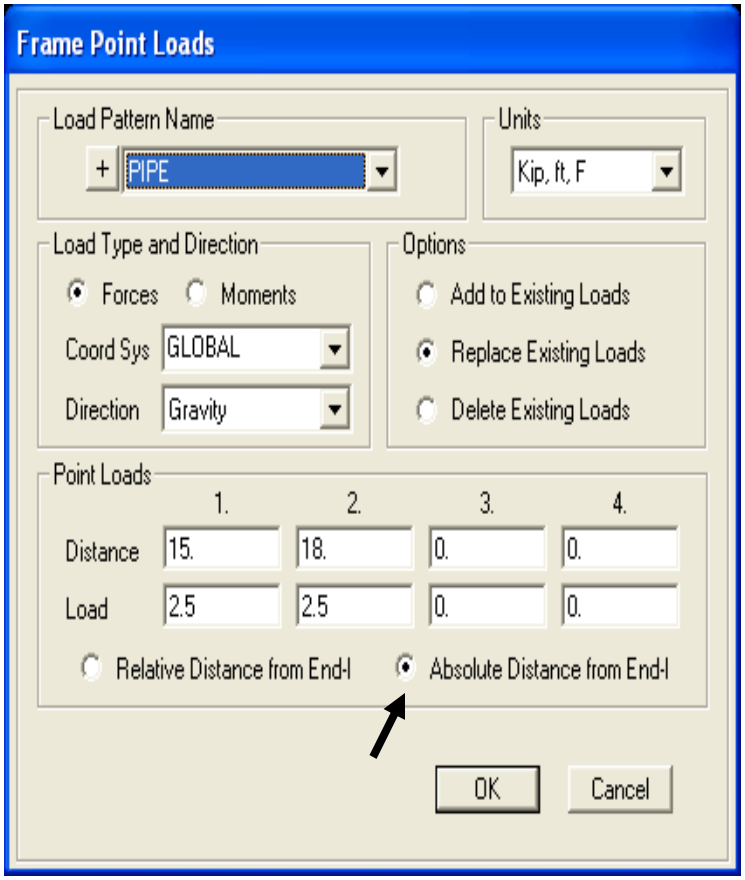


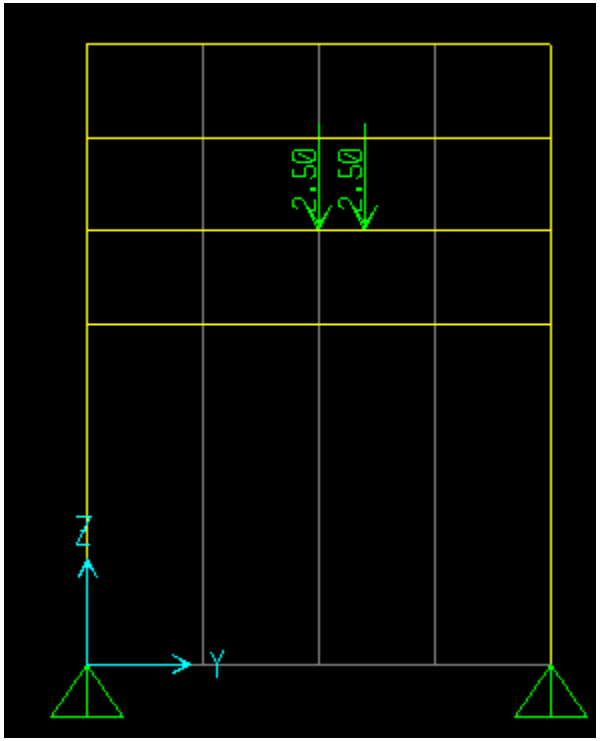
Click + sign to define additional load patterns



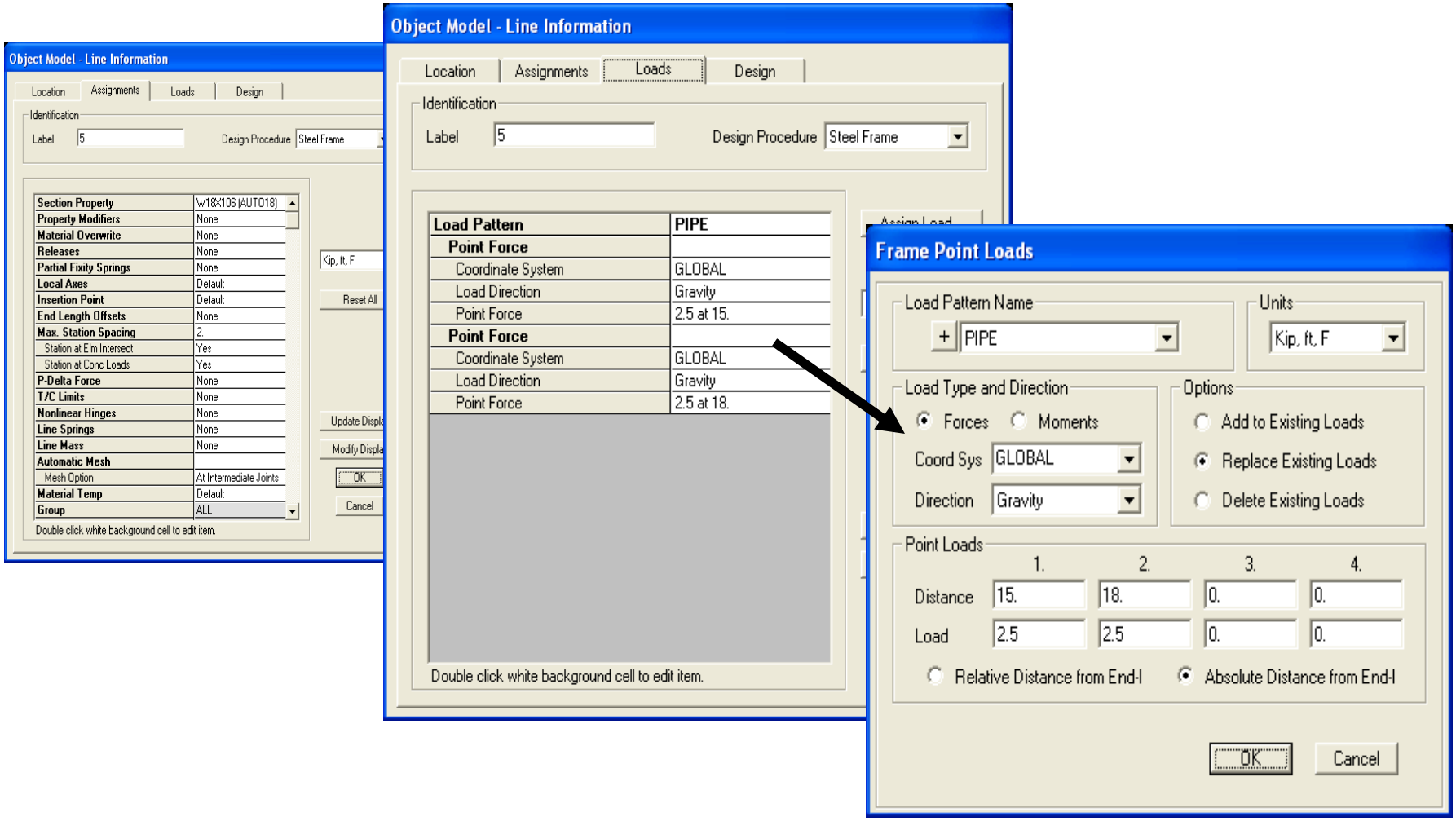


Switch Load Pattern to PIPE, select ‘Absolute distance’ from End I with Kip, ft units and assign a 2.5 Kip load at 15 ft. and 18 ft. as shown below in the gravity direction and press OK.

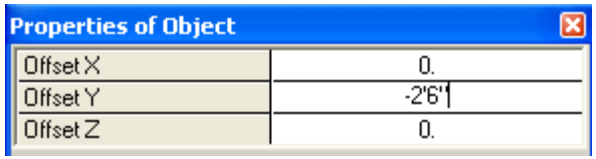


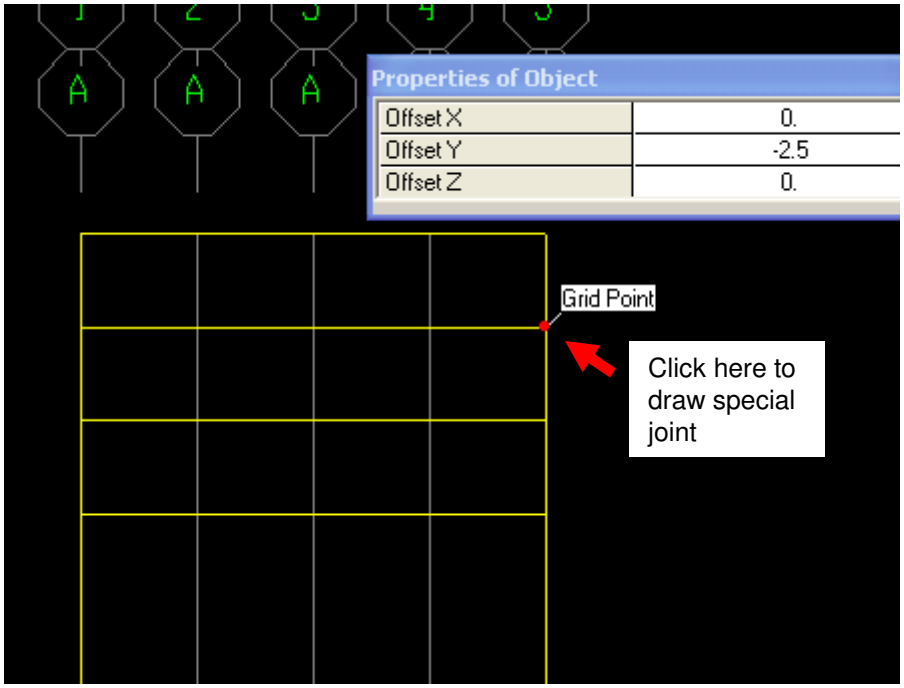


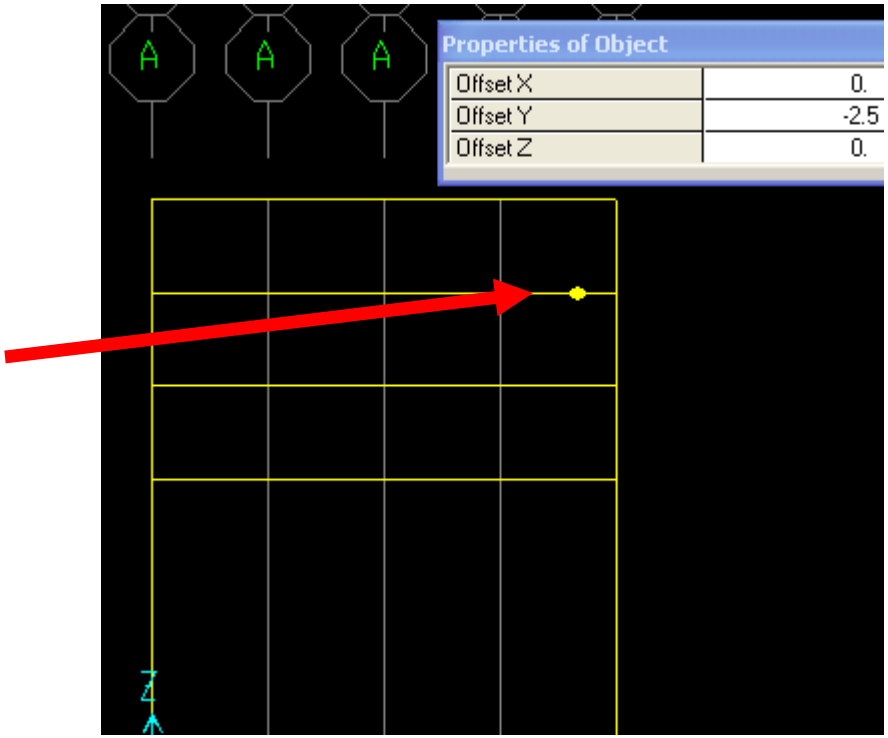
Right click any element or any joint to review or modify information. Here you can modify any parameter on the form using any of the tabs by double clicking with your left mouse key in the white area. Go ahead and press OK or Cancel without modifying anything at this time.



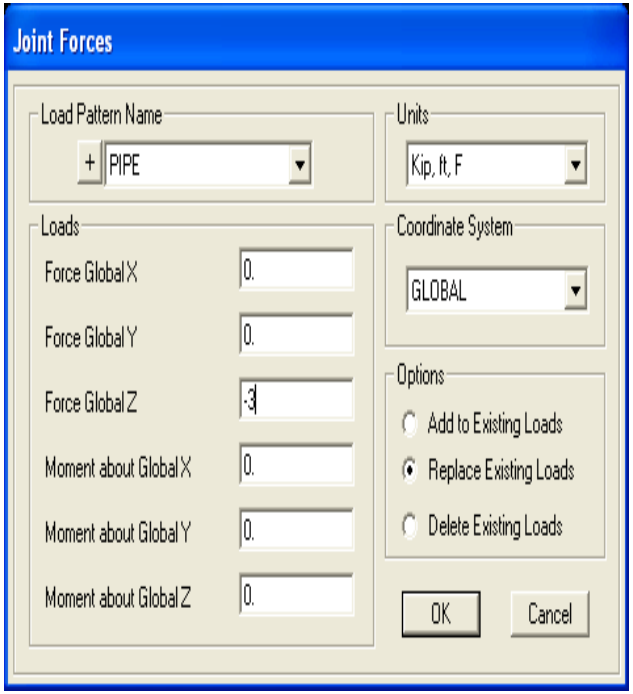
We will use a different joint load assignment method for the load assignment on the W16 beam. Per the drawing, the pipe load is 2'6" away from the right side intersection of the W16 beam and the column, so click the 'Draw special joint' button and type -2'6" in the Y direction as shown. Next, click the intersection of the W16 beam and column as shown below with the grid point bubble to add a joint. The Draw special joint option will draw joints offset from wherever you mouse click. Since drawings often reference offset dimensions, this can be a useful option for assigning joint loads or to create joints in order to connect the dots to model frames.

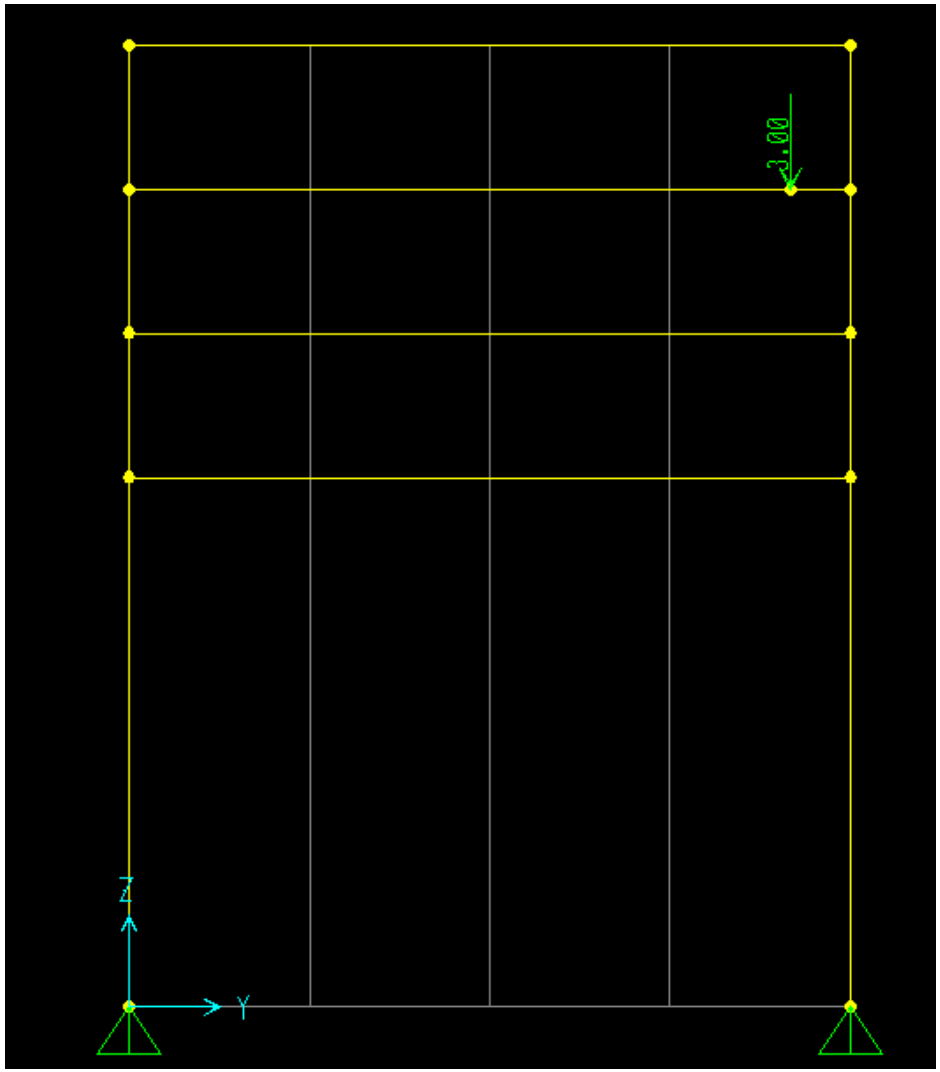




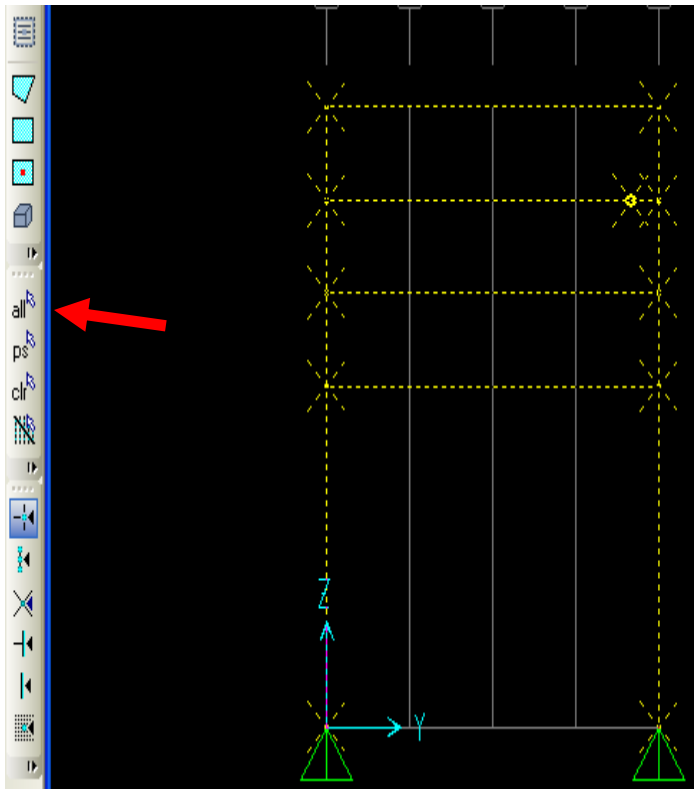


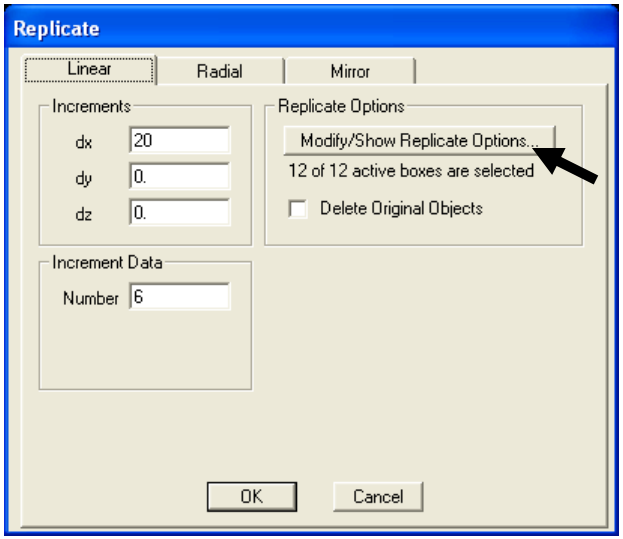
Press select arrow or Esc key to go into select mode, then click the joint to select and Assign>Joint load>Forces. Change load pattern name to PIPE or add a new load pattern if you wish to create separate load cases for different size pipe loads, then assign a 3 Kip joint load in the –Z (gravity) direction and press OK. Press F4 to clear load assignment.

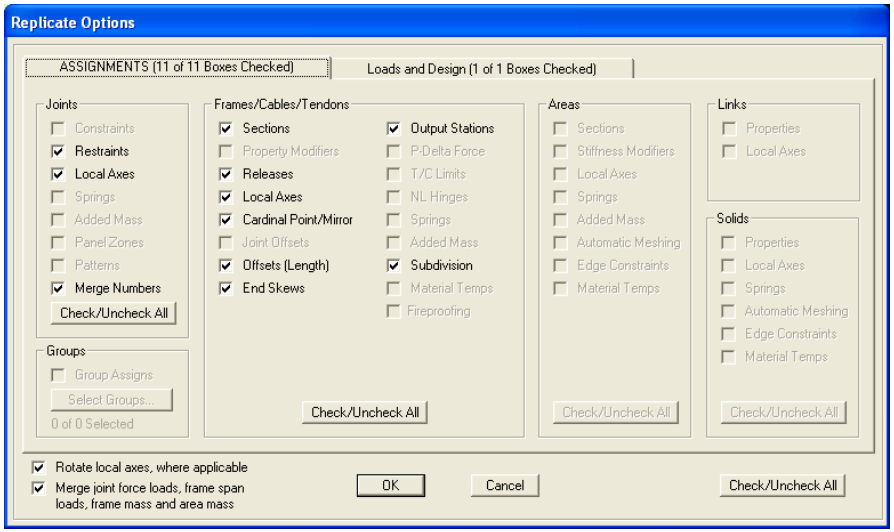




Click ‘Select all’ button as shown, then from the main menu **Edit>Replicate** and type dx 20’ increment 6 to replicate the selected elements and joints 6 times at 20 ft. each in the X direction. As an exercise, click the ‘Modify/Show Replicate Options button. Here you can pick and choose what you want to replicate what you don’t (load assignments, local axis rotation, restraints, etc). Cancel or Press OK on the Replicate Options to accept all, then press Ok on the Replicate screen

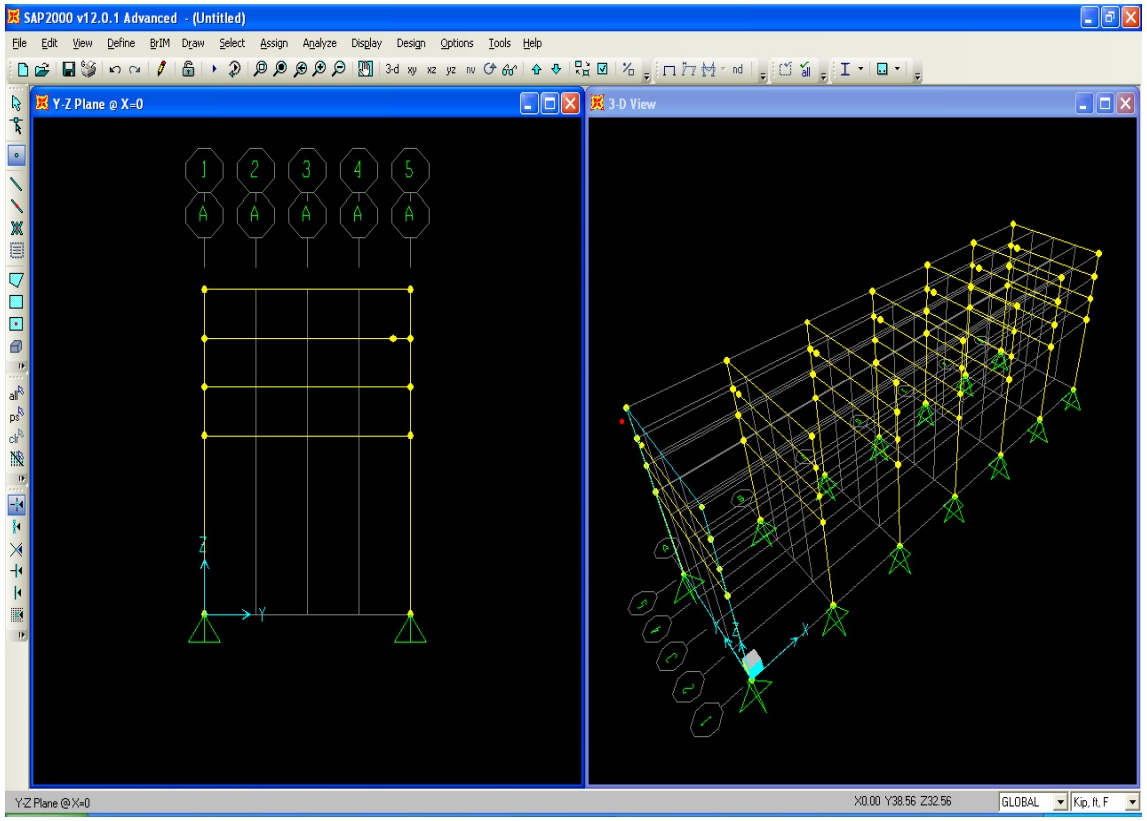




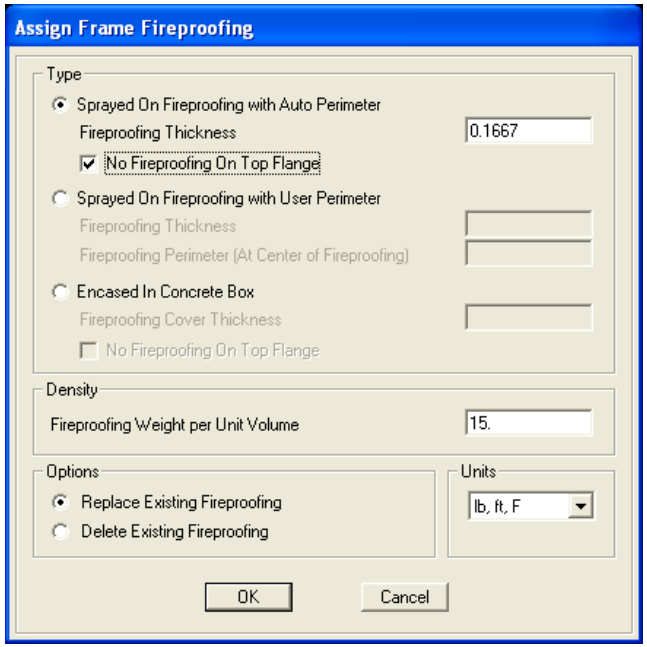


This is what your model should now look like. Click once in the 3D view window to make it the 'active' window. Use 'set display options' to activate extruded view, click the zoom button and hold down your mouse key to window into the area in the far X direction (see next page).

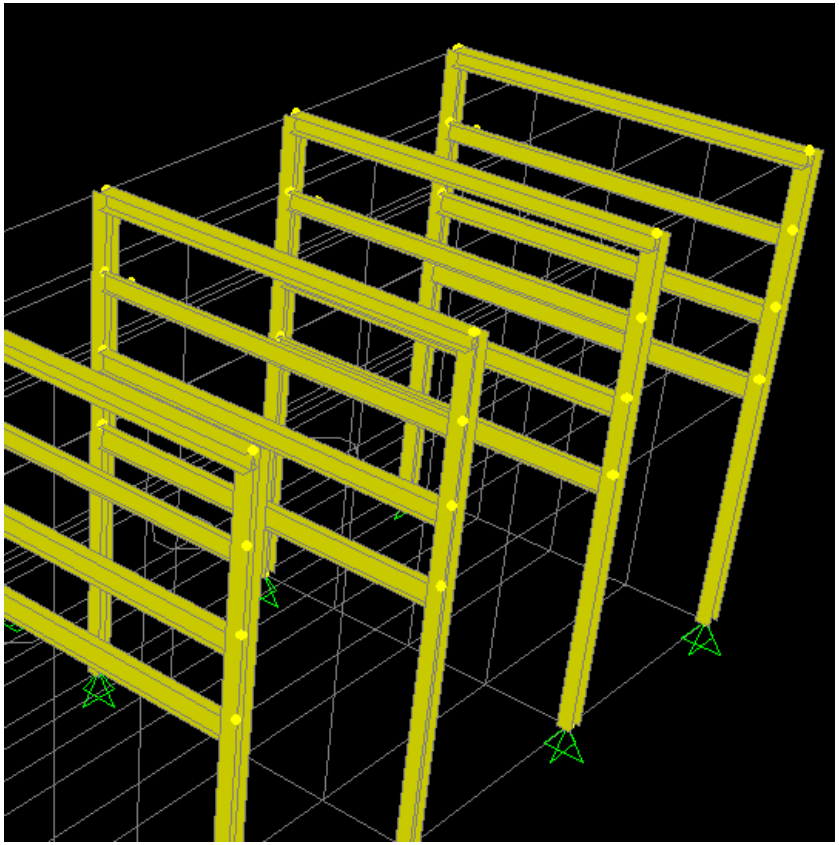
In a real design, we would adjust the pipe loads to account for different tributary area load distribution, but for this example, we will leave all bays with the same pipe loads as assigned on the end frame. By selecting frames, users can **Assign>Frame loads>Distributed** or **Assign>Frame>Fireproofing** to assign other loads which we will ignore for this exercise

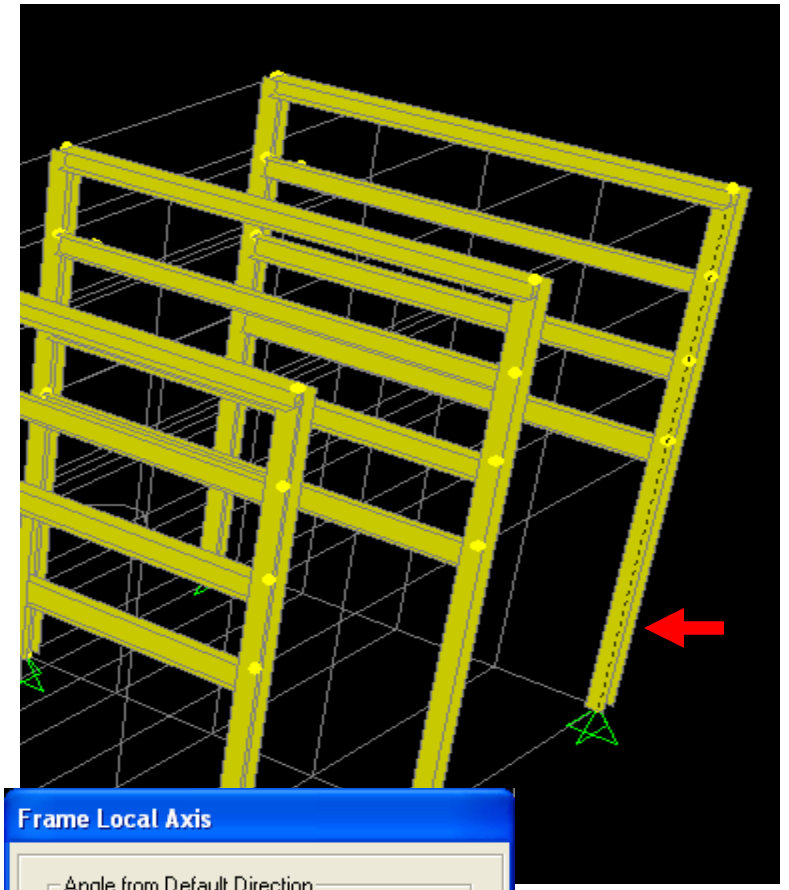


SAP2000 offers an automated feature to assign fire cladding weight which can save time compared to manual calculation and assignment of fire cladding

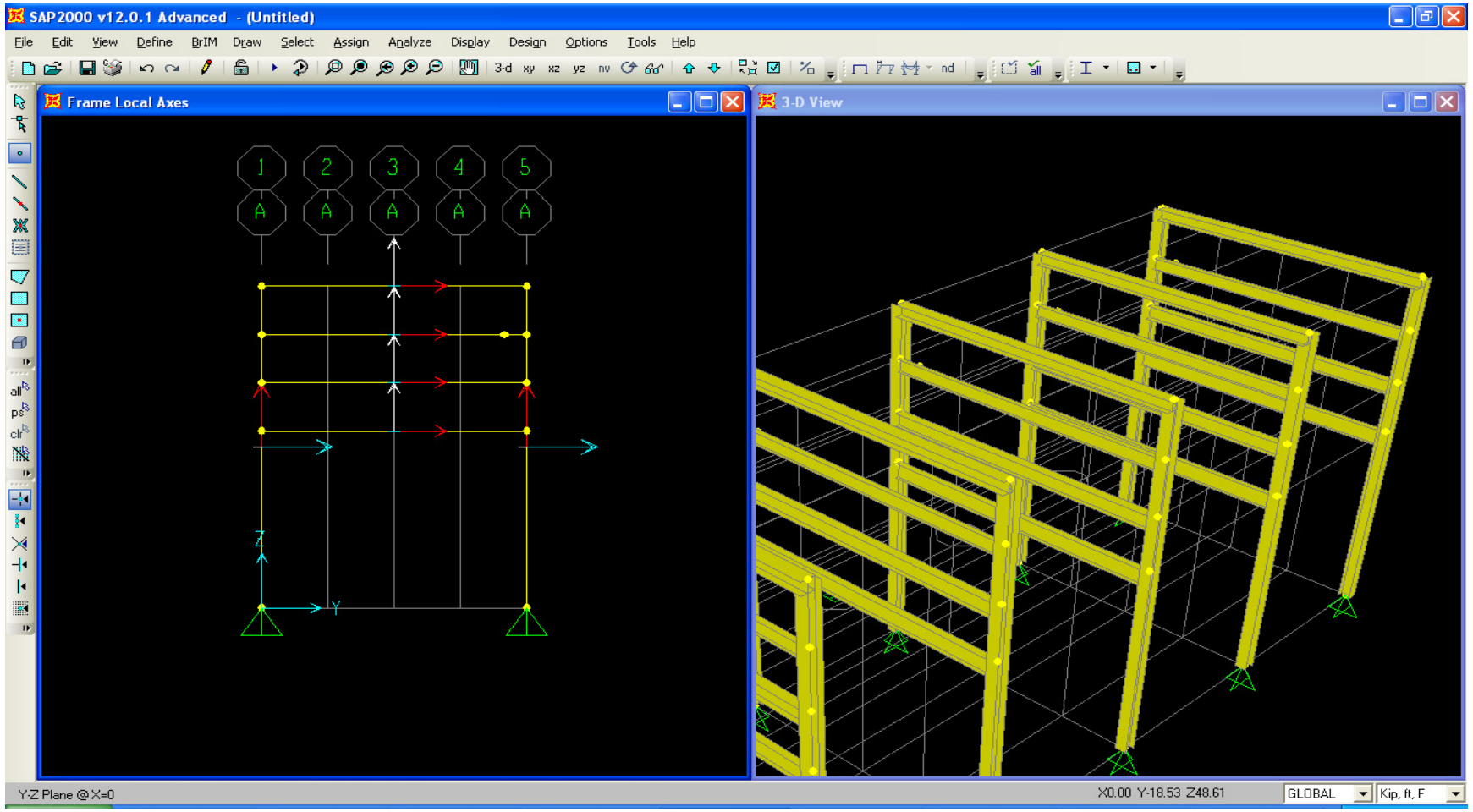


If the extruded view does not appear, press F4 and try again. Next, we are going to learn how to rotate the local axis of frames. Left click the column shown below to select as shown below right and **Assign>Frame>Local axes**, type 90 degrees and press OK.

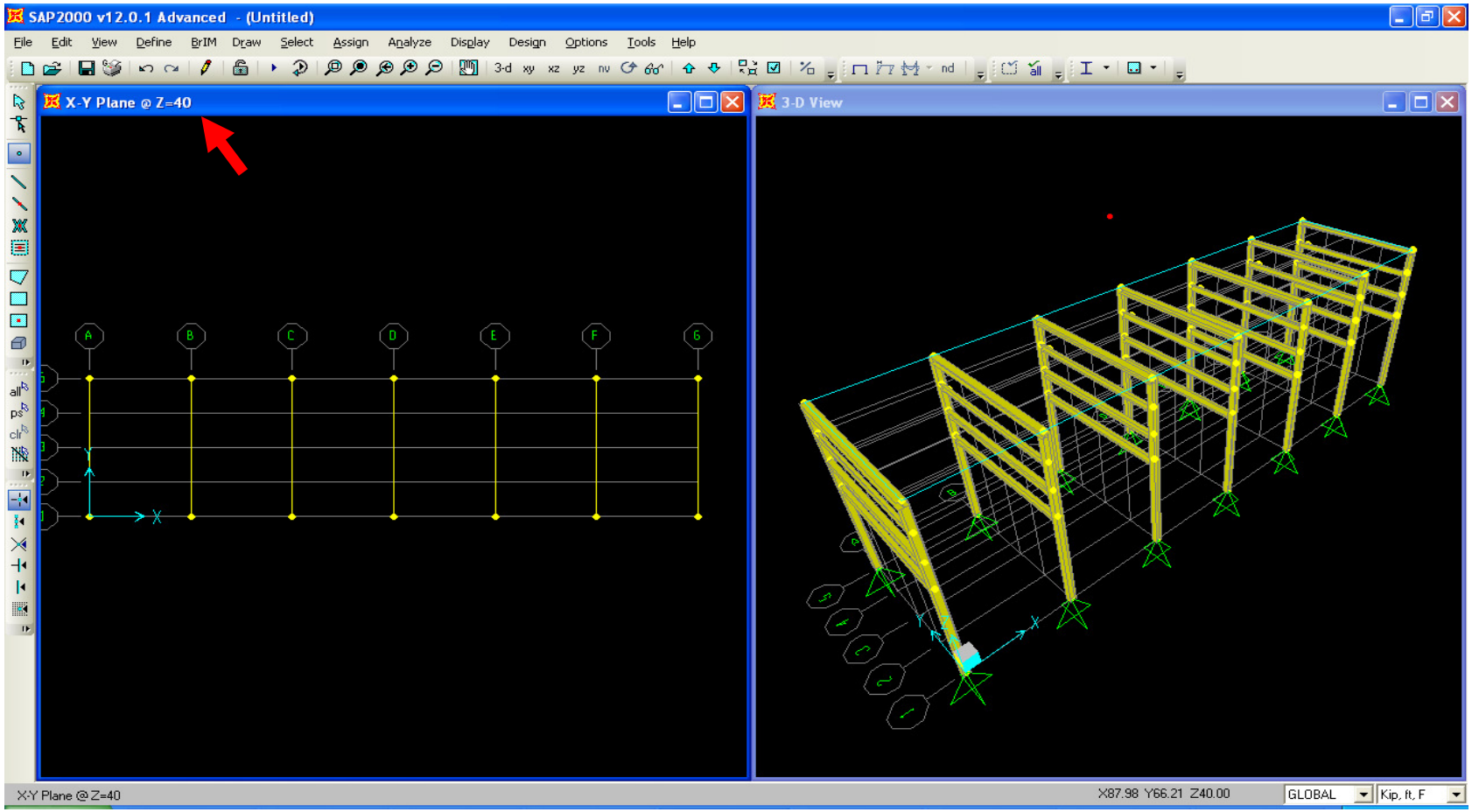




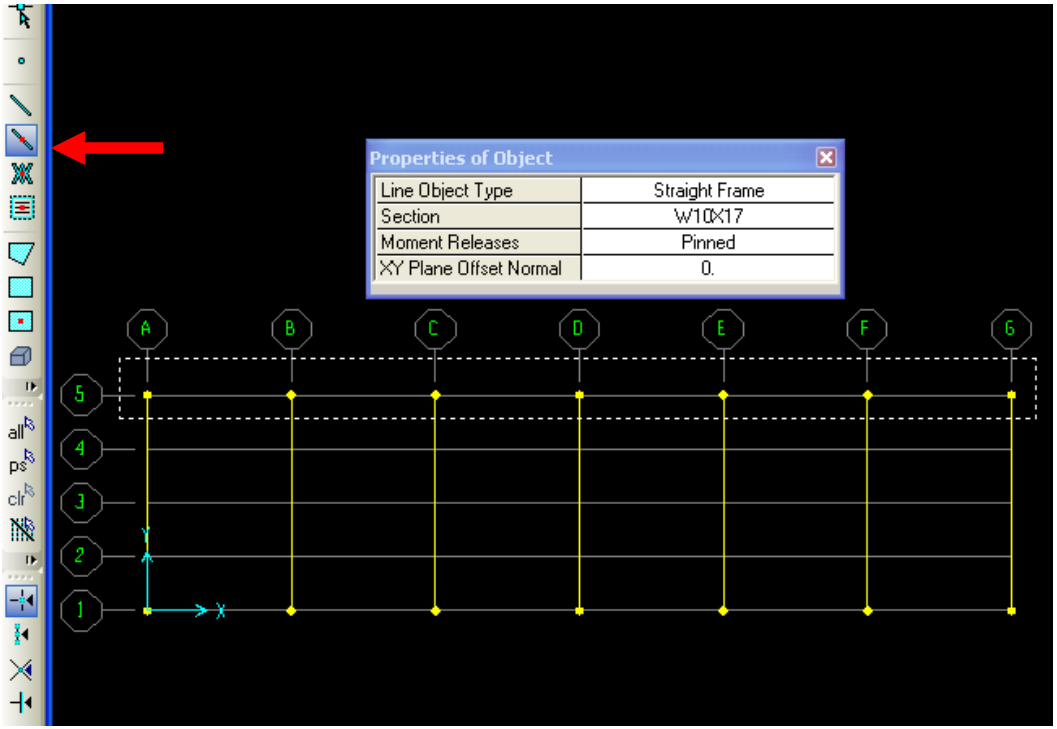
If you zoom in on the extruded view, you can view how the local axis has been rotated. By rotating the local axis, you may activate the red/white/blue local axis arrows in the planar view. Local Axes 1, 2 and 3 correspond to the arrow colors red, white and blue, just like the American flag. Frames, joints, links and area finite elements each have their own local axes which can be viewed anytime using the 'set display options' dialogue. Press F4 to clear the arrows and press undo button to undo the rotation.

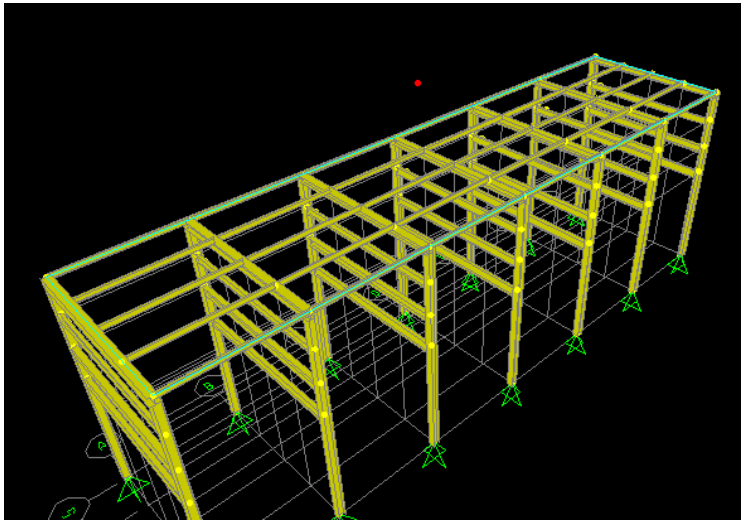


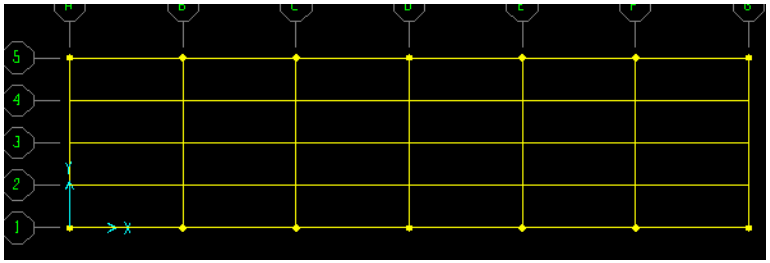
Next we will draw the grating spans for the air coolers. Click anywhere in the left window to make it active, then press the xy button to switch to plan view. You may have to click the up/down arrows in order to move XY plane to Z = 40 ft top level as shown. In order to view full 3D window, click once in the 3D window to make it active, then press 3-d button or press F3 for full view



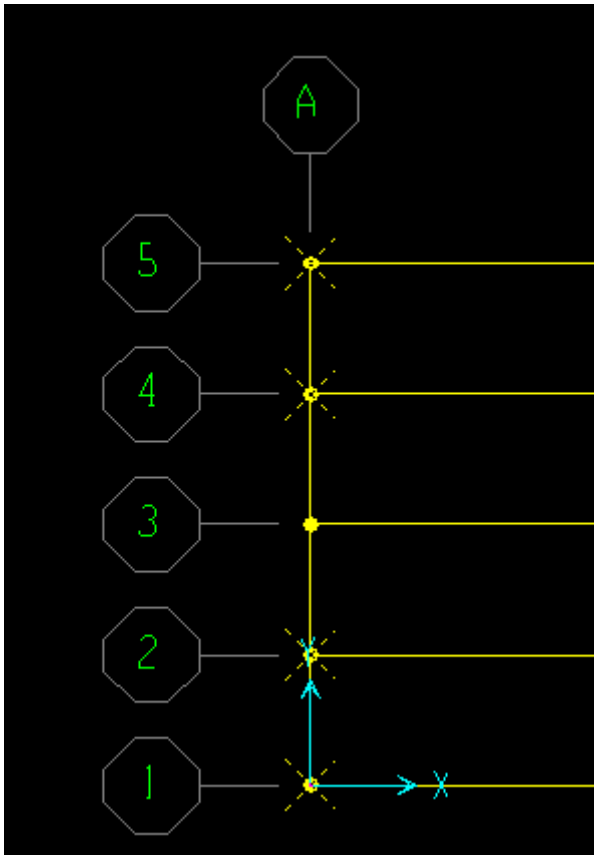
Press ‘Quick draw frame/cable element’ as shown, click to change section of W10X17 Pinned, hold down your left mouse and window around gridline 5 as shown. Repeat this draw procedure one-strip-at-a-time for grids 1 – 4 too. The quick draw frame element enables you to window around an area to draw multiple frames very quickly, but it divides each frame at visible gridlines. So each W10X17 beam is divided at each gridline B through F. Since the ‘Quick draw’ frame option draws and divides the frames at each grid intersection, we draw one strip at a time to avoid drawing beams along the Y direction at grid intersections.

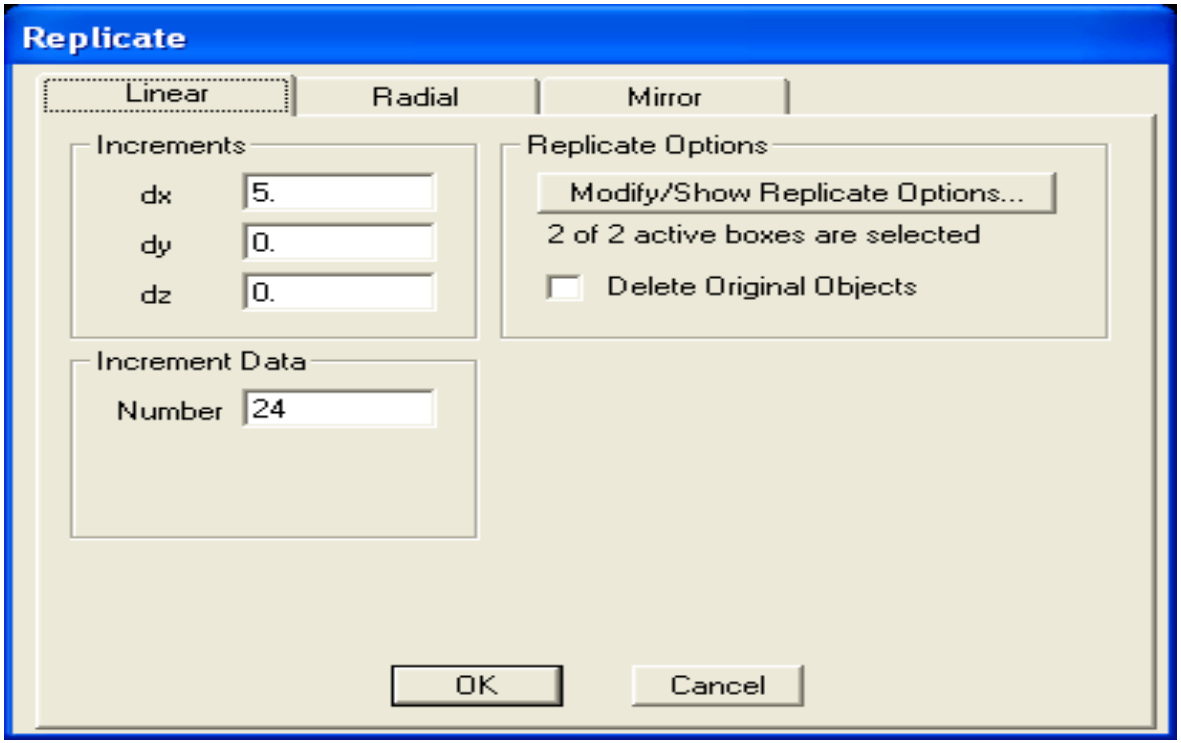




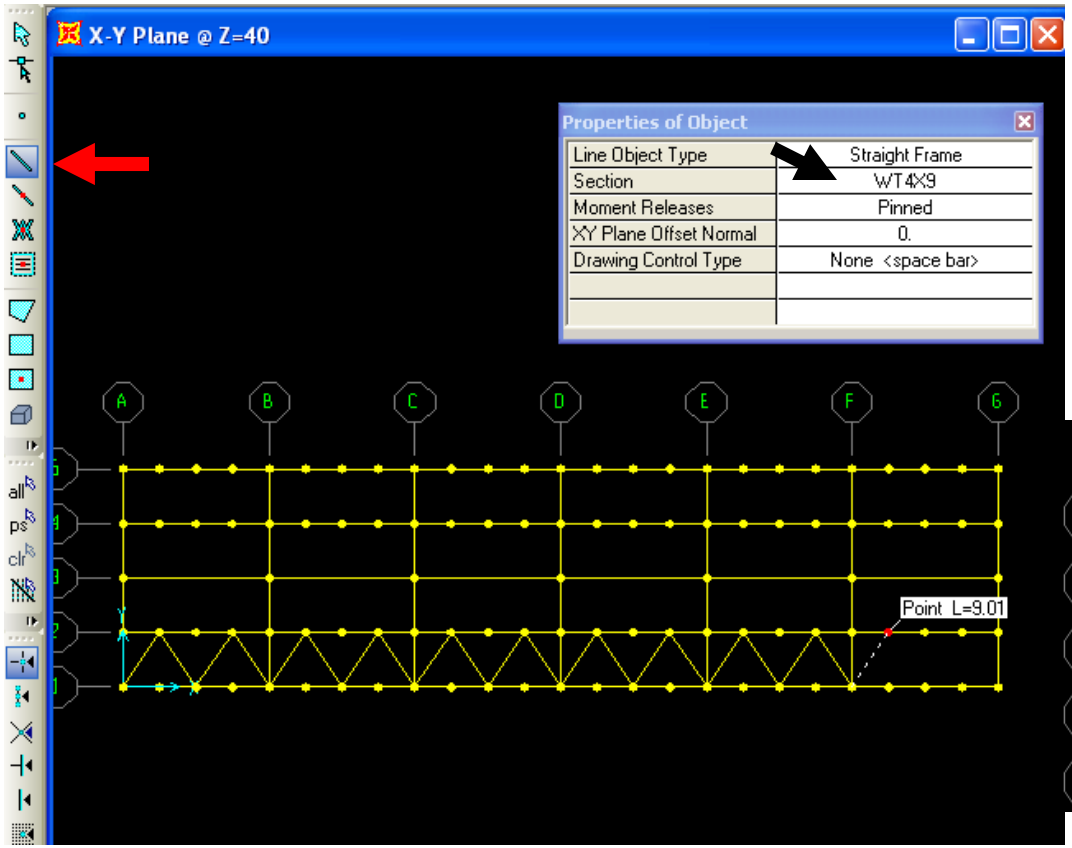


Next, we will replicate joints to make modeling of stitch bracing easier. In plan view, click each of the 4 joints shown below left to select, then **Edit>Replicate** to replicate 24 times (increments) 5 ft. each in the X direction. Press OK

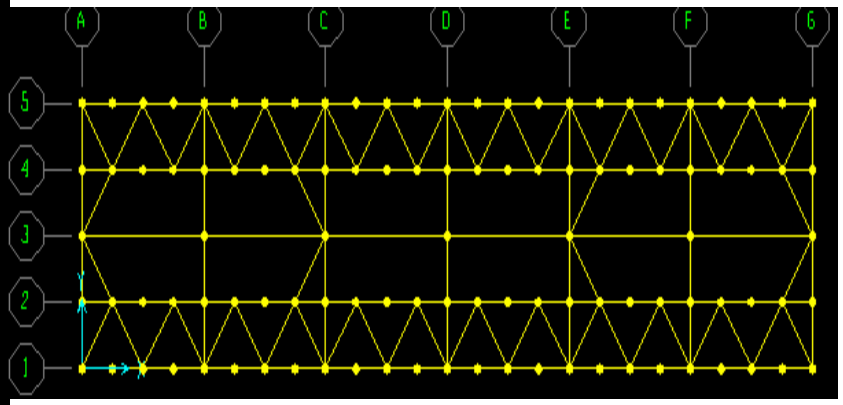




Click ‘Draw frame/cable element’, change section to WT4X9 Pinned as shown and draw the stitch bracing by “connecting the dots” as shown below. Remember to right click in order to move your cursor while in draw mode. The additional joints which were not used to draw bracing will not affect the model at all since they are associated with a frame/beam element object.

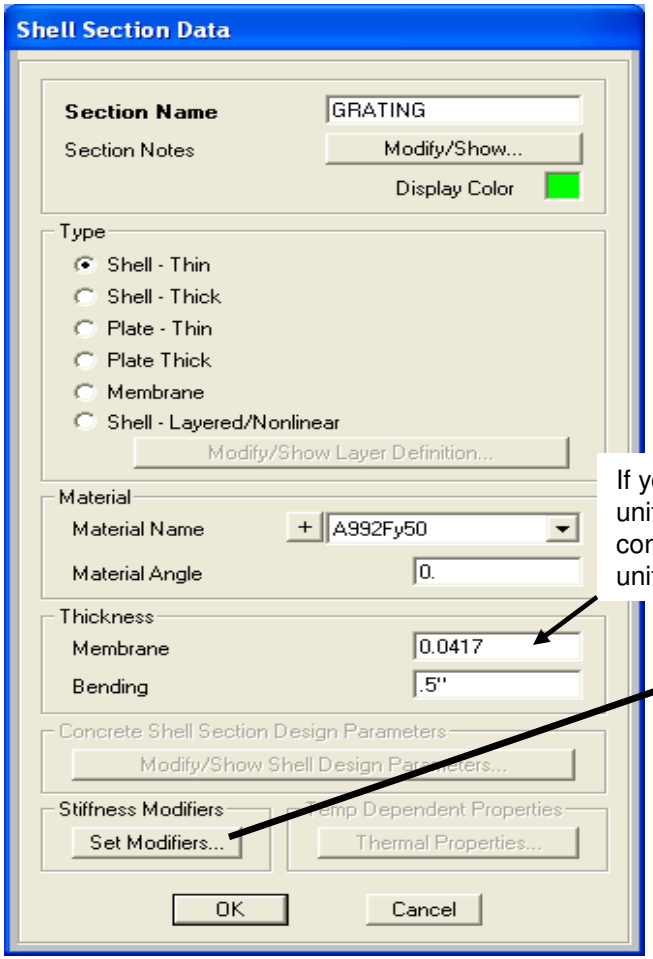


This is how the stitch bracing should look according to the PR1.pdf drawing



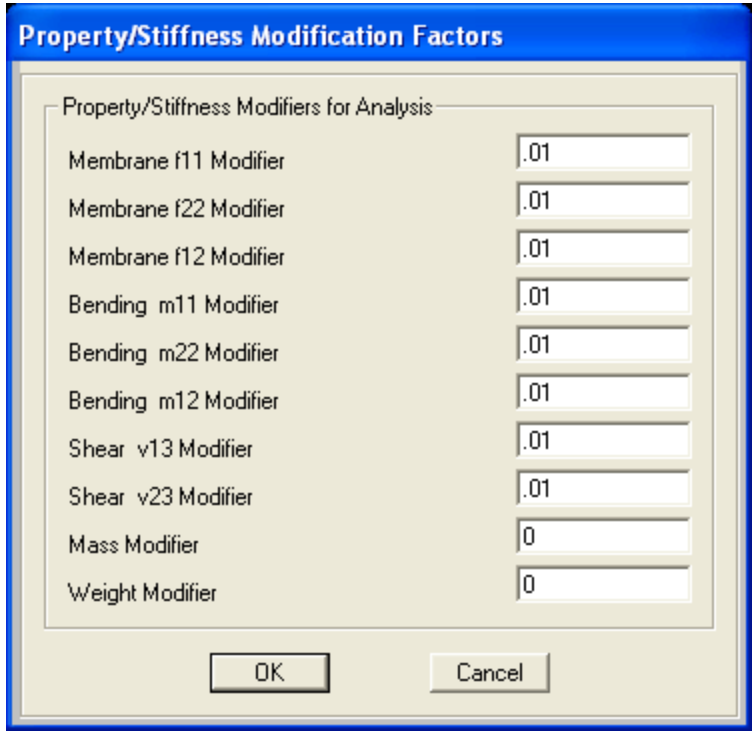
Next, go to **Define menu>Section properties>Area properties** to create a shell finite element for our grating. Change material to A992 steel and type .5" in the membrane and bending stiffness fields. Next, press the Set Modifiers button where we will zero out the weight and mass while minimizing the stiffness, since we do not want to consider the grating as part of our structural system, only a modeling tool to transfer area loads to beams while avoiding load distribution to bracing underneath. Type .01 in every field except Mass and Weight where you type 0 to zero out the weight and mass as we will assign grating self weight and live load manually based on vendor data and known loading.

Shell type area finite elements distribute loads only at meshed joint locations. We will not mesh this Grating shell at locations where there is bracing underneath to avoid distributing load to the bracing

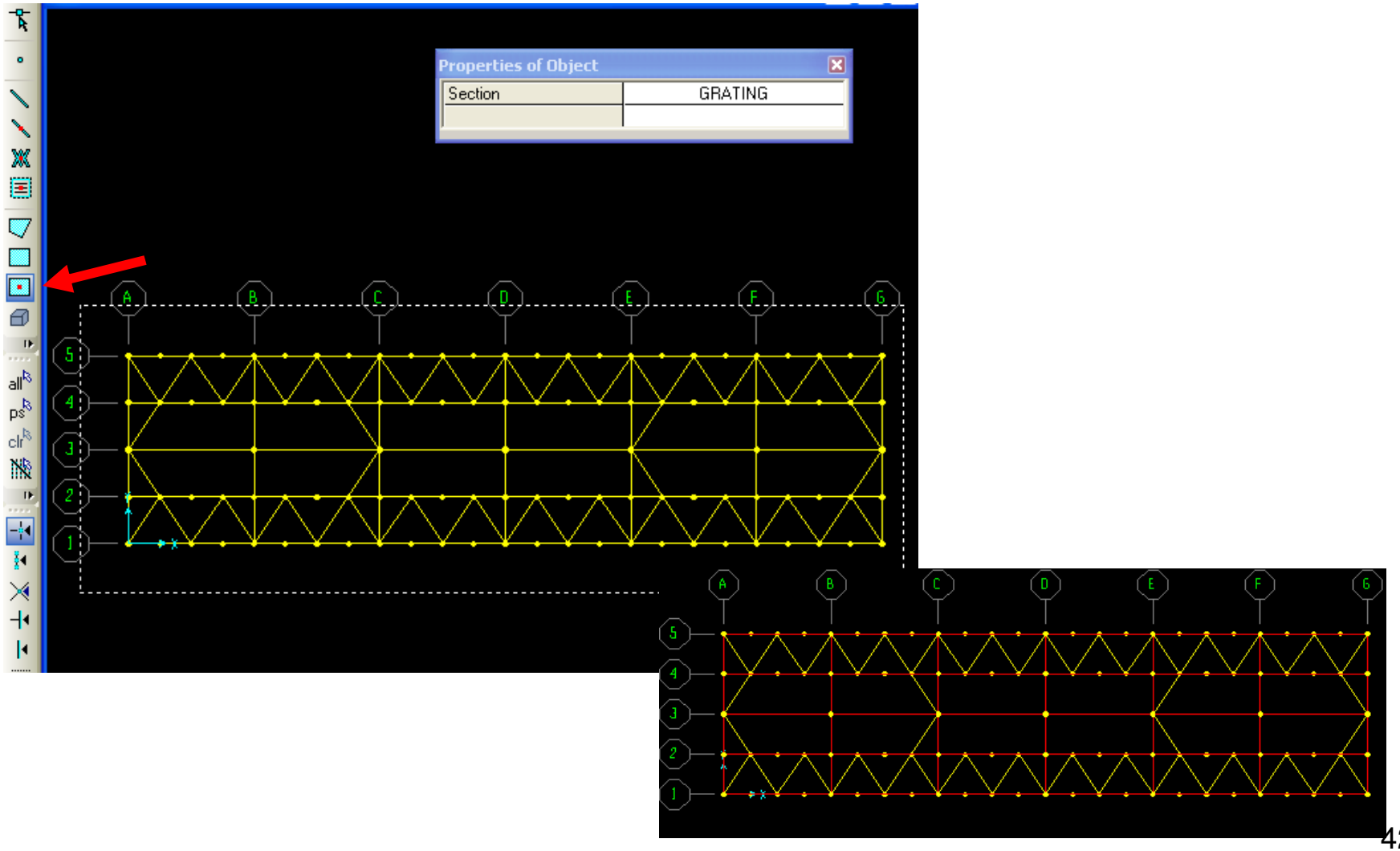


If you are in ft. units, 5" will convert to ft units

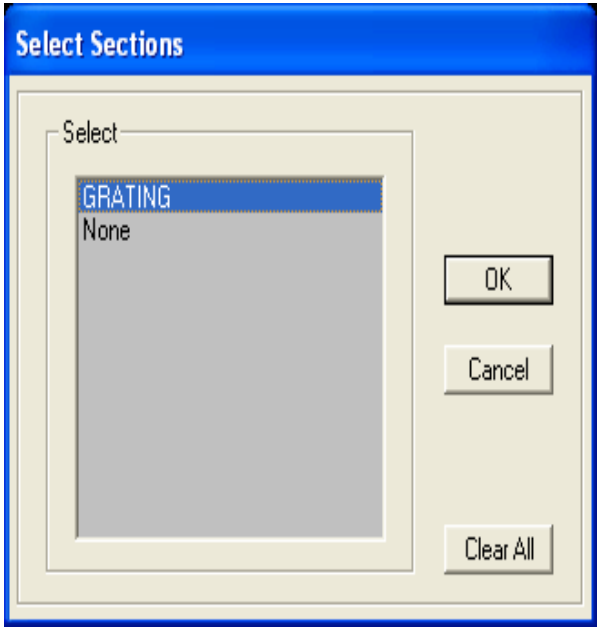
Use .01 to reduce section properties and zero the weight and mass

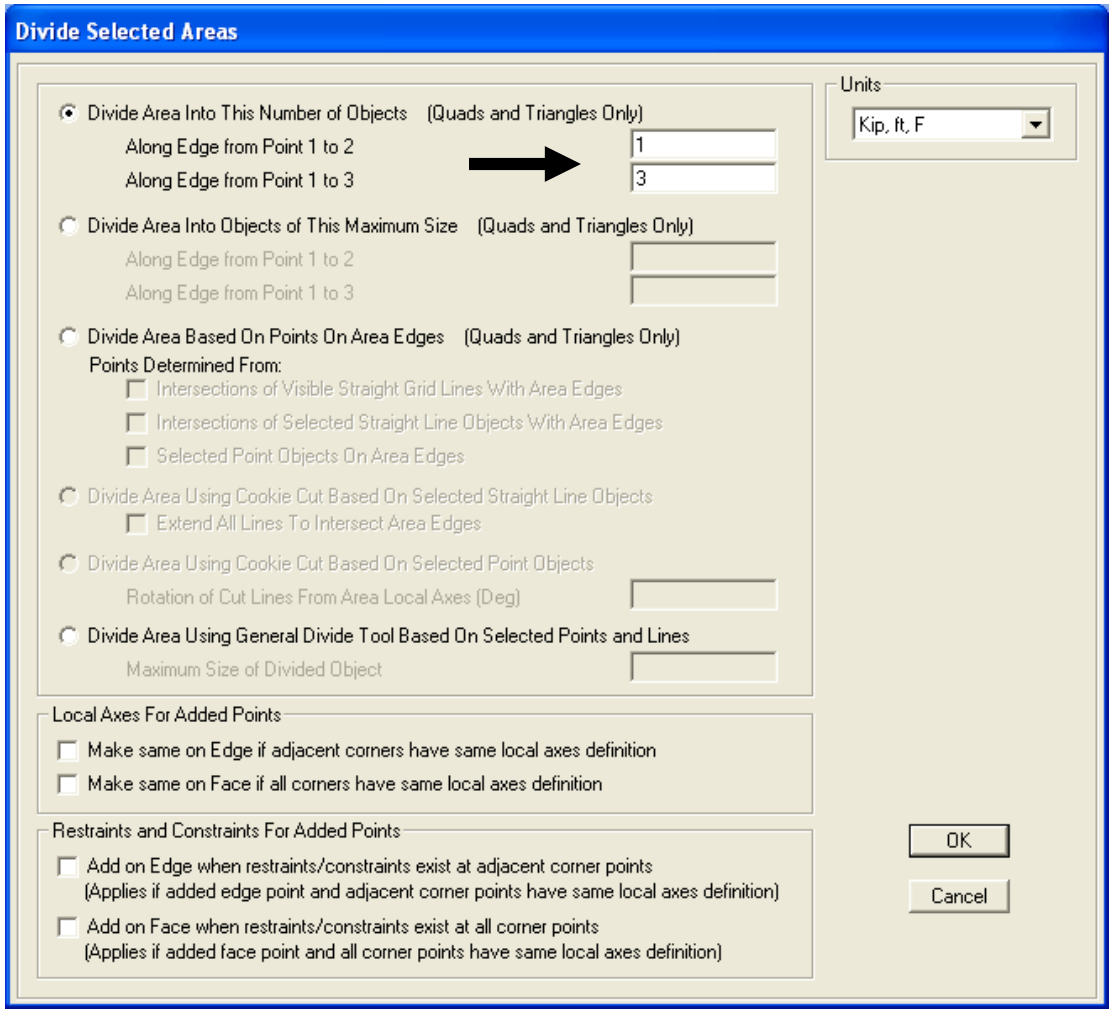


Click ‘Quick draw area element’ button as shown, making sure the section is the GRATING section we just defined and hold down your mouse and window around the entire top level in plan view as shown to draw grating. The Quick draw area element, like the quick draw frame tool, divides the elements at each visible grid intersection. Press Esc key on your keyboard or Select button to switch to select mode



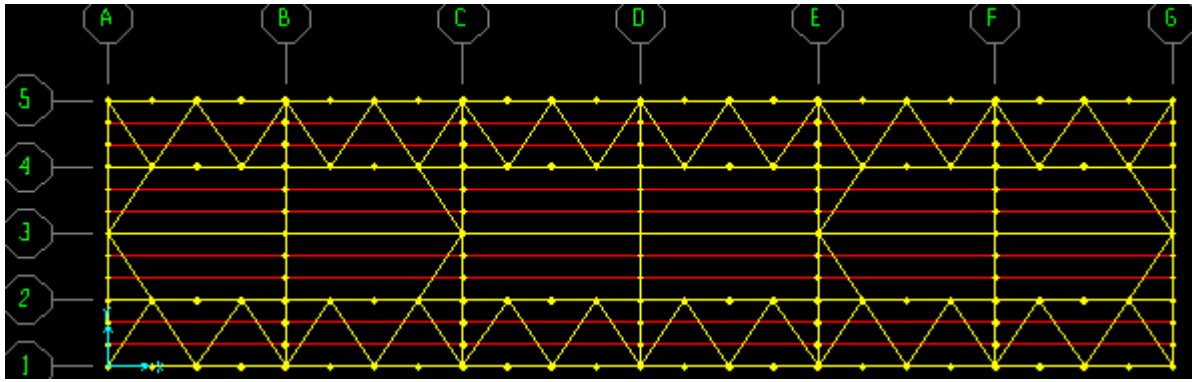
Use **Select>Select>Properties>Area sections** to select GRATING and click OK to select them. Next, use **Edit >Edit areas>Divide areas** and type in a 1 X 3 mesh as shown below in order to divide the mesh into thin strips forcing it to distribute load one-way and press OK. If we wanted the grating to distribute two-way, or one-way in the Y direction, we would have meshed differently or not meshed at all. This exercise helps demonstrate how users can control load distribution.

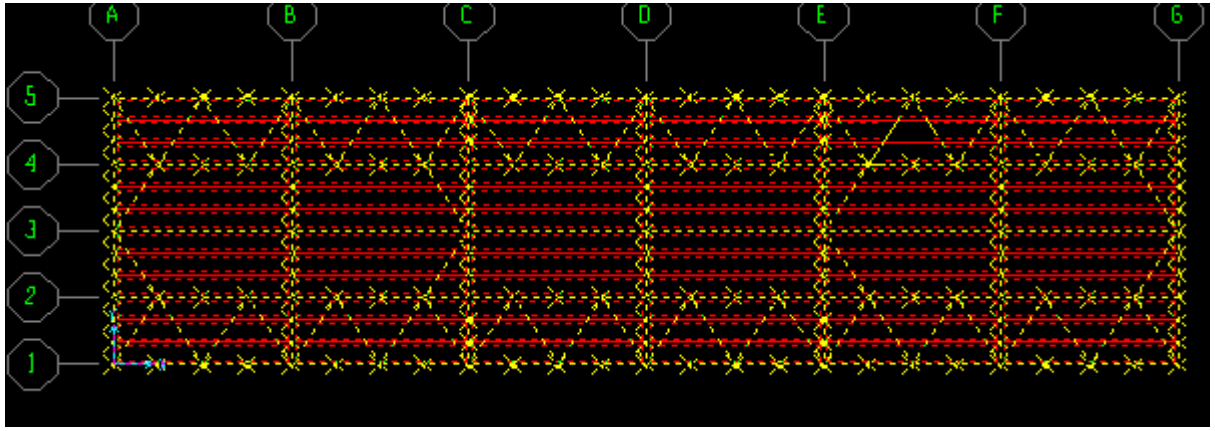




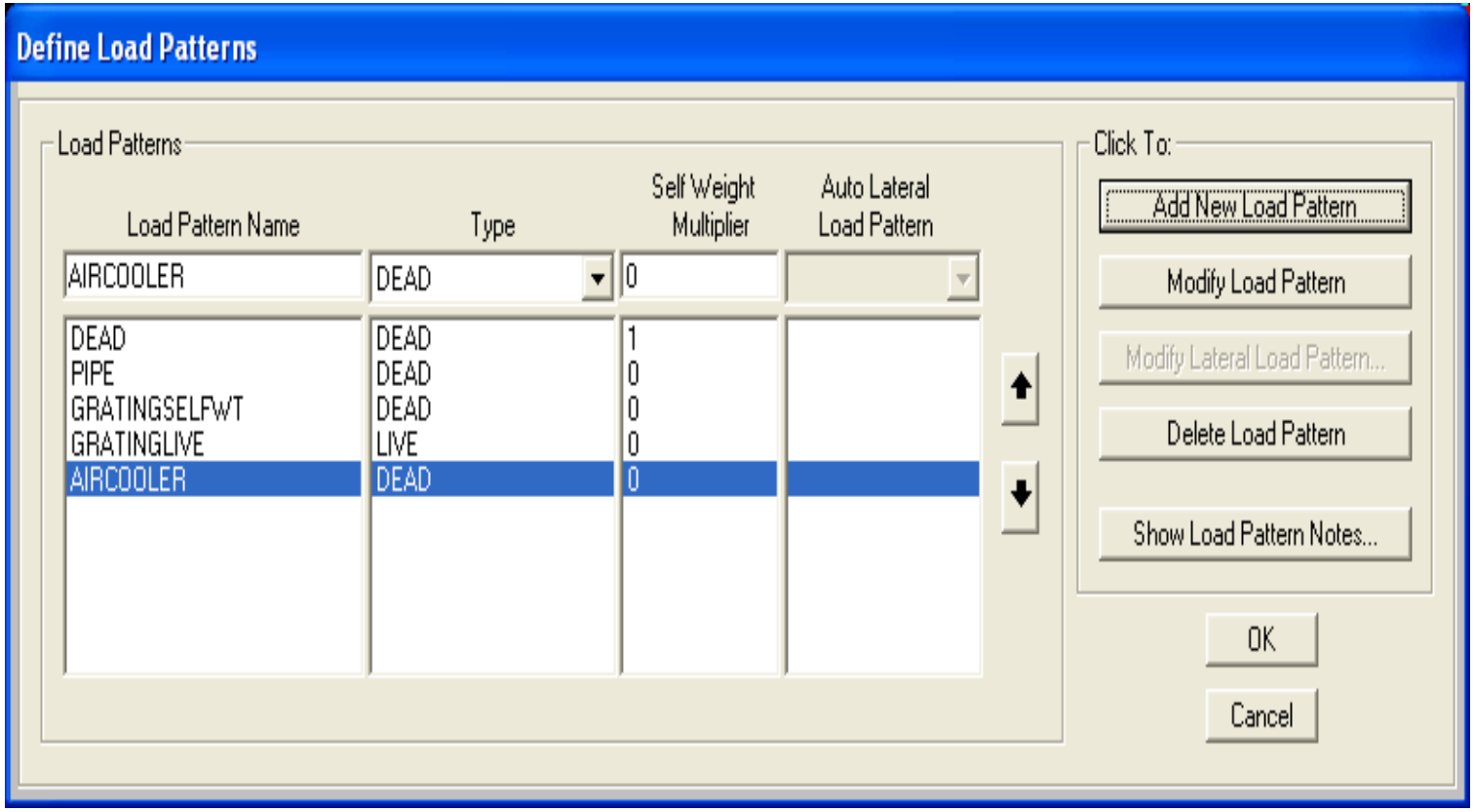
Below is what your plan view should look like now. Next, window around the entire plan view to select area elements in order to assign loads. Don't worry if you also select frame elements and joints along with the areas.

With SAP2000, you can assign area loads before or after meshing. SAP2000 keeps track of area load assignments no matter how the areas are meshed afterward.

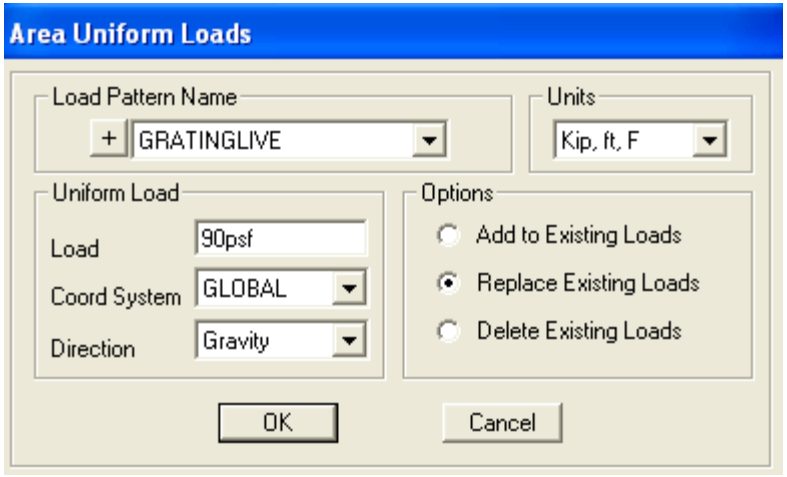


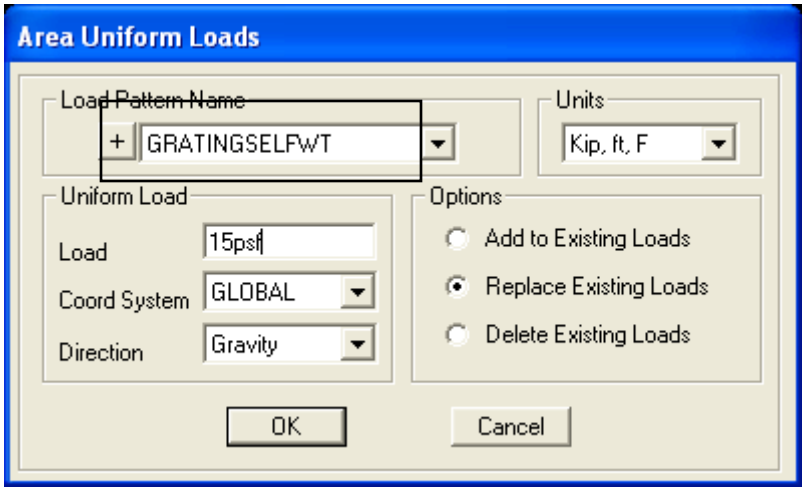


Go to **Define>Load patterns** and define load patterns (which used to be called ‘load cases’ in previous versions) as shown below and press OK. Earlier we added the PIPE load case while assigning loads. This is the more conventional way of defining load patterns. Careful not to have more than 1 load case with self weight multiplier of 1 or you will double count or triple count the self weight of the structure.

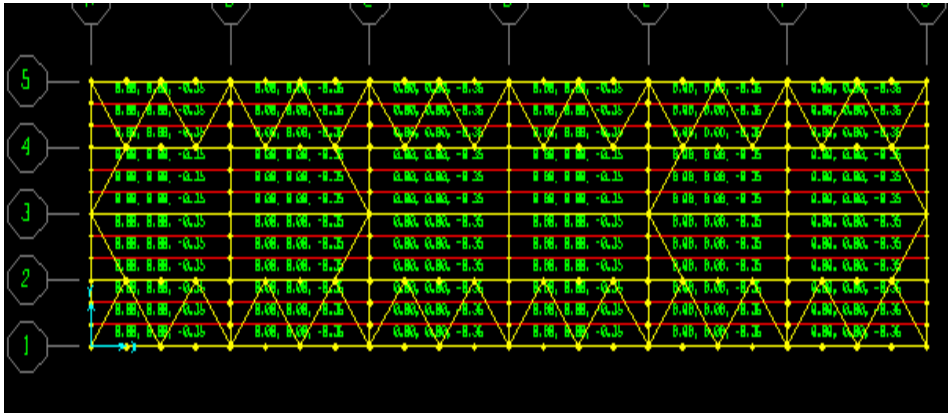


Next, go to **Assign>Assign area loads>Uniform (shell),** change load pattern name to GRATINGLIVE and type 90psf in the load field and press OK. SAP2000 will automatically convert to current units. Click previous selection button to reselect and **Assign> Assign area loads>Uniform (shell),** change load pattern to GRATINGSELFWT enter 15psf and press OK. This is a faster, less error prone method of load distribution as compared to assigning the grating loads as distributed frame loads based on tributary area. Press F4 to clear screen of load assignments if the screen appears cluttered. Save model and name it SAPTutorial. It's a good idea to save your model often

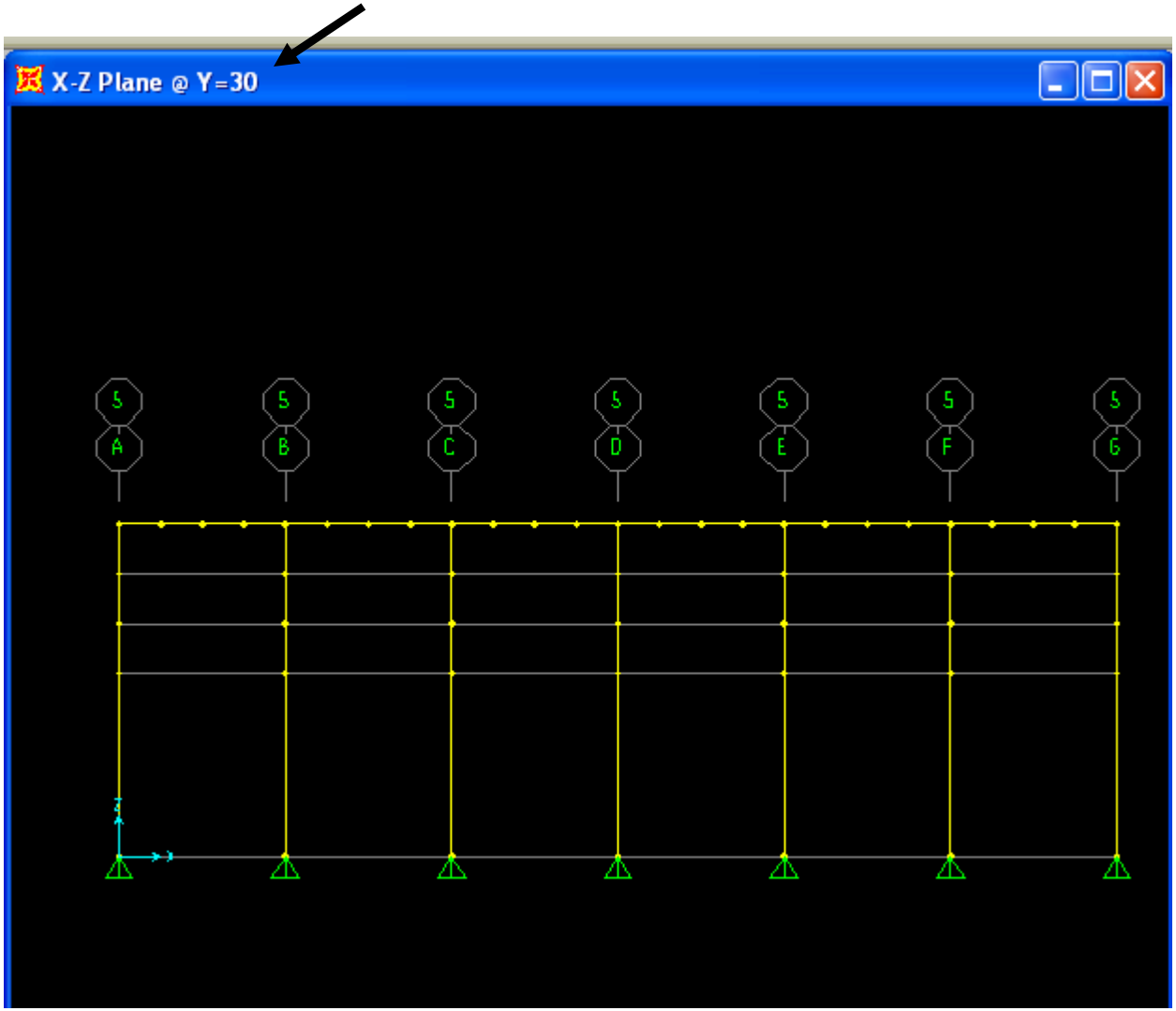




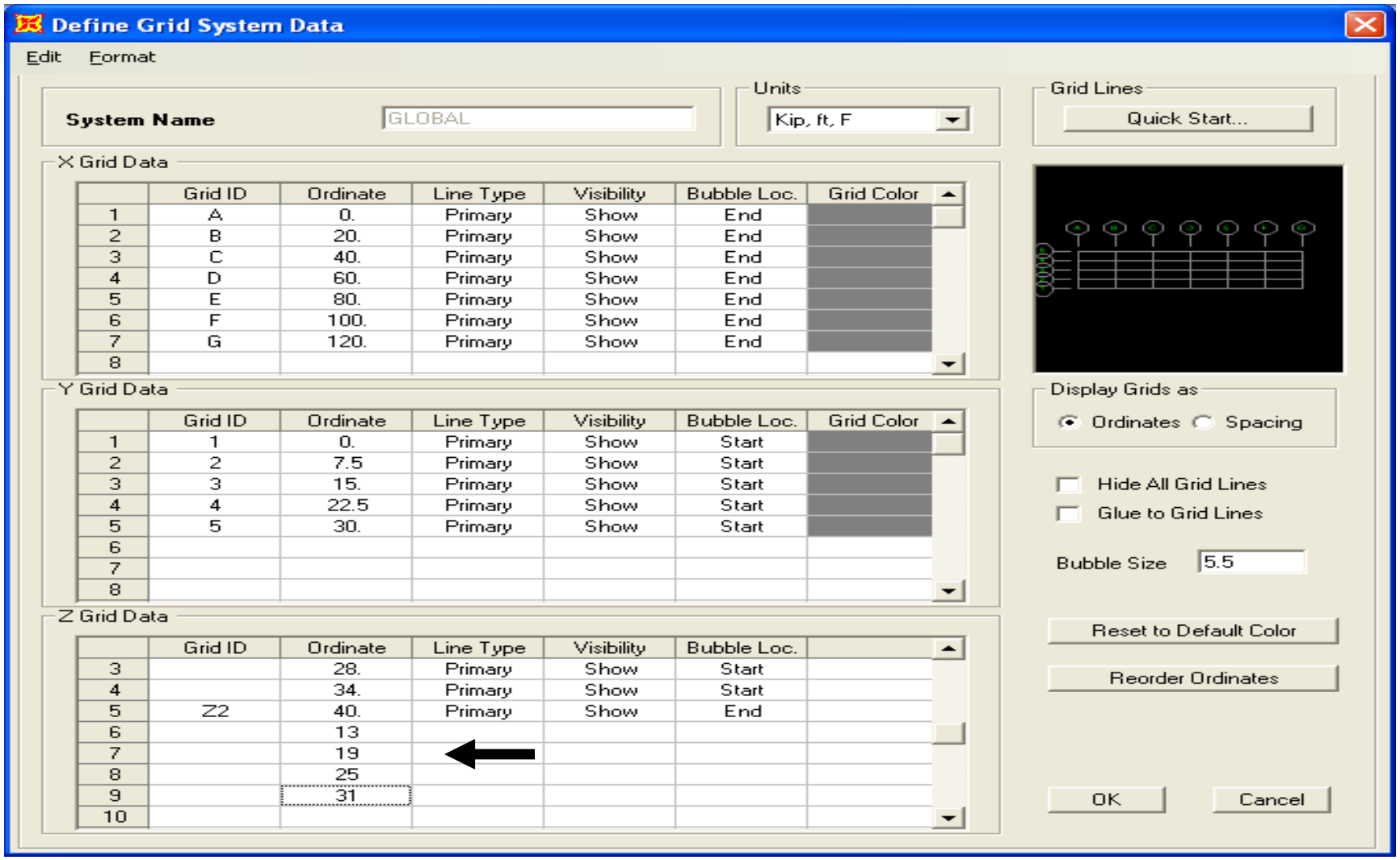
Press F4 to clear screen of clutter from assignments



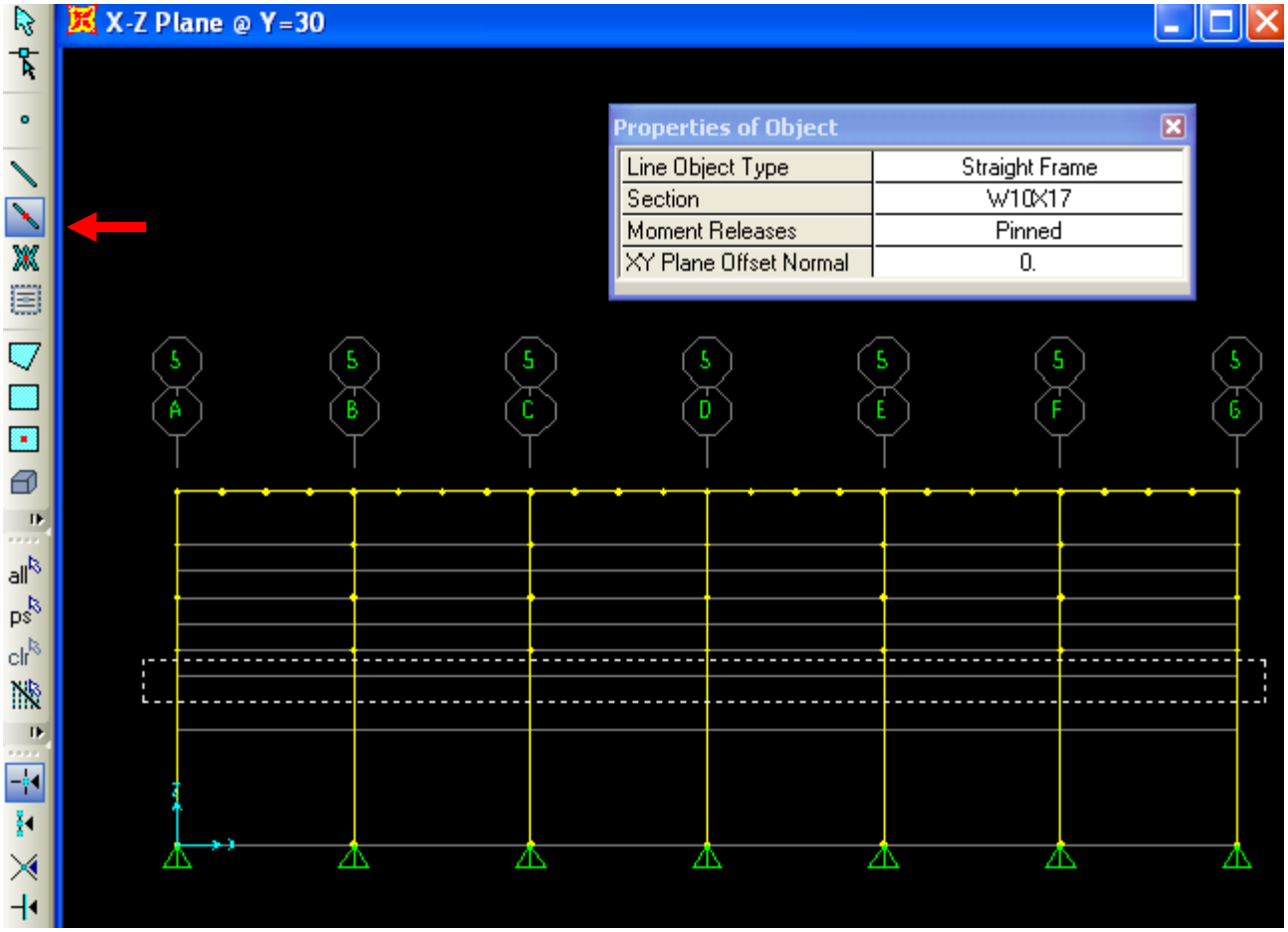
Next, we will add side beams and bracing, so we need to add gridlines to facilitate modeling. Click anywhere in your plan view window to make sure that it's the active window and press xz button. You may need to use up/down arrow keys to move to Y=30 ft. Double click any gridline to display the Define grids dialogue



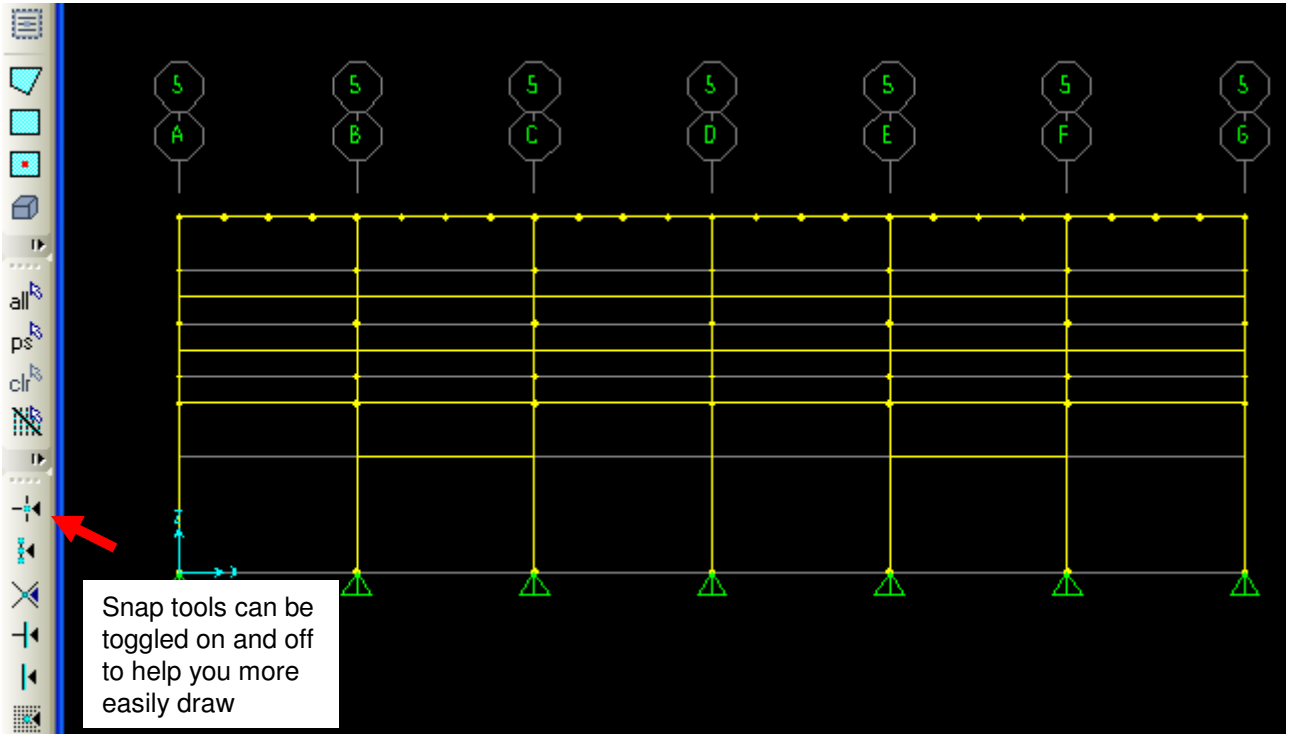
Add gridlines in the Z direction for 13, 19, 25, and 31 feet as shown to facilitate drawing the new beams and bracing. SAP2000 will sort out the sequence when you press OK



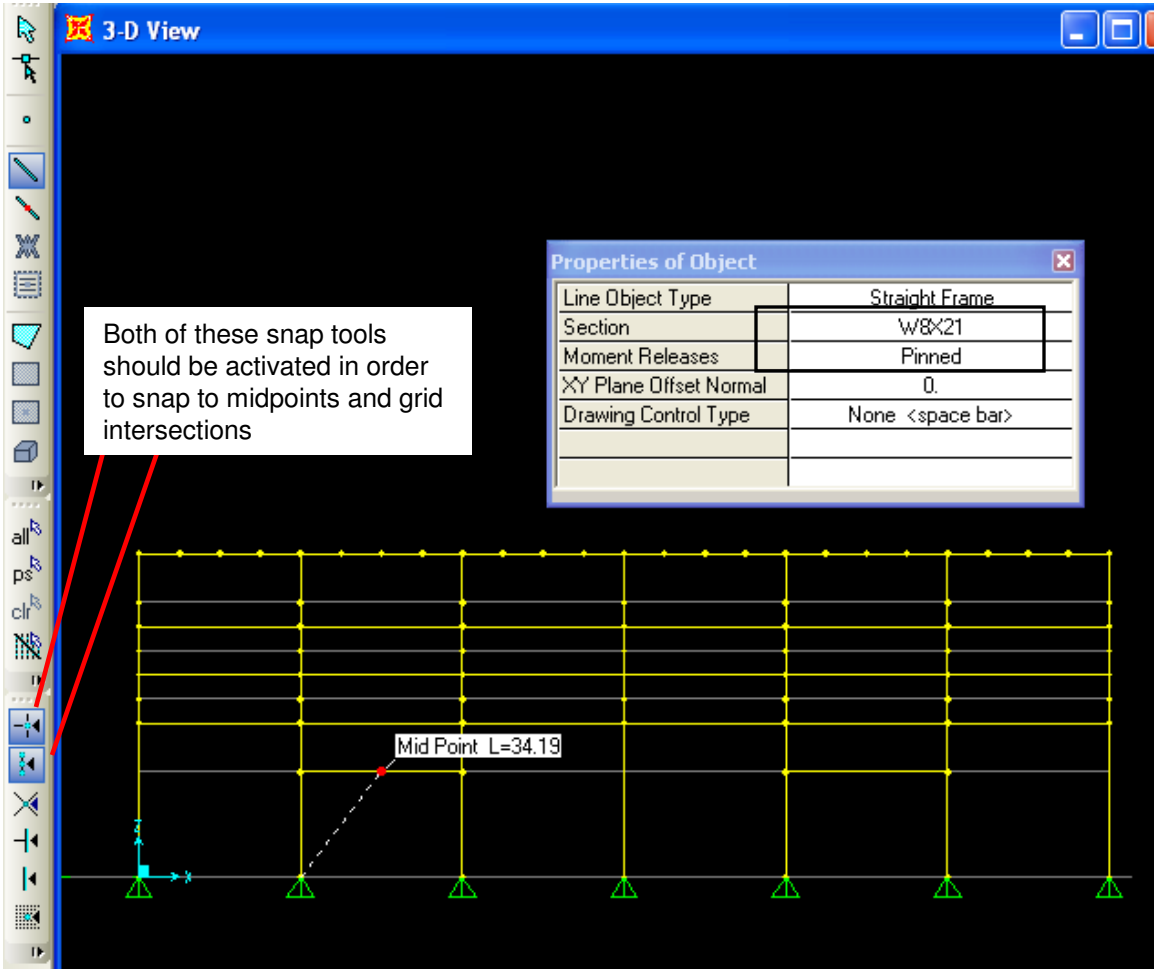
No sections were specified on the drawing for the side beams and bracing, so we will use W10X17 beams and W8X21's for the bracing. Alternatively, we could have assigned an Autoselect list in order to have the program optimize for lightest section. Click the 'Quick draw frame' option and window around the gridline at Z elevation 19 ft. per the drawing. This will draw 6 frames/beams inside the window area, dividing them at each visible gridline.

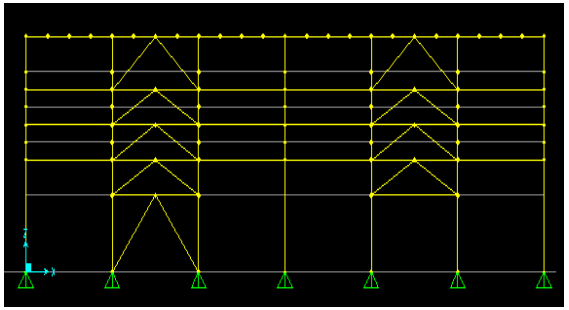


Repeat this draw technique at elevations 25 ft and 31 ft. as well as at 13ft in the two bays between 5B and 5C and 5E and 5F. With gridlines so close together, it may be easier to toggle off your default ‘snap to points and grid intersections’ button temporarily by clicking it.

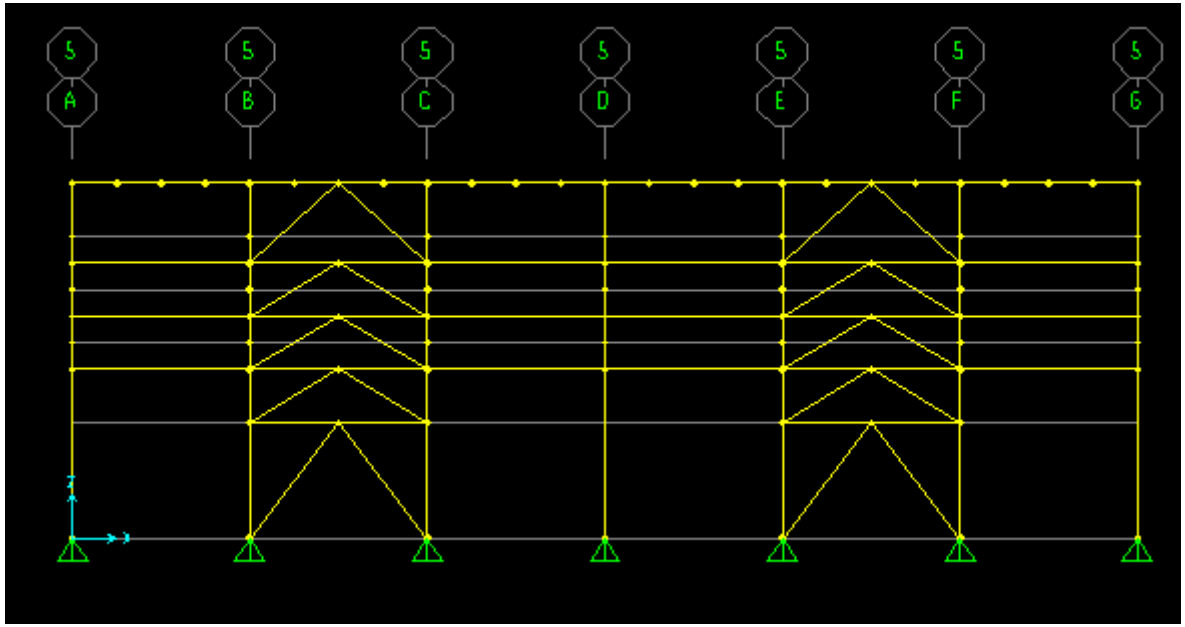


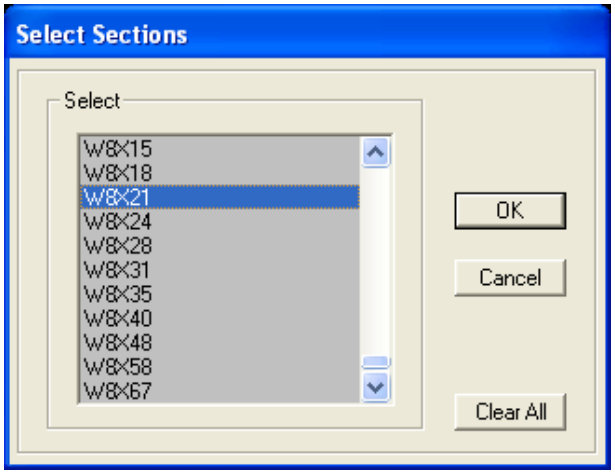
Click to activate snap tools for snap to ‘points and intersections’ as well as the ‘ends and midpoints’ snap tool just below it. Click ‘Draw frame/cable element’ button, change the section to W8X21 Pinned and draw bracing as shown below. SAP2000 will create joints internally to connect the bracing to the beams to transfer shear, but SAP’s object based modeling will reformulate the output in order to report and design the beams as they were drawn without dividing the beams into two pieces in the output.



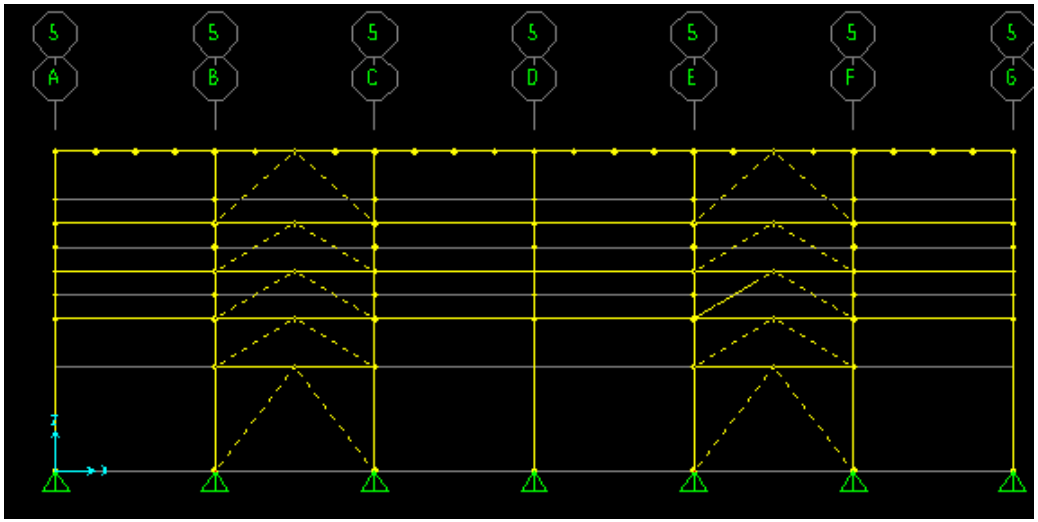


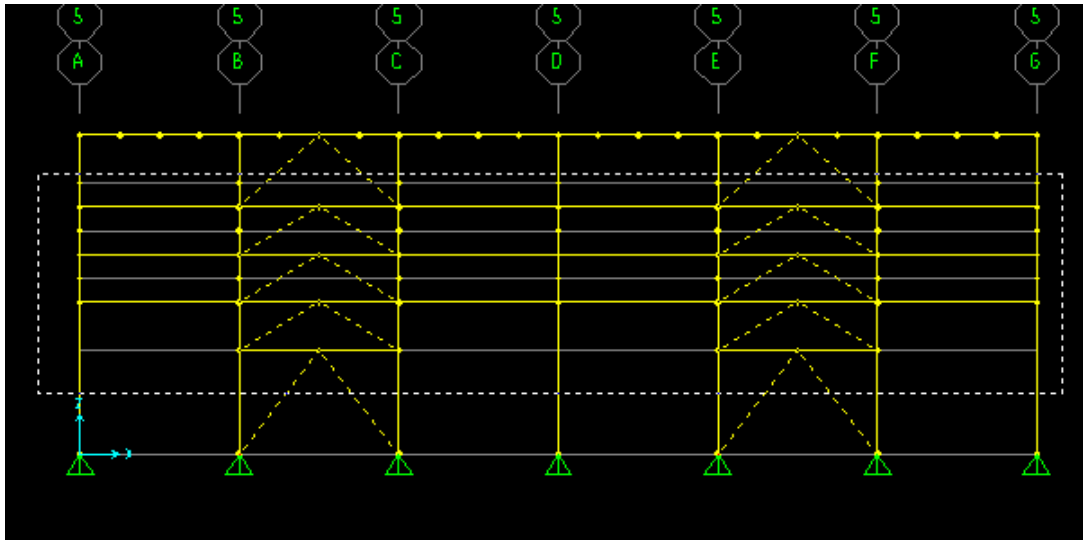
Next, we will select these newly drawn beams and braces in order to replicate them to the other side. Press Esc key or the select button to switch into Select mode, then use **Select>Select>Properties>Frame sections** and select W8X21 as shown. Press OK

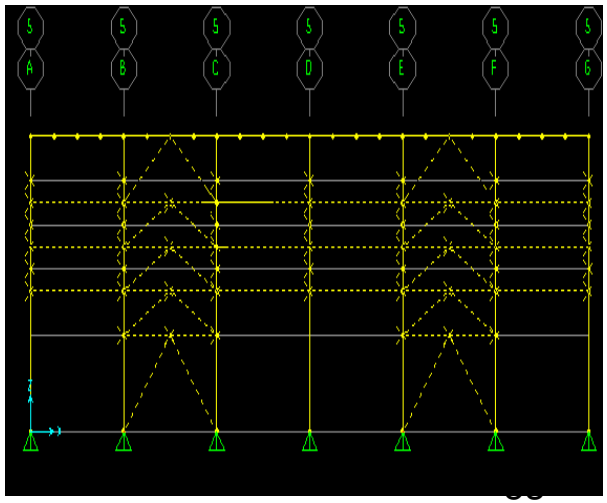




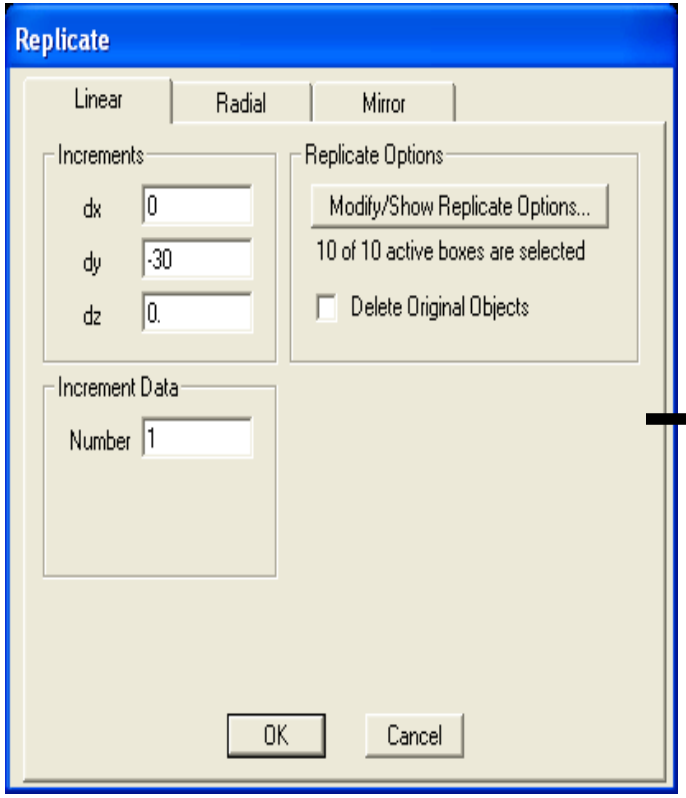
The first graphic shows the W8X21's's selected. Next, window around the area moving the mouse left to right as shown below in the bottom left in order to select the beams without selecting the columns. Note that this selection technique selects some additional joints, but replicating joints won't hurt anything as along as the joints are associated with a frame and not disconnected out in space.

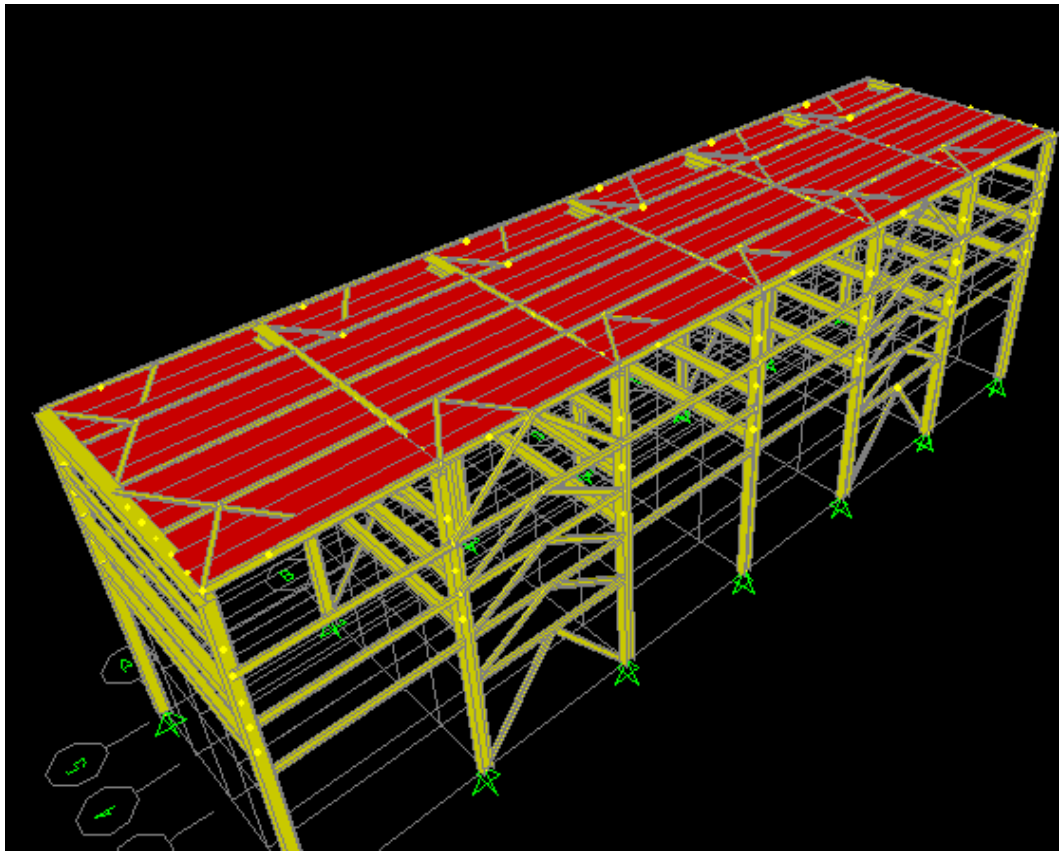




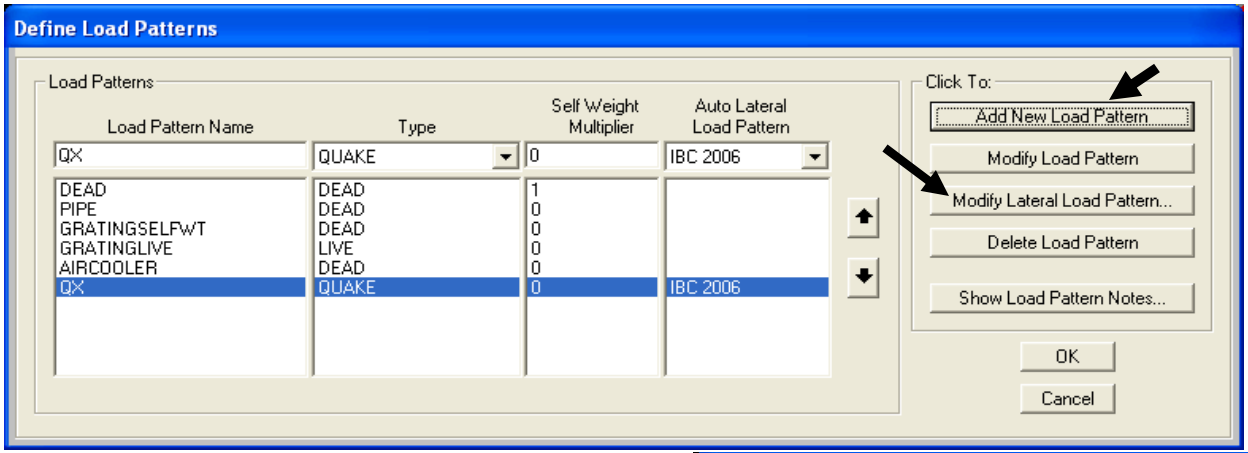


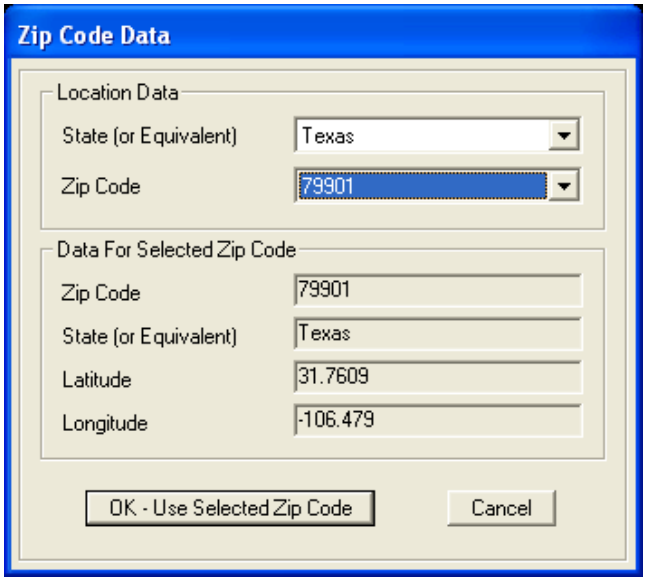
Next, use **Edit>Replicate** in order to copy the selected beams and bracing to the other side, 1 increment 30 ft. in the –Y direction (-30 dy) as shown below. Next, we will add an IBC2006 seismic load and a wind load in order to demonstrate SAP2000's auto-lateral load generation.

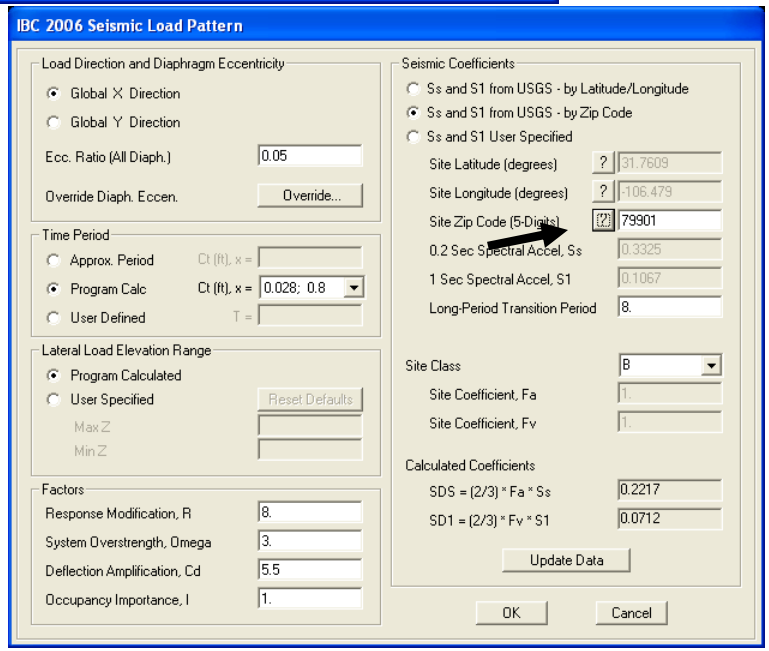




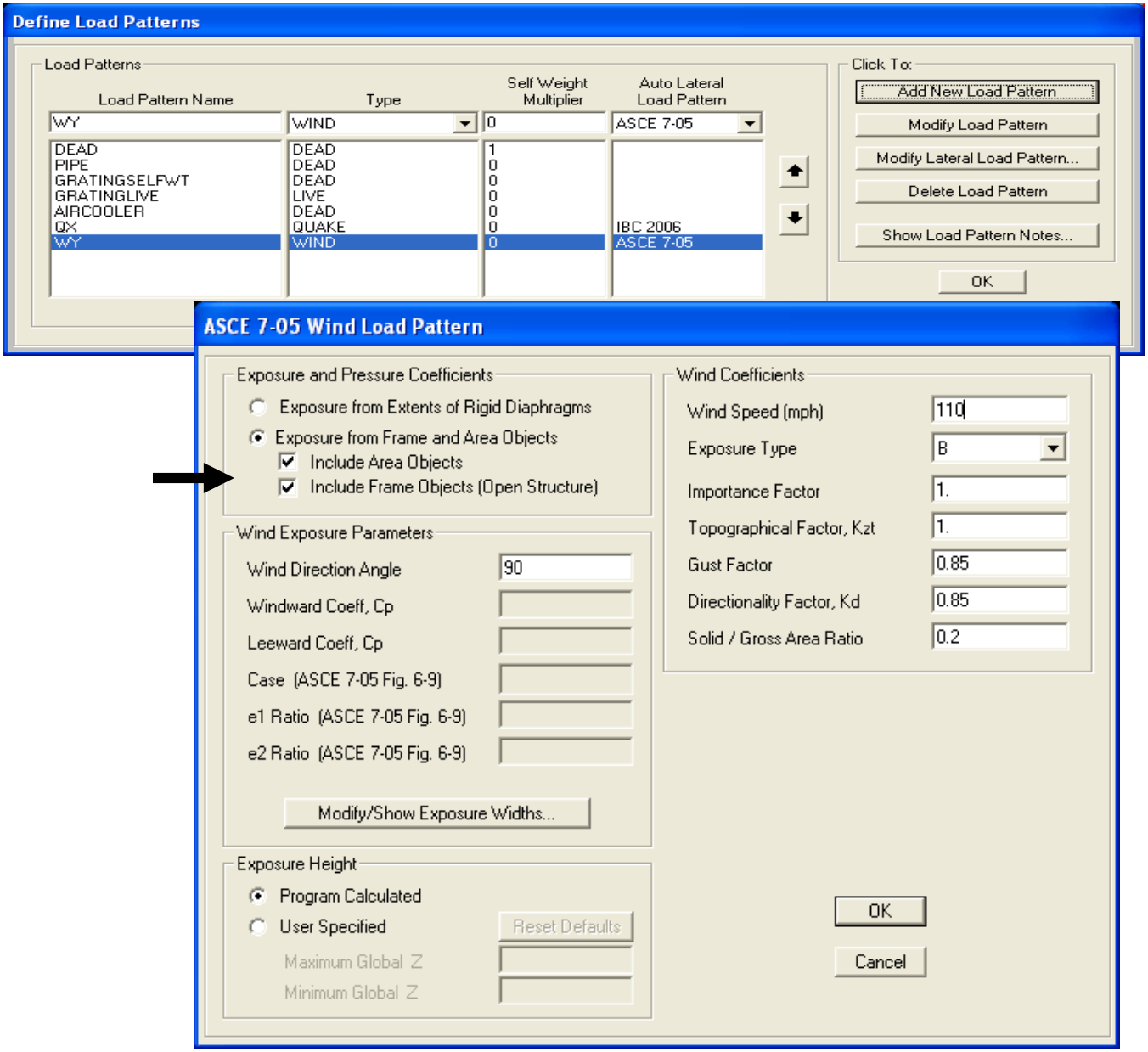
As an alternative to manually calculating seismic and wind loads applied as joint loads or distributed loads, SAP2000 offers options for automatic generation of wind and seismic loads per IBC, ASCE and other standards. Go to Define>Load patterns, add a load case named QX, type Quake, Auto lateral load IBC2006 as shown below and press ‘Add new load pattern’ button to add it, then press ‘Modify Lateral load pattern’ to specify IBC parameters. Select option to lookup Ss and S1 values based on zip code using 79901 zip code below for El Paso, TX. Press OK until you return to the Define load pattern dialogue



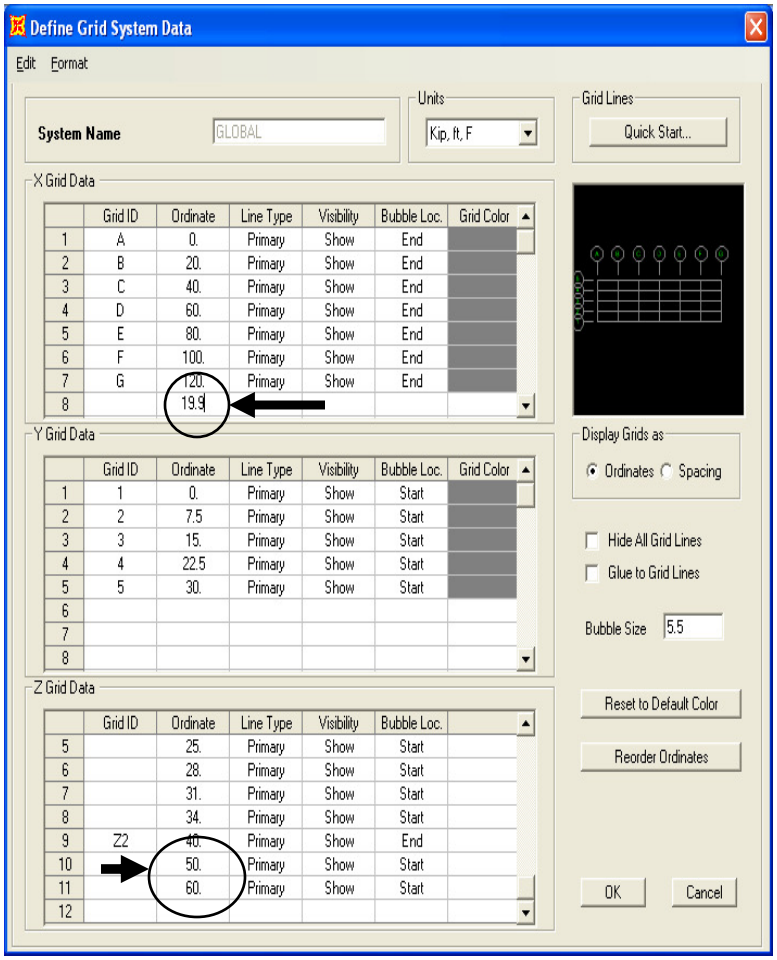




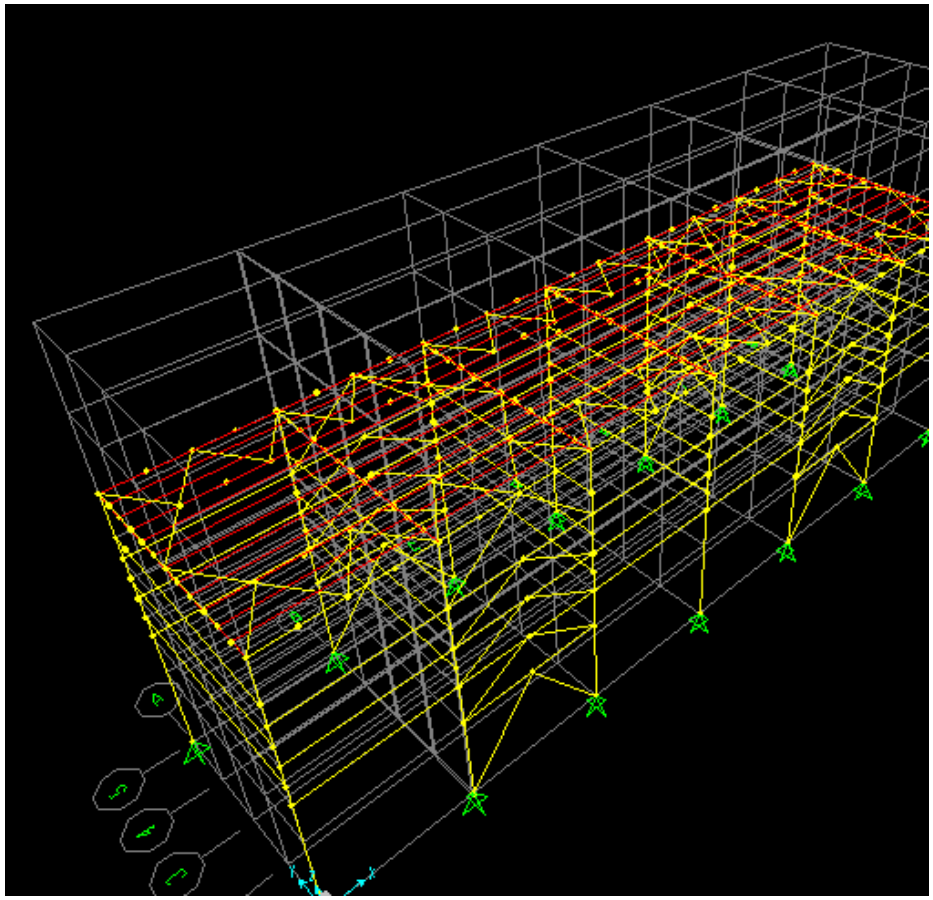
Next, add load pattern named WY type Wind using Auto Lateral load pattern ASCE 7-05 and press ‘Modify Lateral load pattern’ to specify parameters. Change the default to ‘Exposure from Frame and area objects’ Include Area objects which we will add later for the air cooler and include Frame objects (open structure). Wind direction angle 90 degrees (Y direction) for the wind load generation on the frame elements, 110 mph and press OK. Later we will assign wind load coefficients to area objects which simulate the dimensions of the air coolers



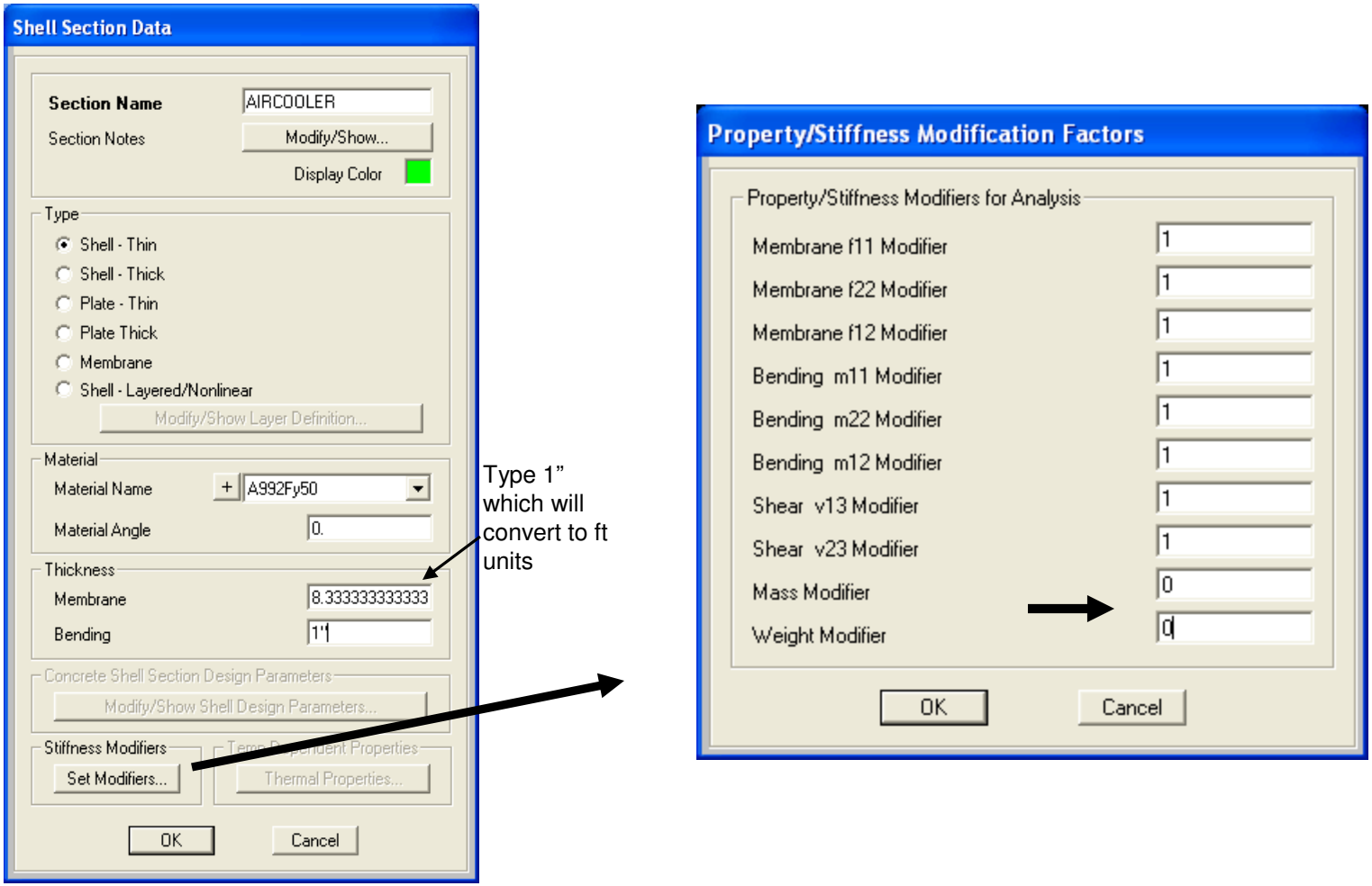
Next, as a modeling technique we will add “dummy” area elements and joint loads to simulate air cooler wind exposure. Double click a grid to display Define grid dialogue and add Z grids at 50 ft and 60 ft. and an X grid at 19.9 ft. The purpose of the 19.9 X grid is to create aircooler areas for wind load that are disconnected from one another, as we will replicate them 20’ apart and keep them separate. Press OK to add the grids.



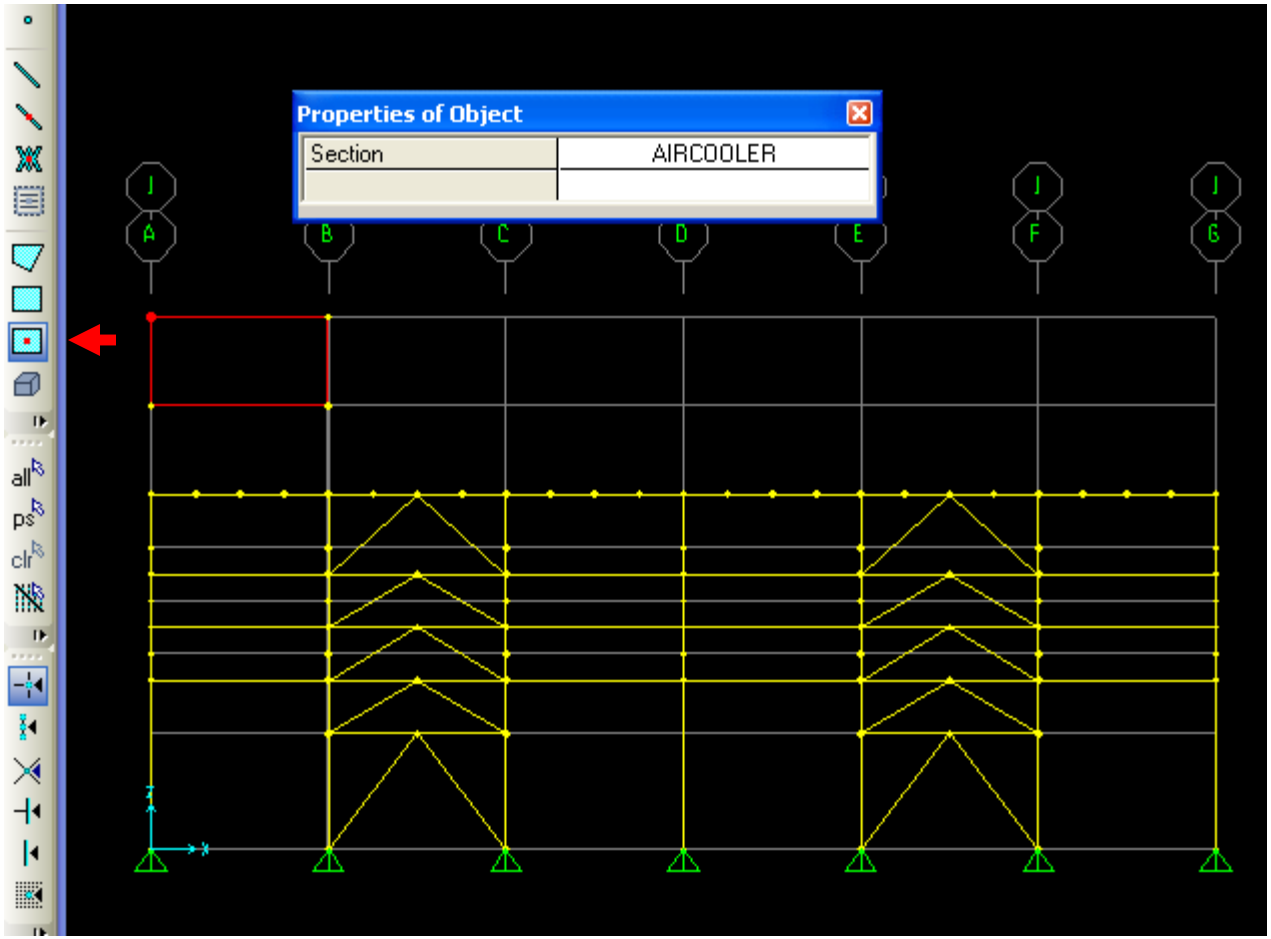
With new gridlines added



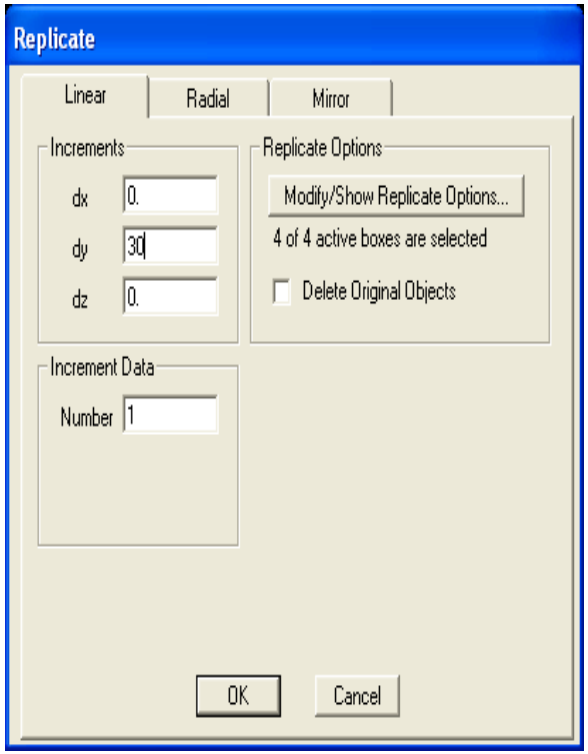
Go to Define>Section Properties>Area sections and Add new section, name it AIRCOOLER, material A992, with both membrane and Bending thickness 1". Next, click 'Set Modifiers' button and enter 0 for mass and weight as we will assign the aircooler weights later at joint locations

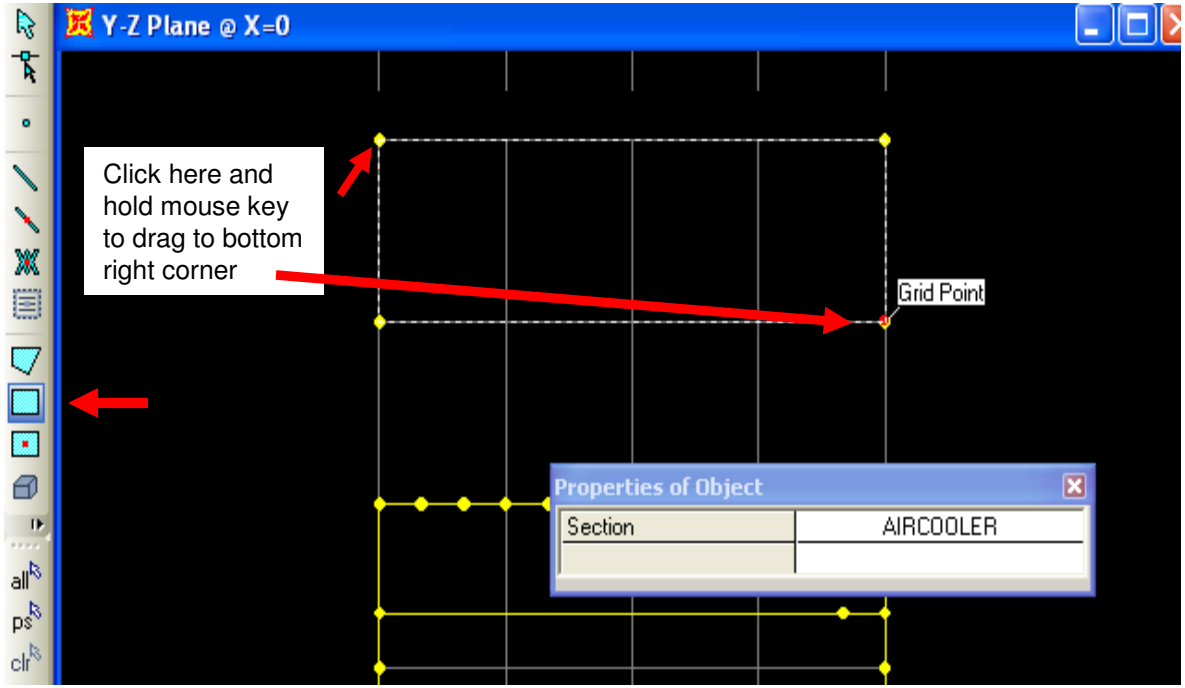


Switch your left side planar view to xz elevation view Y=0. Next click ‘Quick draw area’, change section to AIRCOOLER, and click just once in the upper left area shown below to the right of 1A. The addition of the 19.9 ft gridline keeps this area disconnected from the next bay by .1 ft. We will assume that the air cooler will catch wind load between 40’ and 50’ and transfer it to the structure below

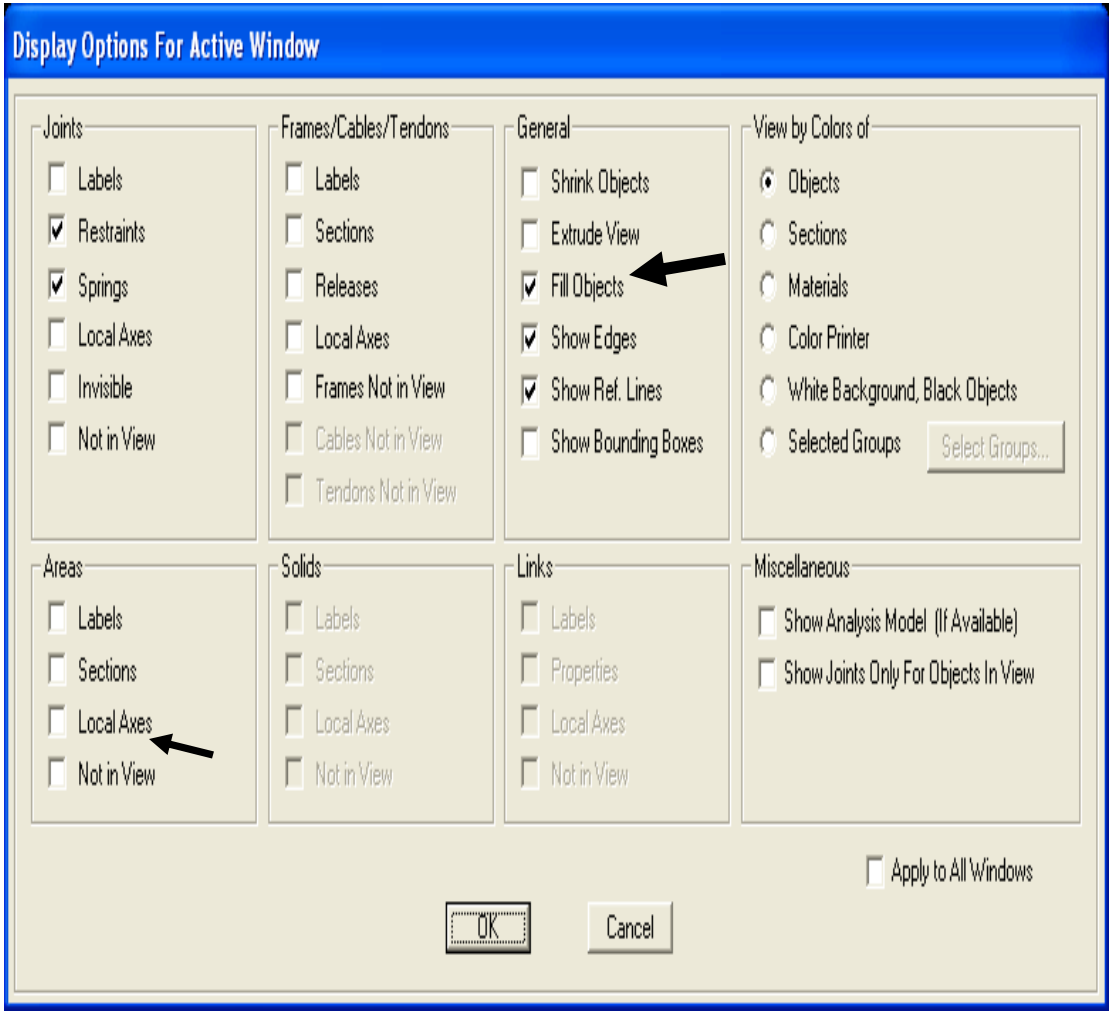


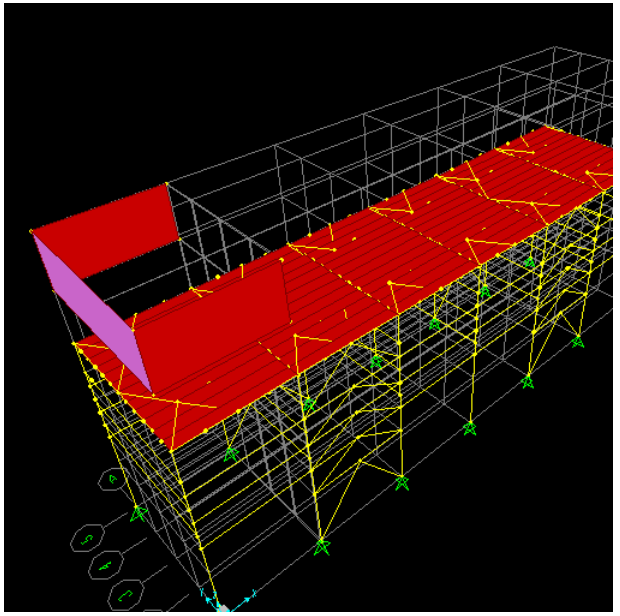
Click to select this AIRCOOLER area and Edit>Replicate as shown 30’ in the Y direction to copy it to the other side. Next, switch your planar view to yz yz with X=0, click ‘Draw rectangular area element’ and then click once on upper left corner and drag it down to the right corner as shown below. This draws 1 area element without dividing it at every visible gridline



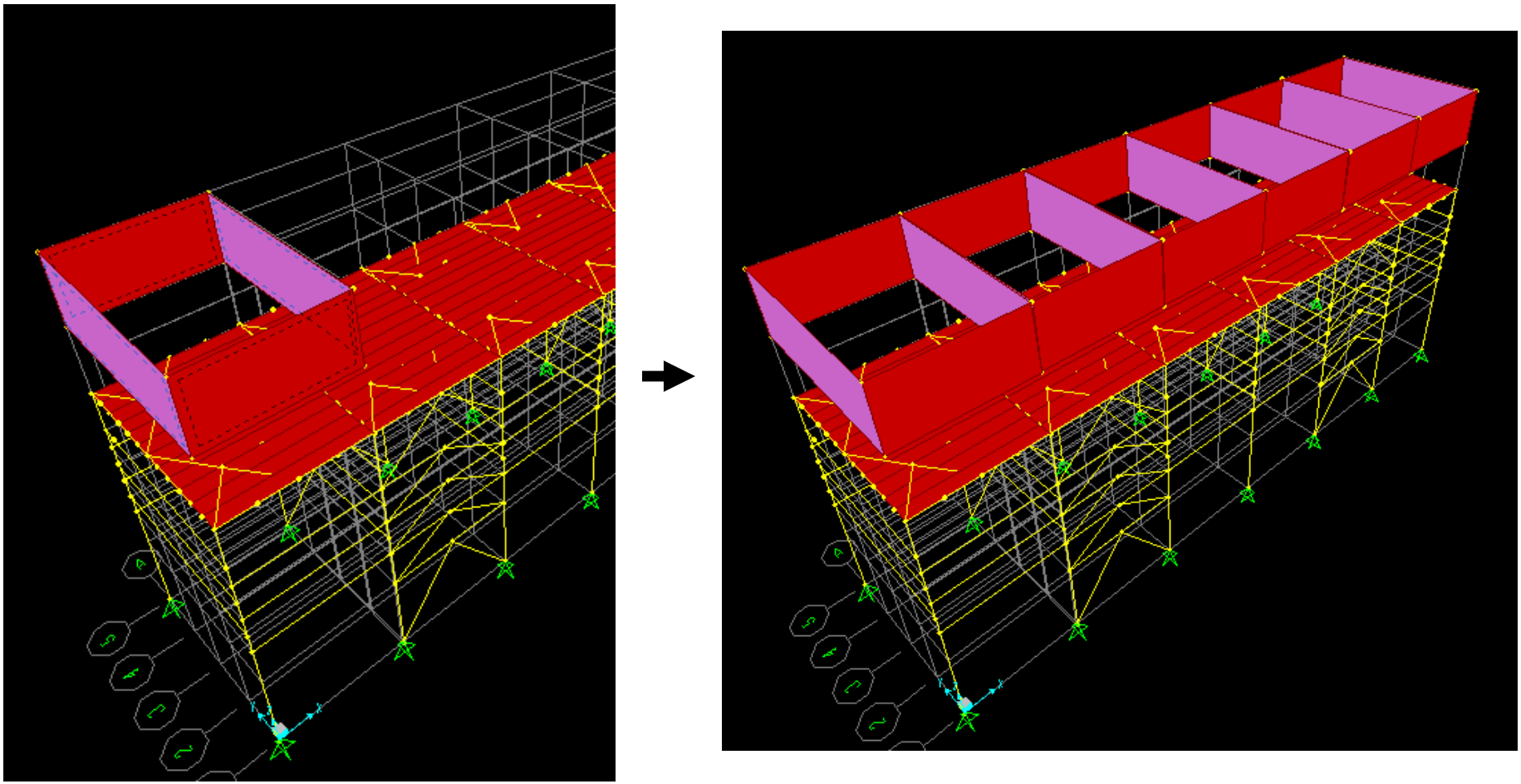


Use ‘Set display option’ to activate ‘Fill objects’ view and press OK. The color of the area objects indicate the local 3 axis. The pink color shown below right indicates that the local +3 direction for that area element is = +X direction since +3 is perpendicular outward and away from the red face whereas -3 is outward and away from the pink face. When assigning area wind load coefficients, you must know the local 3 direction. Select the area shown below in pink and replicate it 19.9 ft one time in the X direction to complete the air cooler box

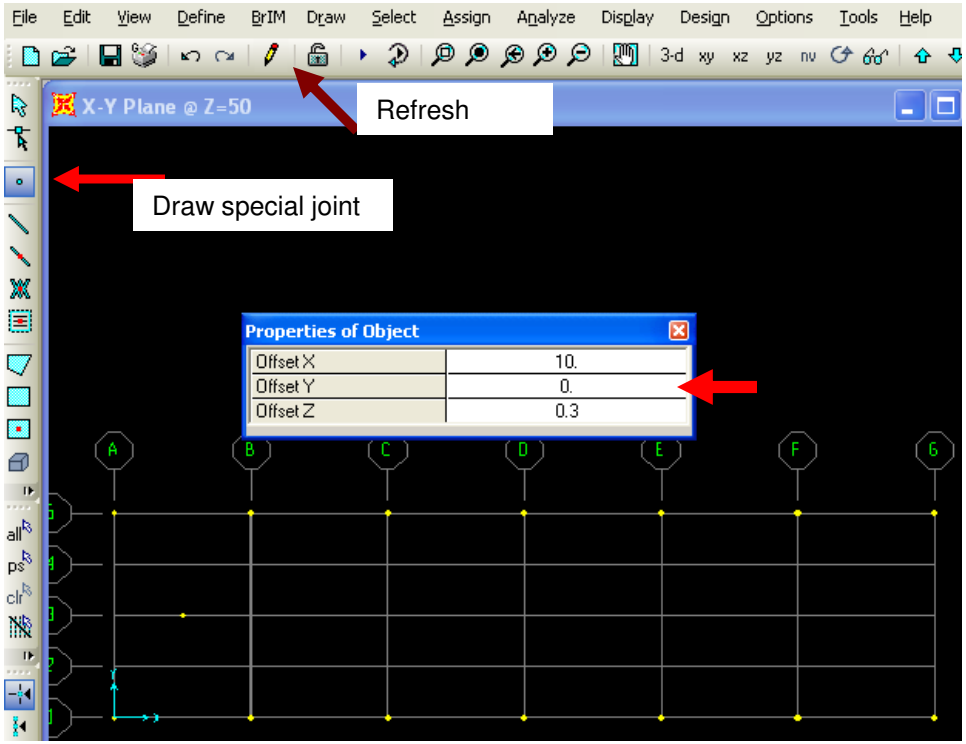




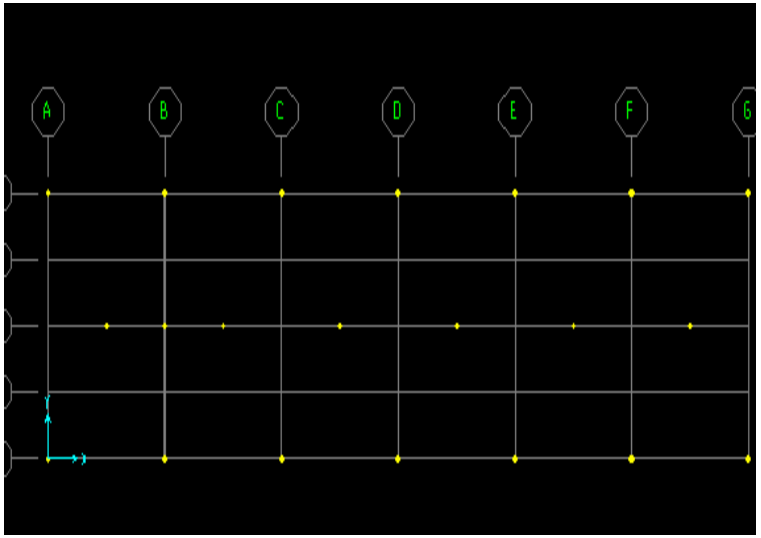
Next go to Select menu>Select>Properties>Area properties and click AIRCOOLER to select. Then Edit>Replicate the selected areas 20ft in the X direction 5 times/increments.



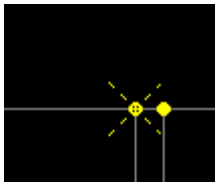
Next, switch to xy plan view Z=50 ft. Here click ‘draw special joint’ to add joints at the aircooler CG to assign weight. Input Offset X = 10’ and Offset Z = .3 ft as shown, then click the intersection of gridlines 3 and A, and click refresh to view the joint. By default, SAP2000 enables you to view any object .83 ft on either side of the 2D planar view. You can change the .83 ft. viewing tolerance default with the Options menu>Dimensions/Tolerances. Next, press Esc key to go into select mode, select the new joint, and replicate it 20 ft. in the X direction 5 times/increments

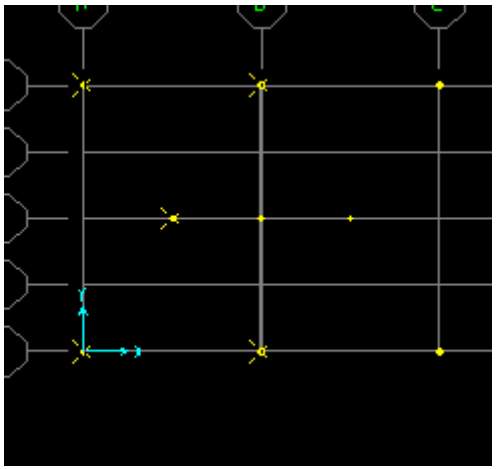


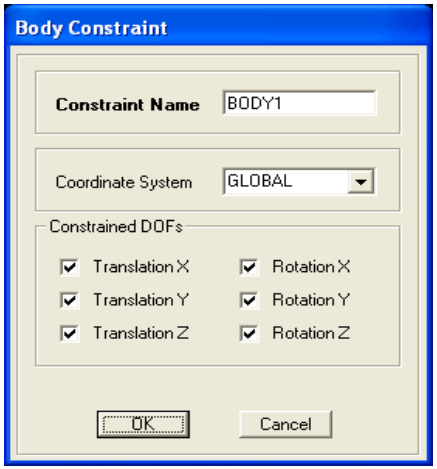
Replicate the joint 5 times

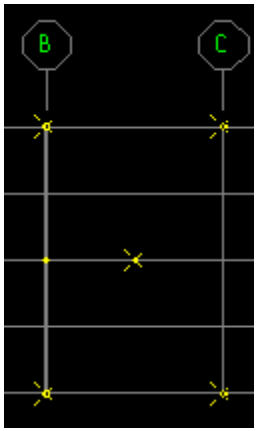


Still in xy plan view, zoom in and zoom out to select the 4 bottom corner joints of the aircooler elements plus the CG joint (5 joints per bay) where we will later assign the aircooler weight at the CG joint location. Then Assign>Joint>Constraints, body type to rigidly link the CG joint to the 4 corners one bay at a time. The body constraint rigidly links the joints with consideration of the moments from the offset distance. You need to assign a joint body constraint to the CG joint of each aircooler "box" in each bay. You will need to zoom in to each of the 6 bays one at a time, because the edge joints are so close together. You will need to assign a different joint body constraint for each bay which means that you will have a BODY1, BODY2, etc.



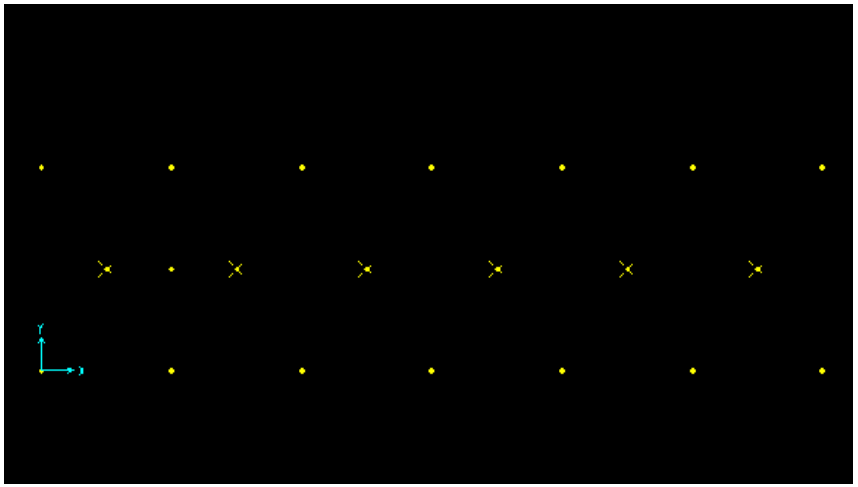


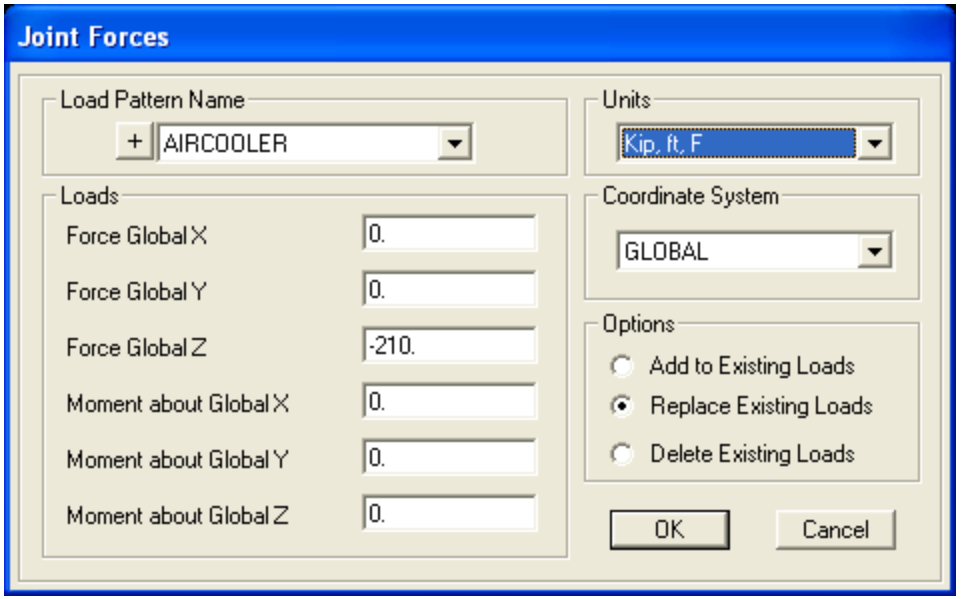




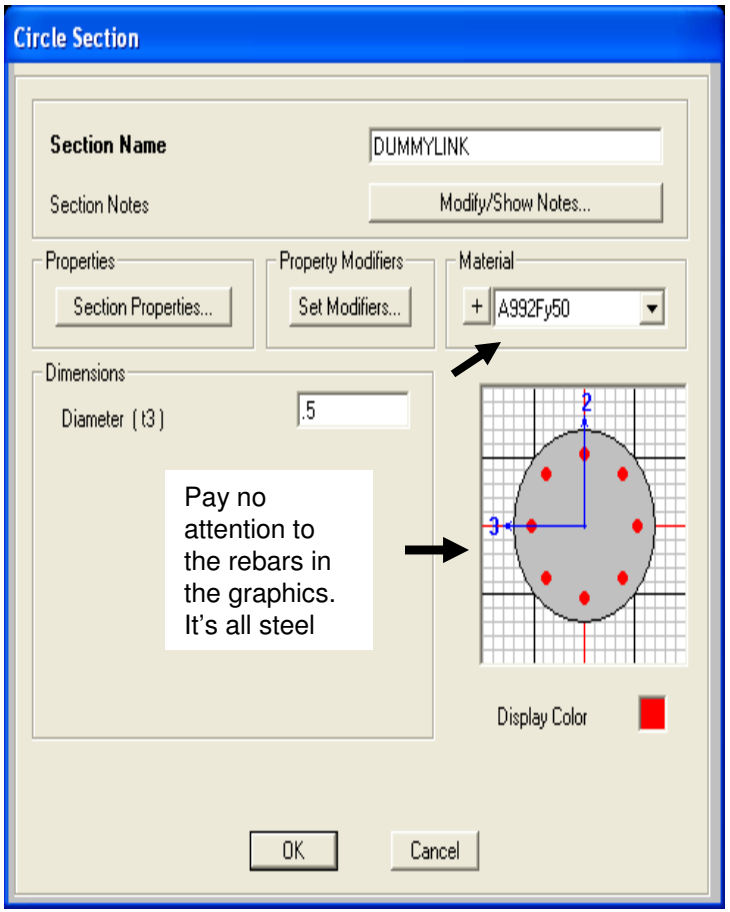
Add a different BODY constraint to each 5 joints (4 bottom corners of air cooler + 1 CG joint) for each bay

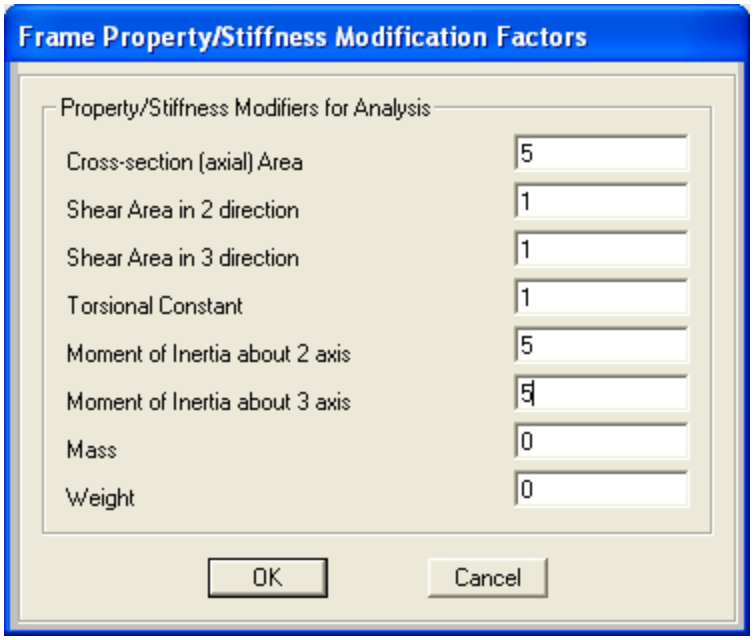
Left click to select each of the 6 aircooler CG load joints as shown on bottom left, then Assign>Joint loads>forces 210 kips in the –Z direction under Load pattern AIRCOOLER as shown on bottom right. According to the PR2.pdf drawing, each aircooler should be considered 350psf. Given that each bay is 20’ X 30’, that makes the weight of each air cooler 210 kips, which we lump at the CG of each air cooler



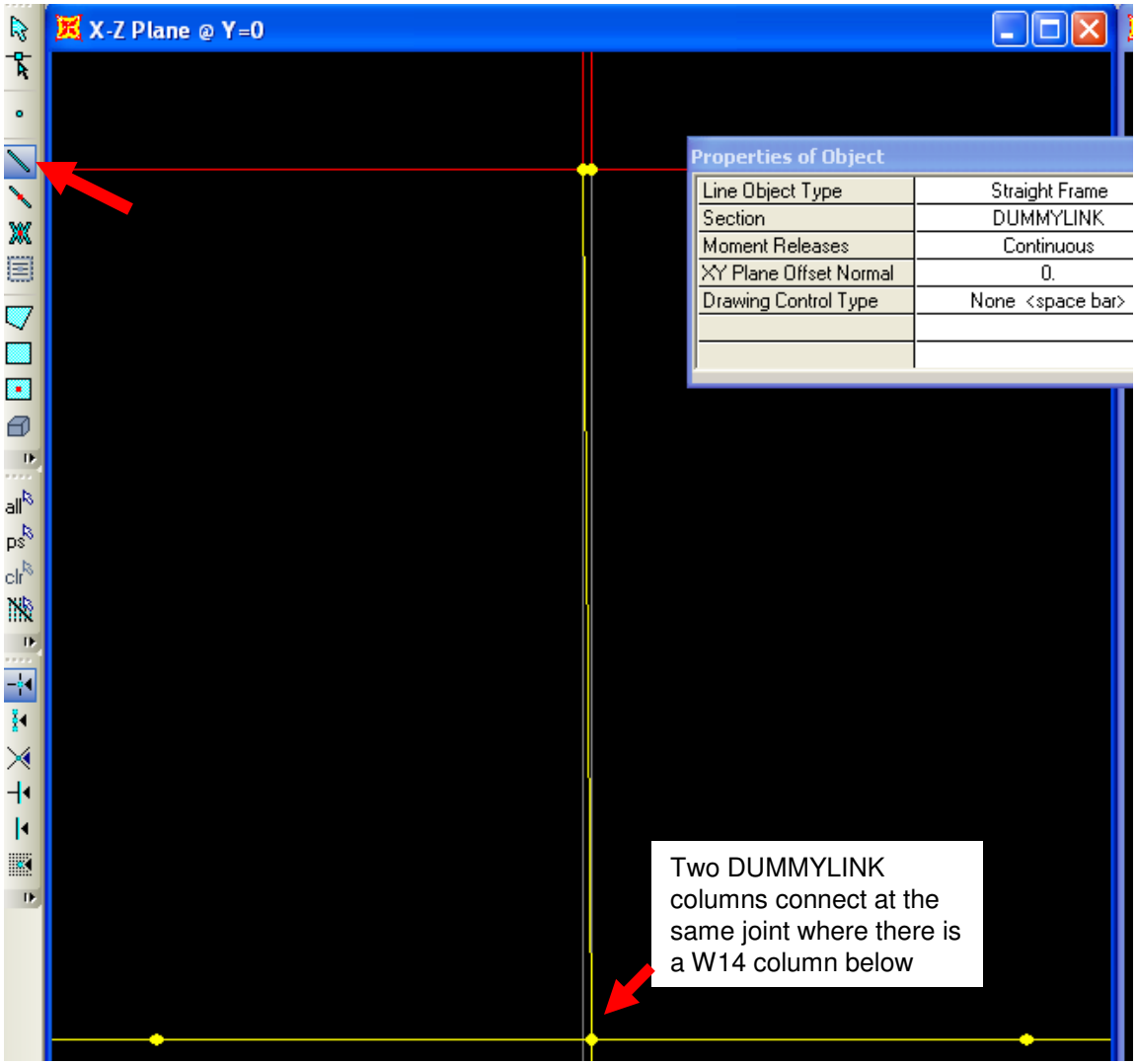


Next step is to link the aircoolers to the structure. You can link the aircooler to the structure using frame elements (W14 or whatever section) or link it rigidly with joint constraints. In this exercise, we will assume that the aircooler manufacturer has already designed the legs and will deliver each aircooler as a pre-designed module. Therefore, we will link each cooler to the structure using an alternative to the joint constraint method we used before using a “DUMMYLINK” zero weight frame/column element. Go to Define>Section properties>Frame sections and click ‘Add new property’. Change frame property section type to Concrete, then click circular. Change Material to A992 steel, .5 ft. diameter, then click ‘Set modifiers’ button. Enter 0 for Mass and Weight which we lump at the CG joint, and enter 5 multiplier for area and EI as shown below to define a stiff, but not completely rigid zero weight section. Click OK until the section is added to the working list

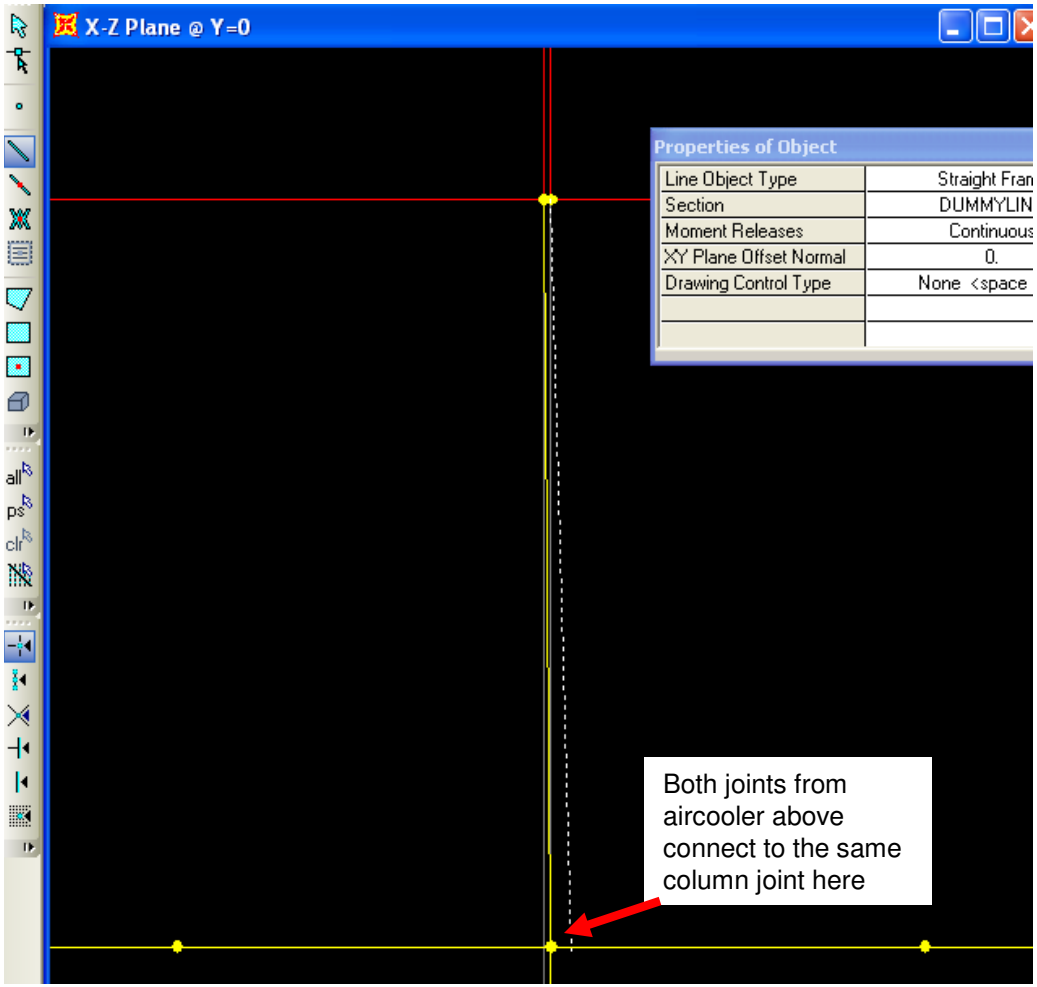


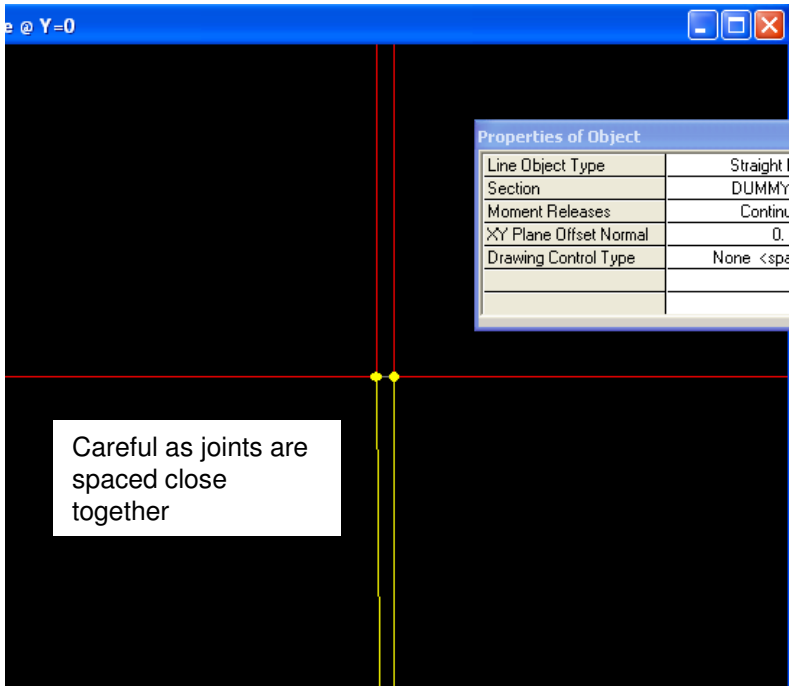


Go to xz planar view Y=0 and press F4 to clear the screen. Click ‘draw frame/cable element’ then change section to ‘DUMMYLINK’ frame section which we just defined. You will need to zoom in tight in order to draw the frames to the same joint shown below, connecting the AIRCOOLER joints to structure. Both (2) air cooler joints will need to be attached to the structure in this fashion. In this screenshot, only 1 of the two are attached.

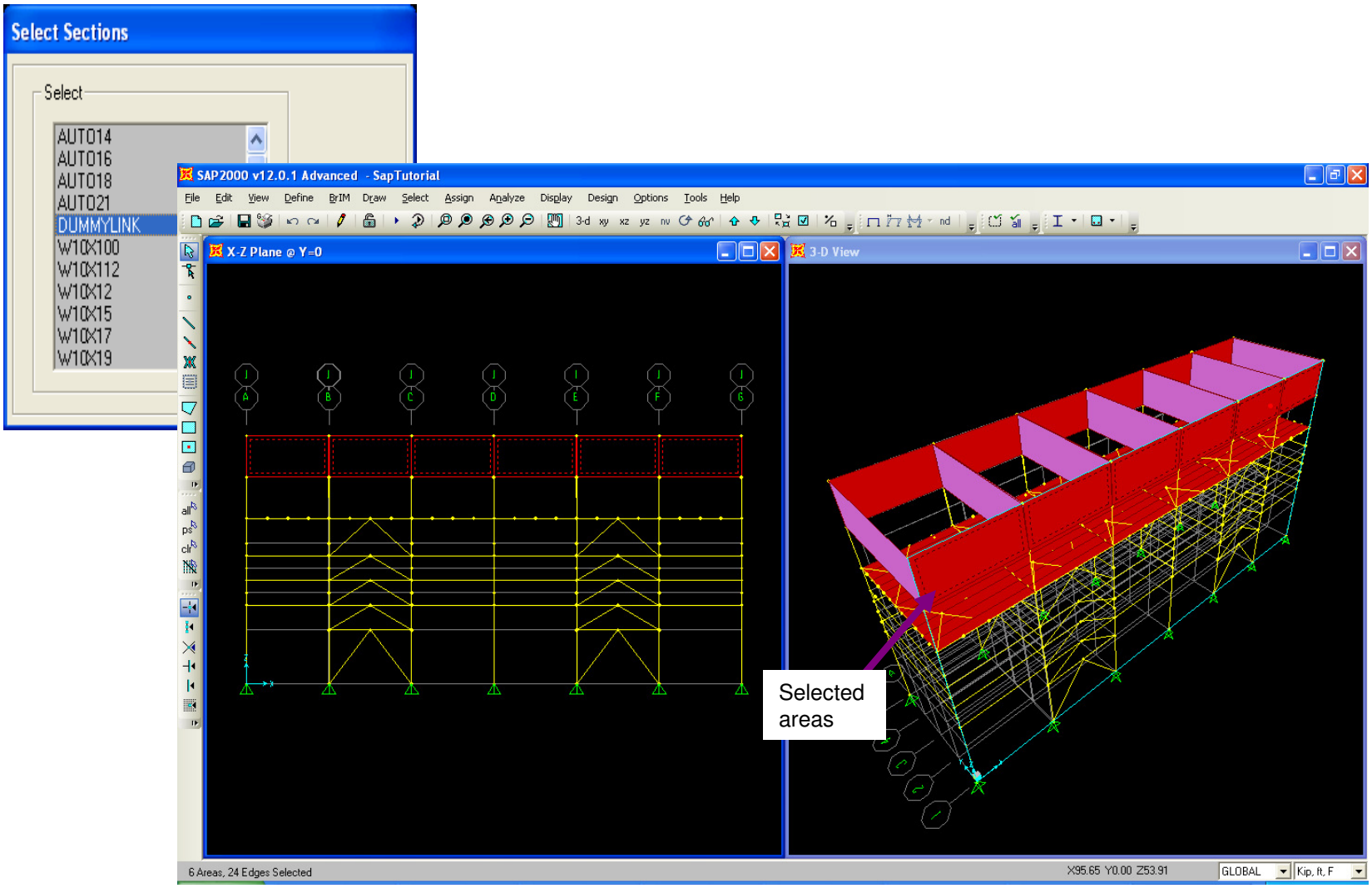


As you can see, you will need to connect both closely spaced DUMMYLINK columns to the same structural joint below where the W14 column is located. After finishing this, press Esc key or Select arrow to switch to select mode. Draw these aircooler DUMMYLINK columns for each bay in xz view Y=0.



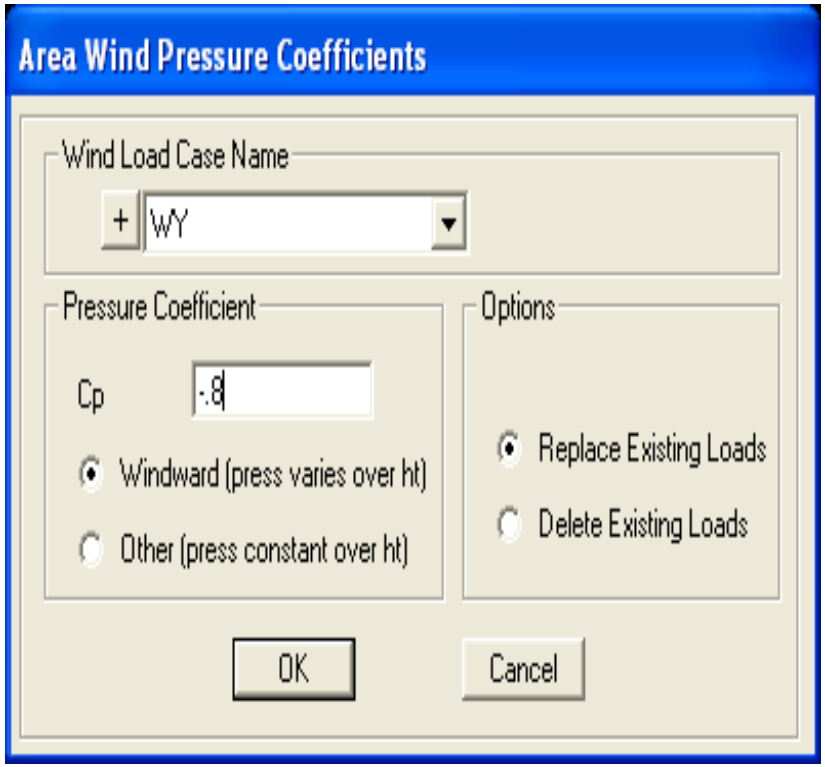


Use Select menu>Select to select by frame property “DUMMYLINK” to select these newly drawn columns. Next, use **Edit>Replicate** to copy 30 ft in the Y direction 1 time/increment in order to copy to the other side. Next, go to xz elevation view Y=0 and select the AIRCOOLER <u>areas</u> in that view along gridline 1 as shown below by clicking them.

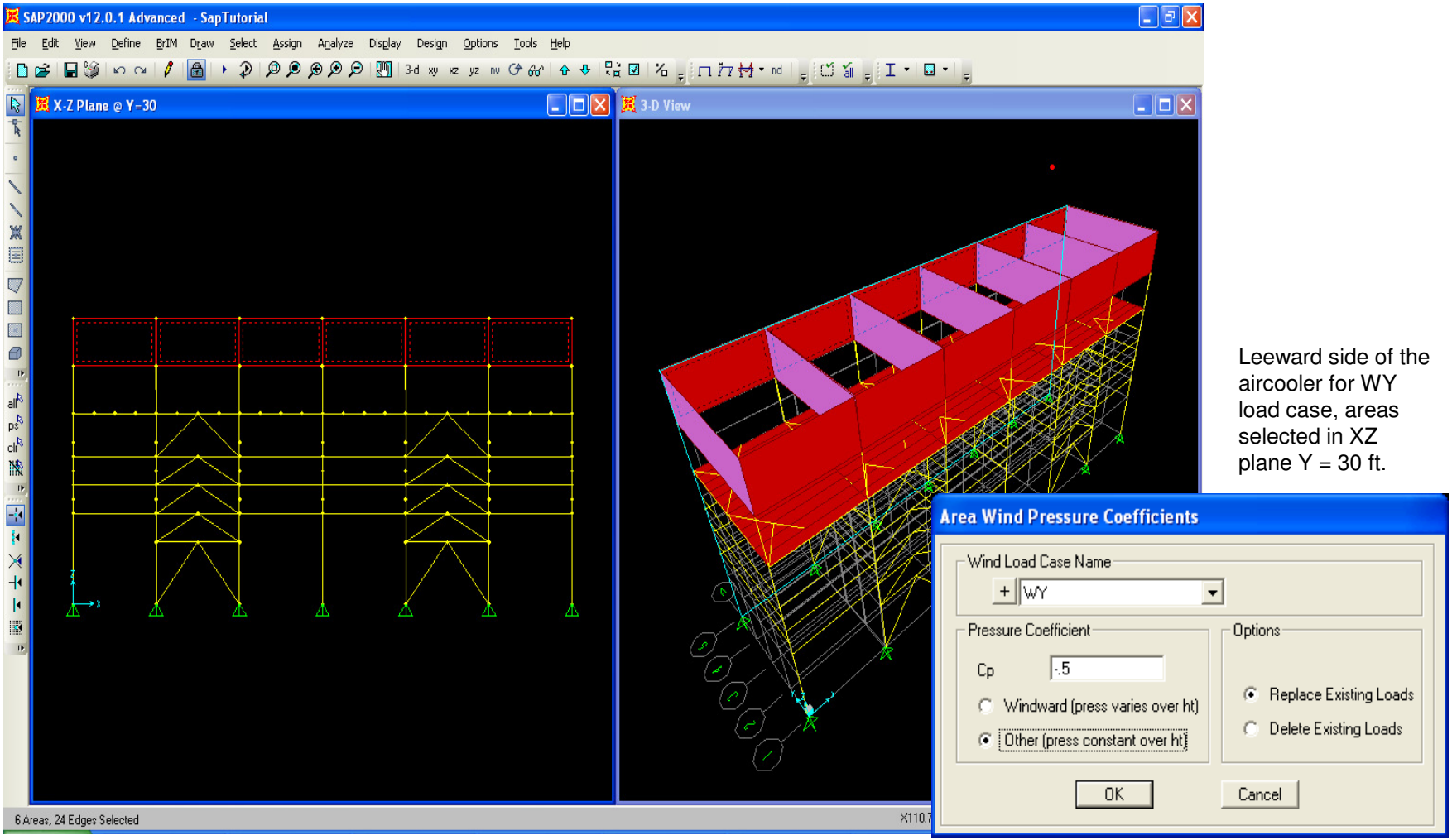


The red face of the AIRCOOLER area elements in this plane tells us that the local perpendicular axis of the selected elements is (+3) is = -Y according to the way we modeled these area shell elements. Use Assign>Area loads>Wind pressure coefficients (Shell). Since we are assigning a windward load in the +Y direction to areas which have a local axis of +3 = -Y, we will need to assign a windward Cp of -.8 (which is +Y direction) to area elements in XZ plane Y=0. Area wind pressure coefficients are relative to the local 3 direction of each area. Press F4 on the keyboard to clear this load assignment.

Next, use arrow keys to move to XZ plane Y=30' to select aircooler areas for leeward wind assignment

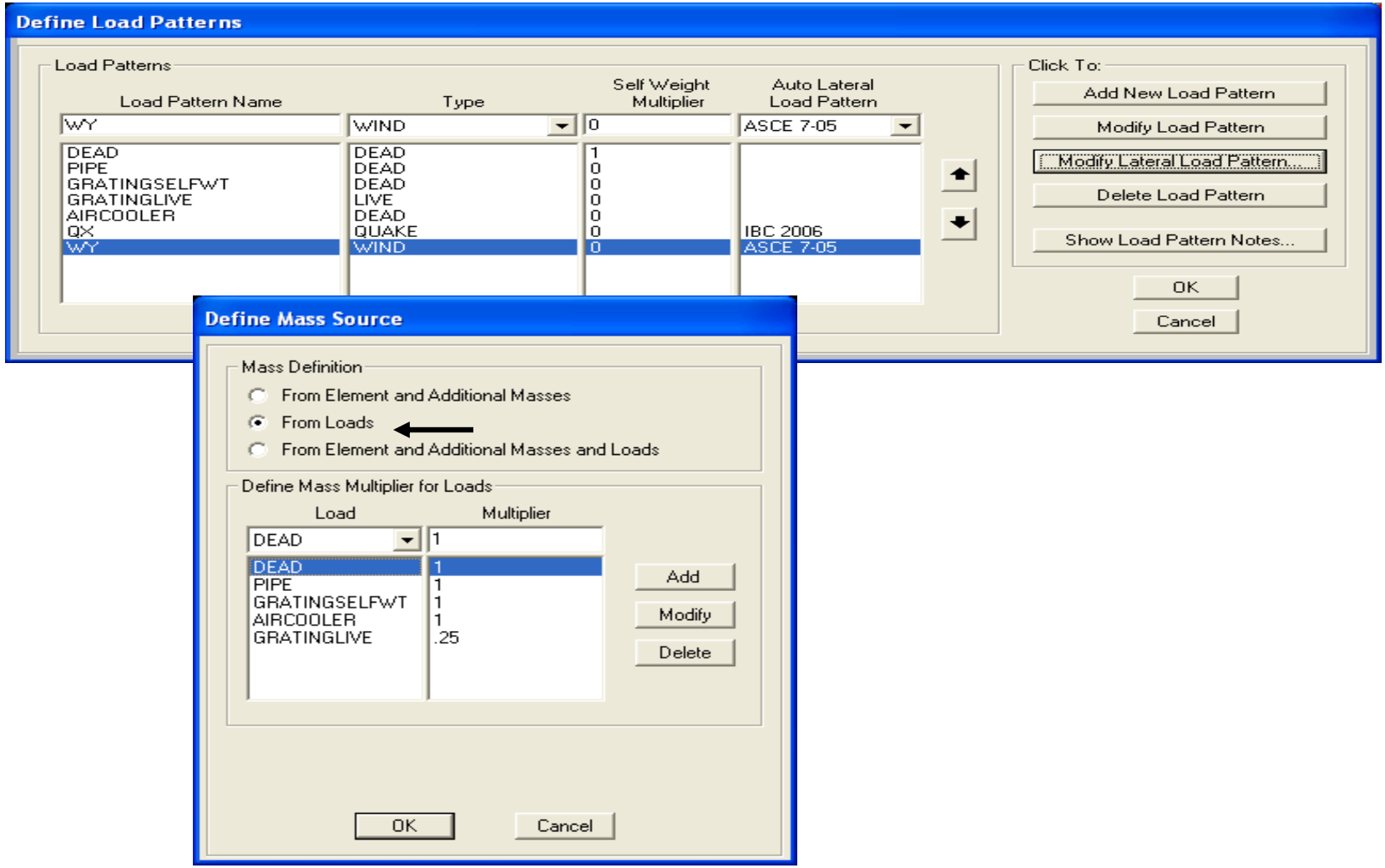


In X-Z, Y=30 ft. view, select the areas along gridline 5 in order to assign the leeward side of WY load case. Assign>area loads>wind pressure coefficients as shown below on bottom right using Cp of -.5 in the “other” (leeward) direction for the same wind load case. You can use ‘Set display options’ to check local axes of area elements. Blue arrows = +3 area/shell axis ASCE wind load generation on frames is automatic, but wind loads on area elements require an additional assignment. With this in mind, we could have also modeled pipes and/or vessels using frame elements (with or without zero weight sections) and had SAP2000 automatically generate wind load for them too.

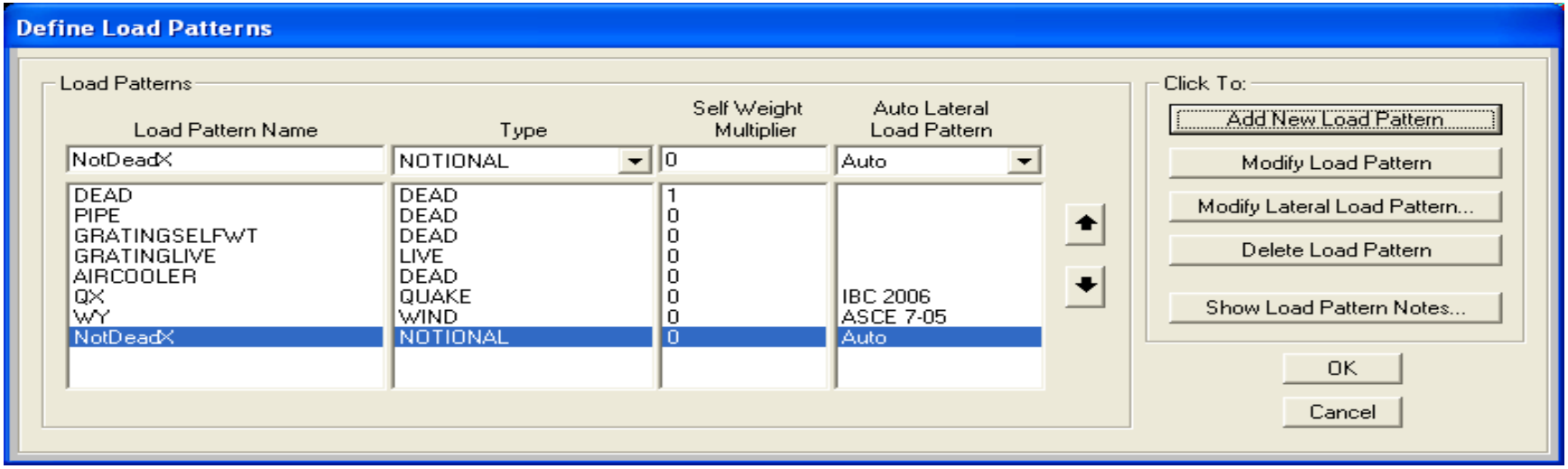


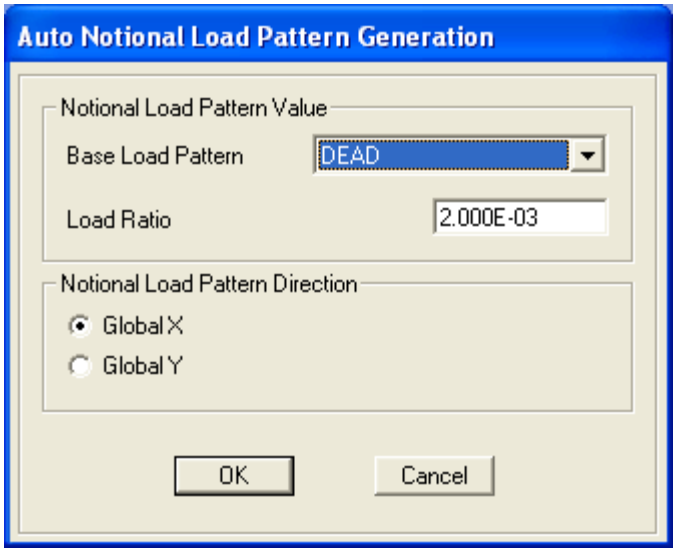
Leeward side of the aircooler for WY load case, areas selected in XZ plane Y = 30 ft.

For auto lateral load calculation of static equivalent IBC seismic loads, SAP2000 has to determine fundamental periods of the structure, which means that a dynamic modal analysis will be performed. Typically you will want to modify the mass model using **Define>Mass model**. SAP2000 offers an option to convert load cases into mass. Please specify your mass model as shown below and press OK

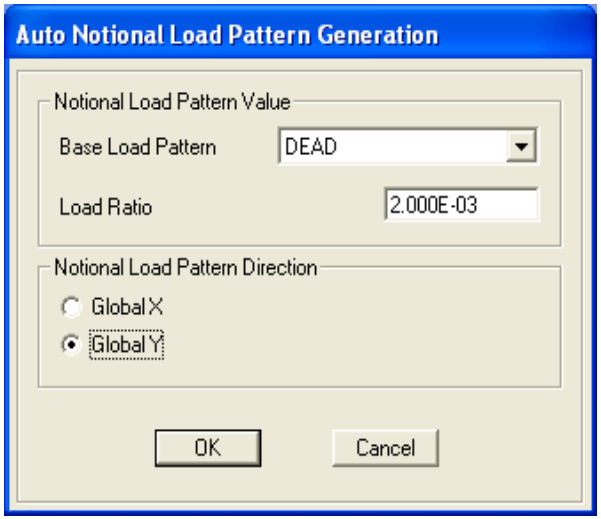


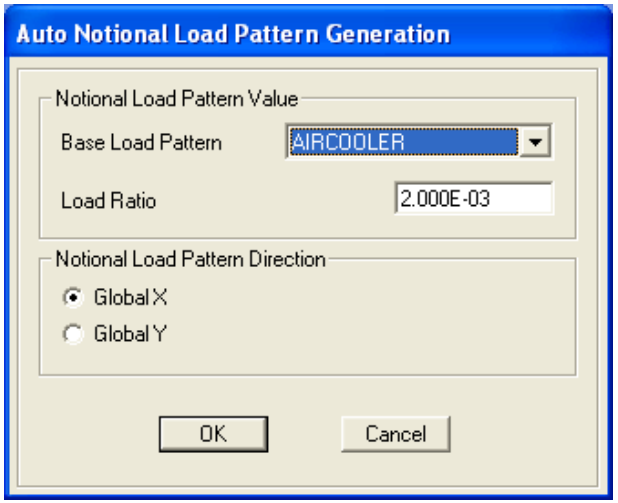
Since this design will be performed using 2005 AISC code, we will need to generate “notional” lateral loads, which is a new AISC code requirement to account for out-of-plumbness. SAP2000 offers the capability to automatically generate lateral notional loads, but users must specify the gravity loads to be used. Add a load pattern named NotDeadX, type Notional (you may have to click ‘Other’ before notional type appears) and press ‘Add new load pattern button’ then press ‘modify lateral load pattern to specify parameters. For this example, let’s generate notional loads based on DEAD and AIRCOOLER load patterns since these are the two most significant gravity loads in this model

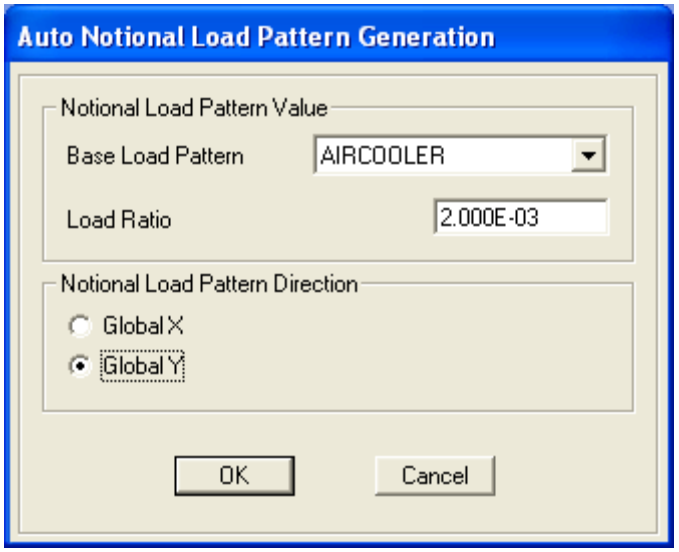


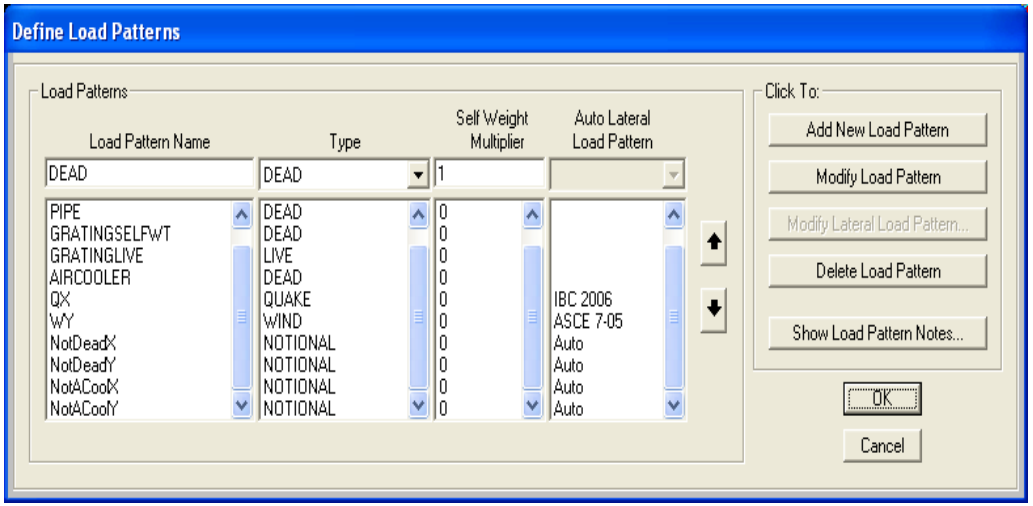


Below are the auto notional load patterns named NotDeadY (notional Y direction DEAD case), NotACoolX and NotACoolY for the air cooler notional loads. SAP2000 will automatically generate the notional lateral loads at each level

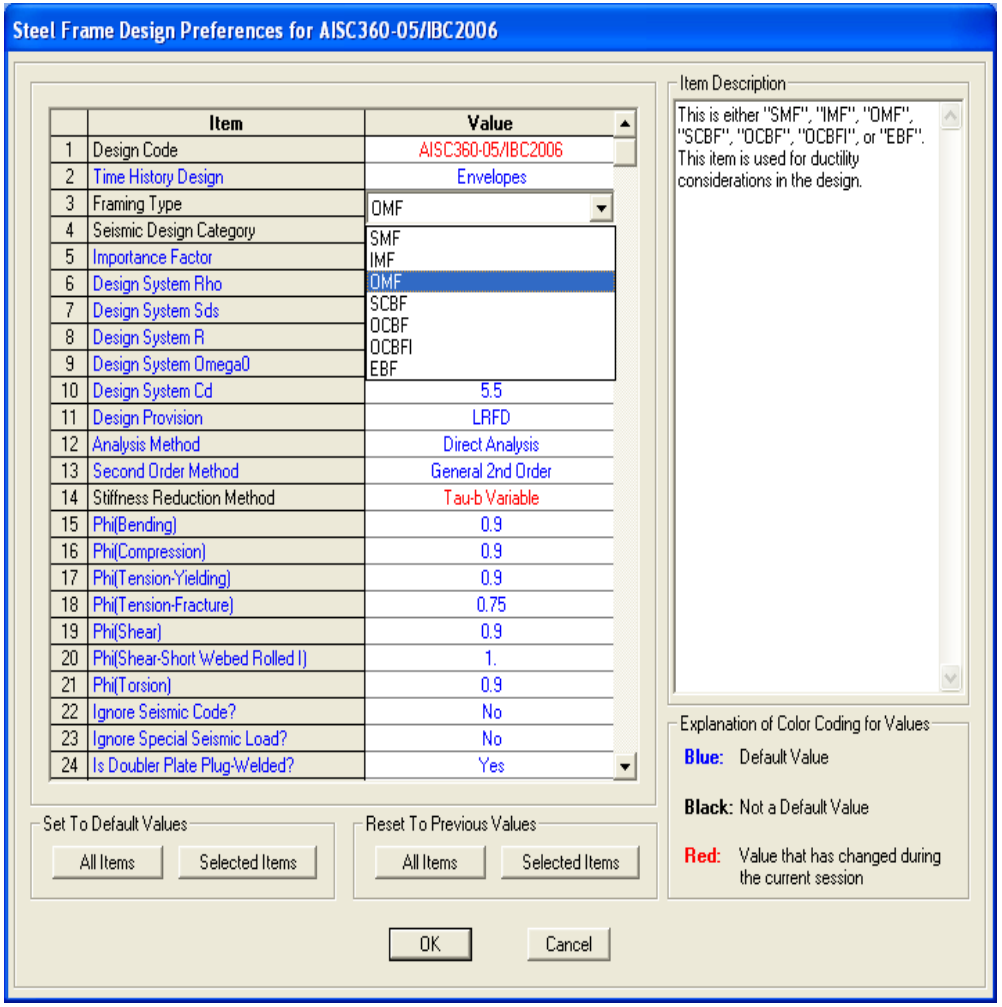




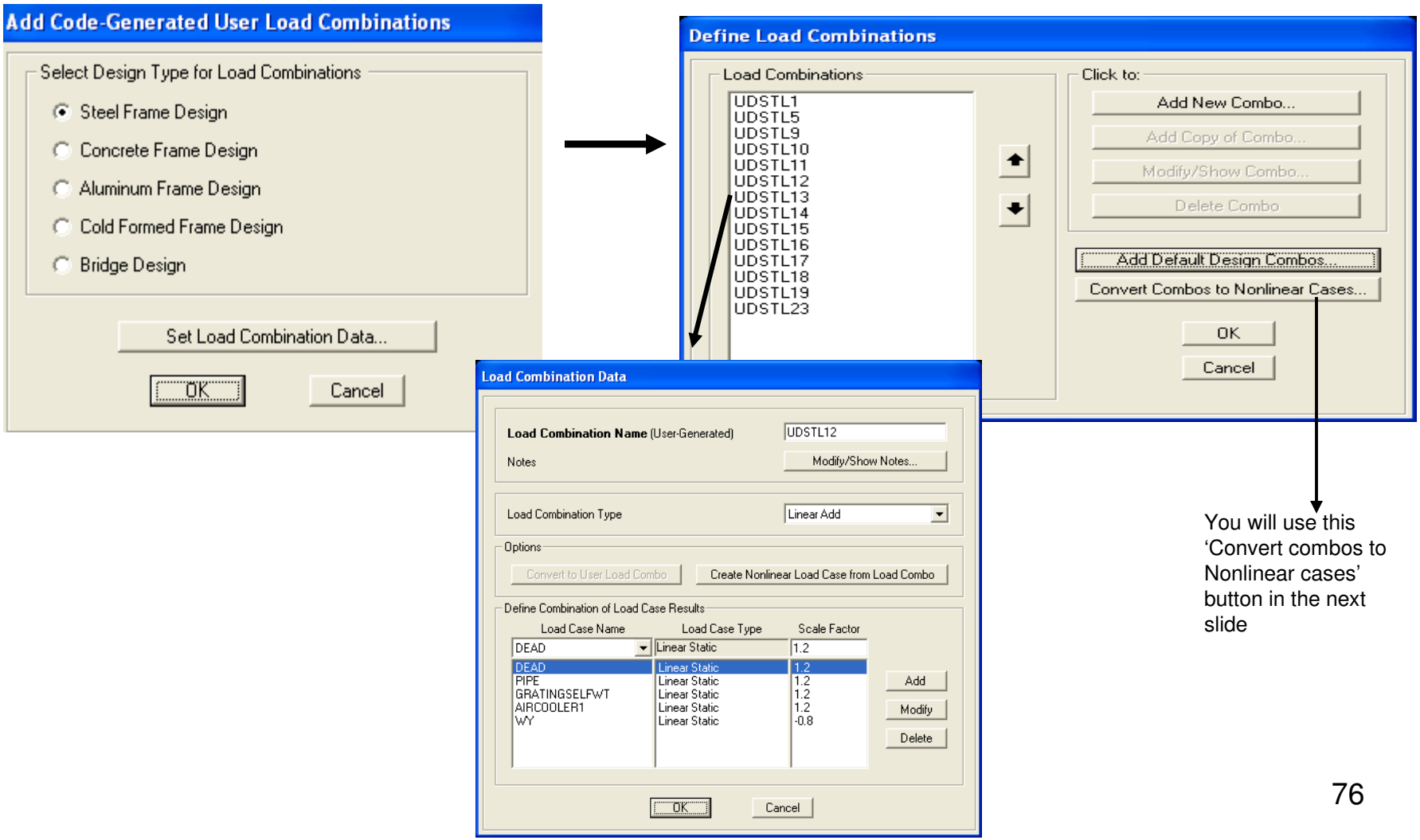




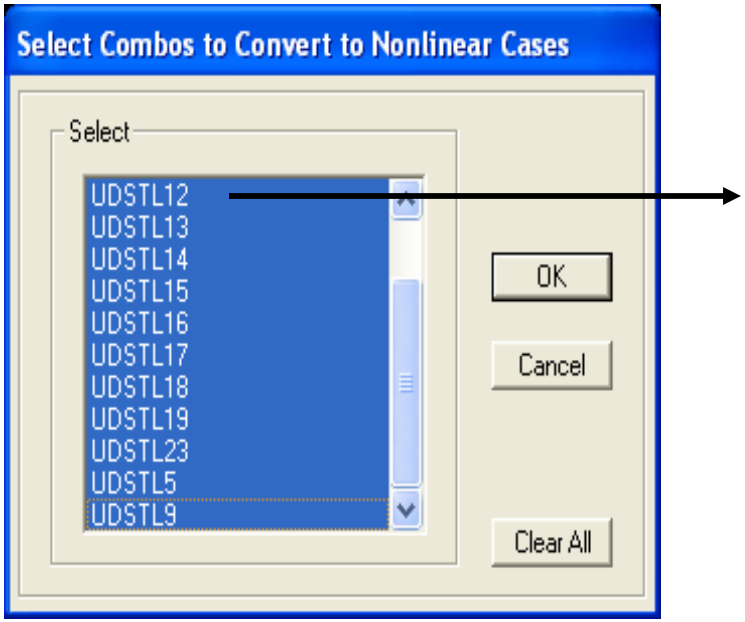
Next we will specify design parameters. For this example we will use the 2005 AISC Direct analysis method which enables us to avoid the time consuming task of determining K factors from alignment charts while often times achieving more economical designs compared to ELM. Go to Design>Steel frame design>View/Revise overwrites and input design code and other parameters as shown, OMF Framing type, Tau-B variable Direct analysis. You can select individual frame elements and assign design parameters using Design>Steel frame design>View/Revise overwrites. There is an excellent steel frame design manual which installs with each version of SAP2000 V12 under the 'Manuals' subdirectory.

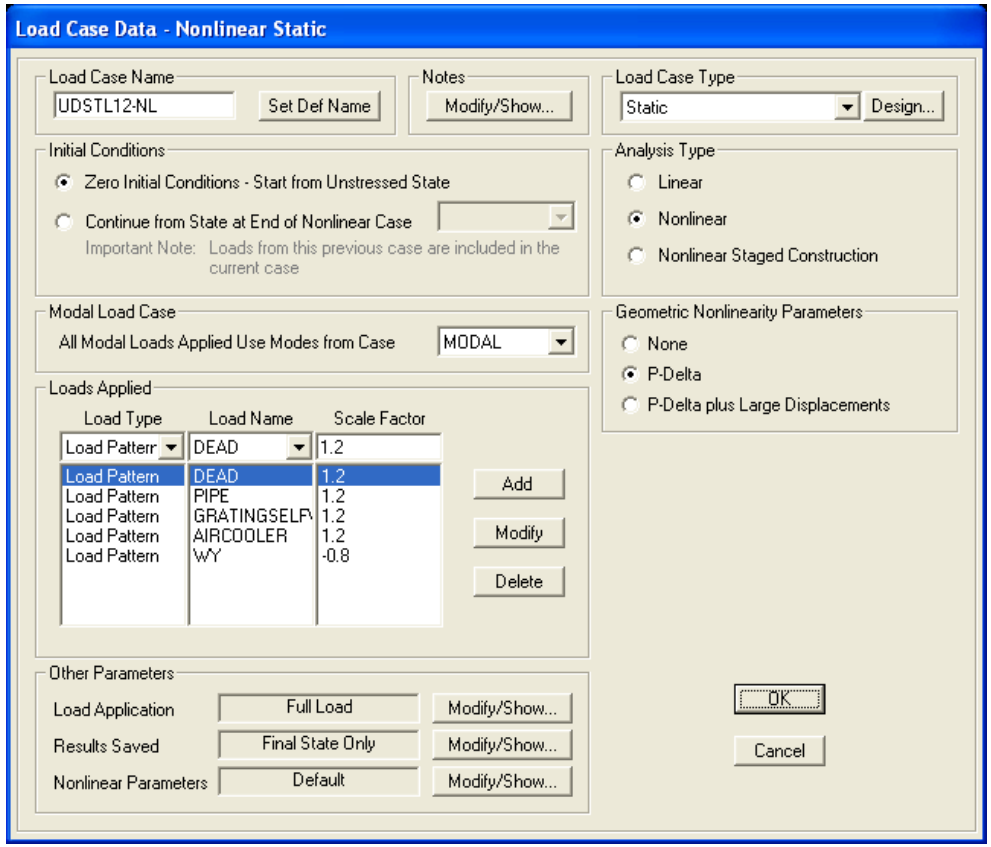


Next we will go to **Define>Load combinations**. Users can define their own load combos, or they can let SAP2000 create factored load combos for them based on the design code and load type. Click the 'Default design combos' option then select Steel frame design and press OK. This automatically generate factored load combos which can be reviewed or modified. Alternatively, users can create their own load combo spreadsheets in Excel and import.

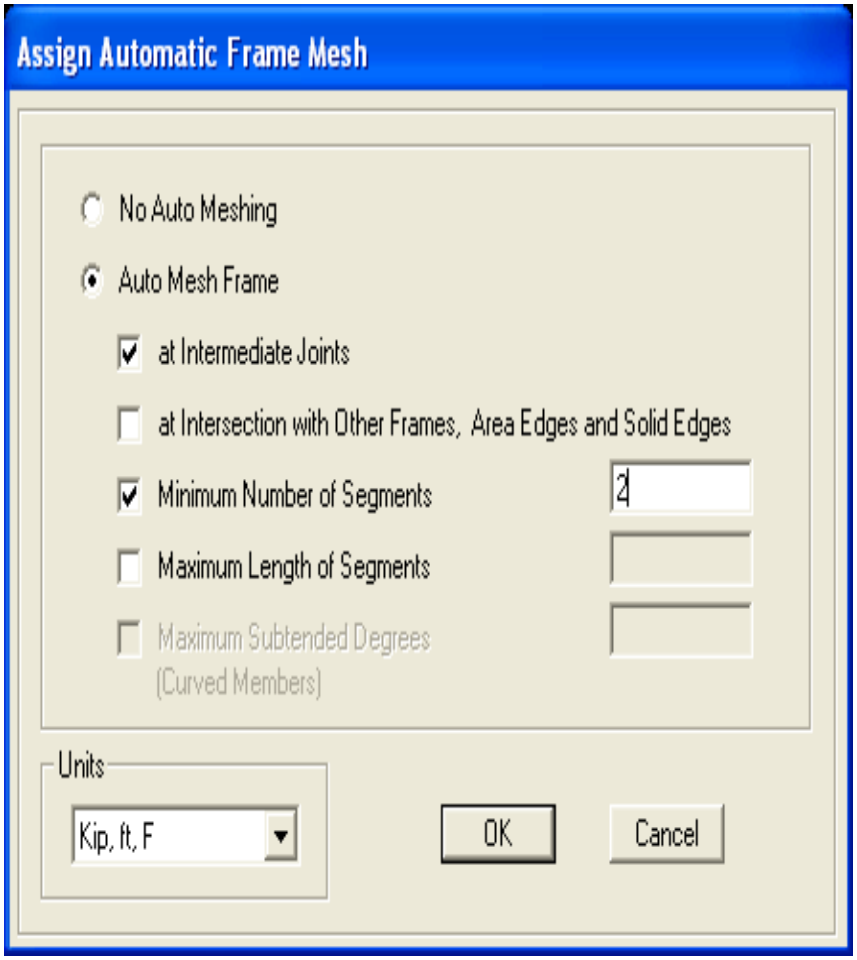


The 2005 AISC code requires either a nonlinear P-delta analysis, or a time consuming 1st order amplification procedure which has a number of limitations in its permitted use. For this exercise we will perform a P-delta analysis. Since P-delta is a nonlinear analysis, loads cannot be superimposed. This same concern is also true whenever modeling nonlinear tension only bracing and other NL applications. SAP2000 enables you to convert factored load combos to NL cases considering P-delta by clicking the 'convert to nonlinear cases' button and selecting which combos you wish to convert to NL. In most instances, you will want to convert all load combos to NL so select/highlight them all with your mouse as shown below and press OK. You will then see a message similar to the one below indicating how many combos were converted. You can then go to Define>Load cases to review the NL load cases which SAP2000 has automatically generated like the example below right:

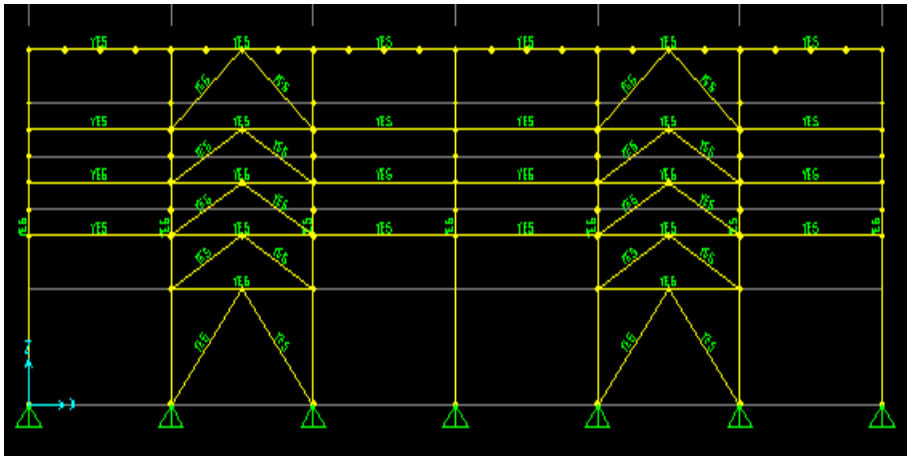




SAP2000's frame element formulation automatically accounts for both big P-delta sway as well as little p-delta local member deformation as required by AISC benchmark problems. In many cases no additional intermediate joints are required for good results. But in some cases, one additional intermediate joint may be required to accurately capture little p-delta reactions. Unlike most other structural software products which do not incorporate little p-delta analysis in the element formulation, with SAP2000, you never need to add more than one intermediate joint to accurately capture p-delta effects, no matter how complicate the loading. Automatically generate 1 intermediate joint for internal analysis only by selecting all, then Assign>Frame>Automatic frame mesh using the parameters shown below with minimum number of segments 2 and press OK. Press F4 to remove the automatic mesh assignments from the graphics

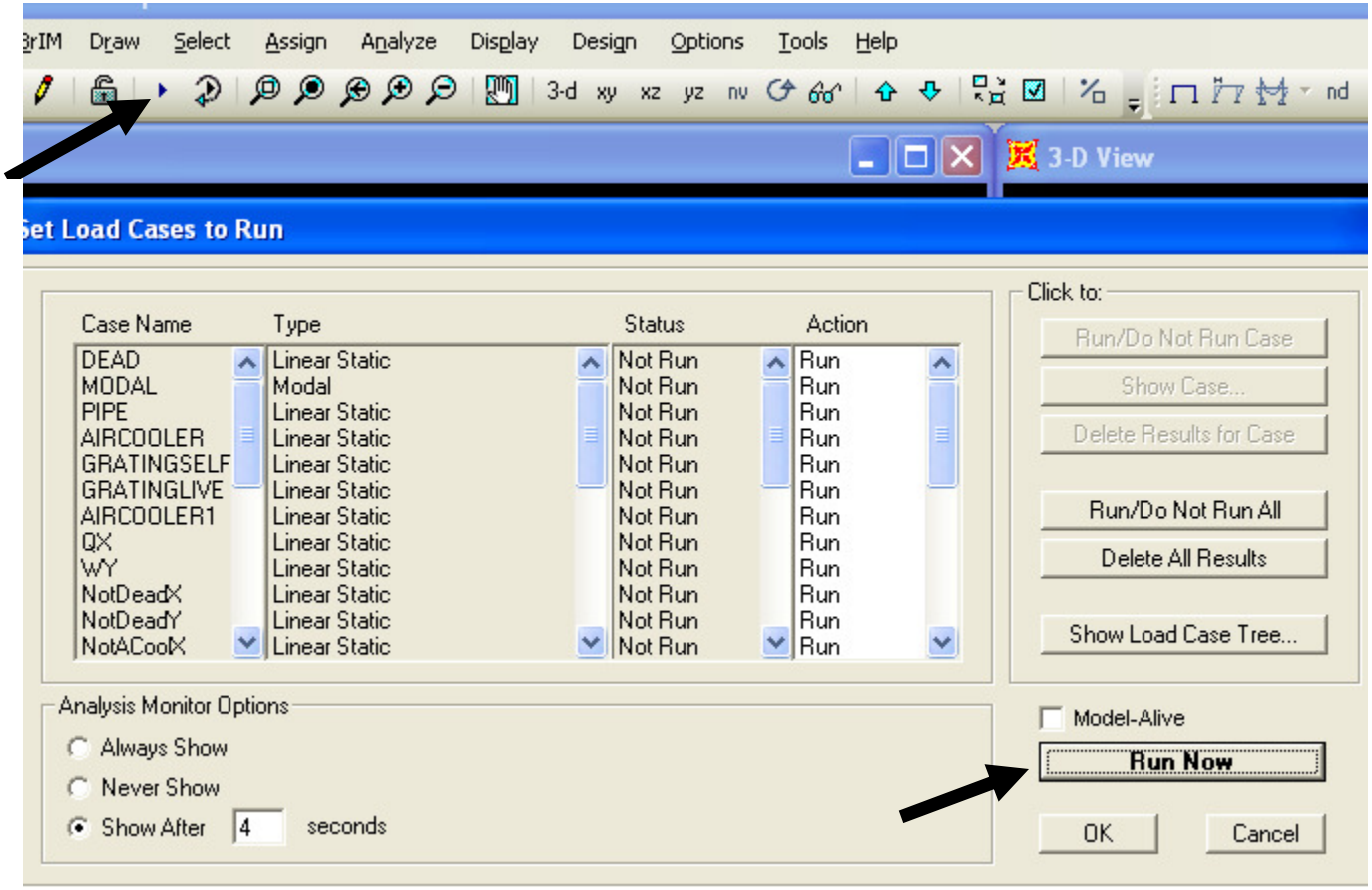


Press F4 to remove these assignments from the graphics

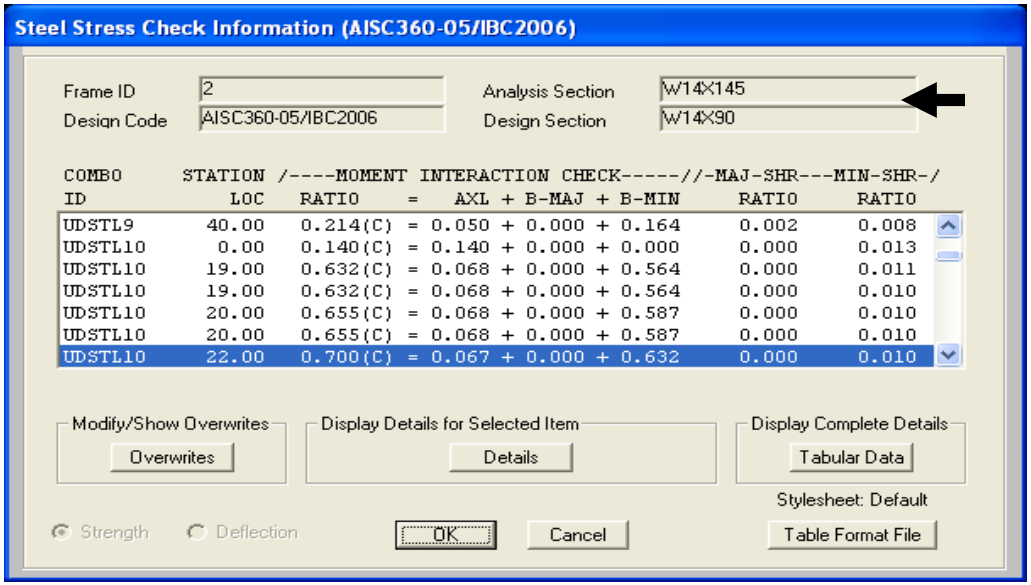


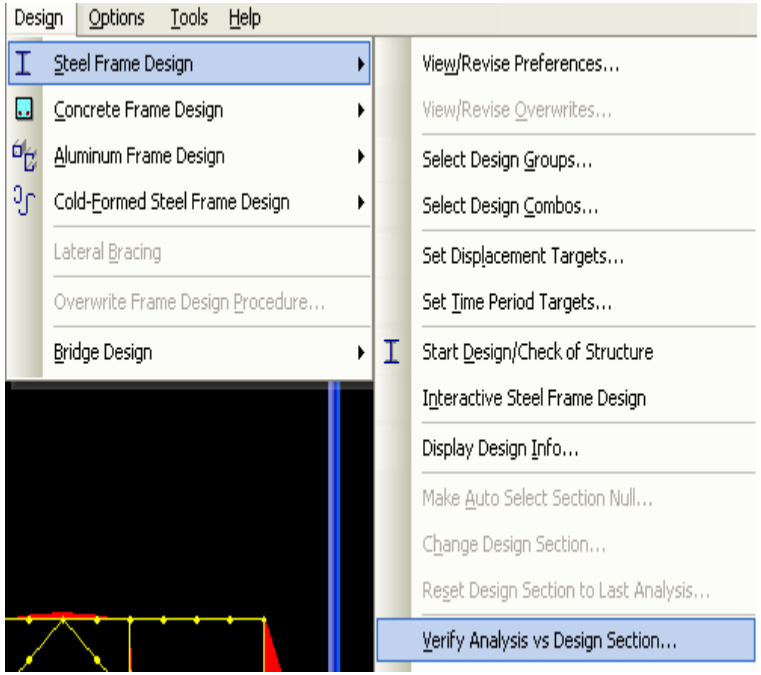
Click ‘Run analysis’ button, then press ‘Run Now’. After the analysis is completed, next click the ‘Start steel design’ button to perform the AISC steel design check. With SAP2000, analysis is run separate from design.

Run analysis

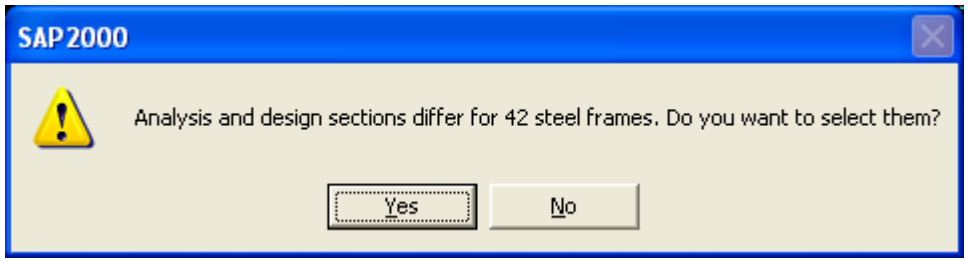


You can right click individual frames to review design results, but because we used autoselect lists, you can see below that a different section was selected for design than was used for the analysis. SAP2000 uses the median section in each autoselect list as a starting point to determine loads, and we need to iterate back and forth between analysis and design, typically 4 – 6 times if lateral loads are involved, in order to match the analysis sections with the design sections. You can check analysis vs design sections using **Design>Steel frame design>*Verify Analysis vs Design section*** shown below

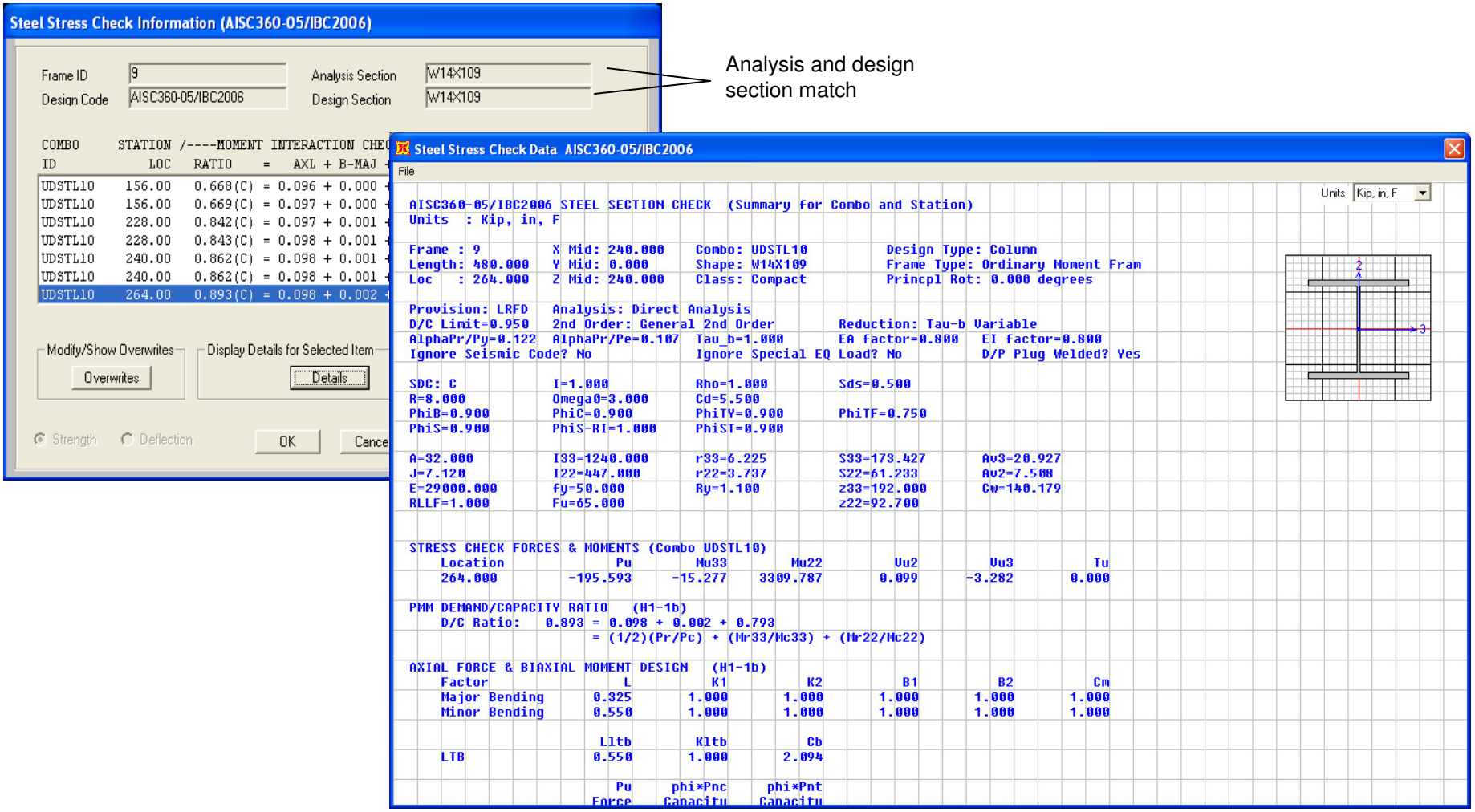




When you see this message, typically just click No, then analyze and design a few more times until analysis and design sections all match

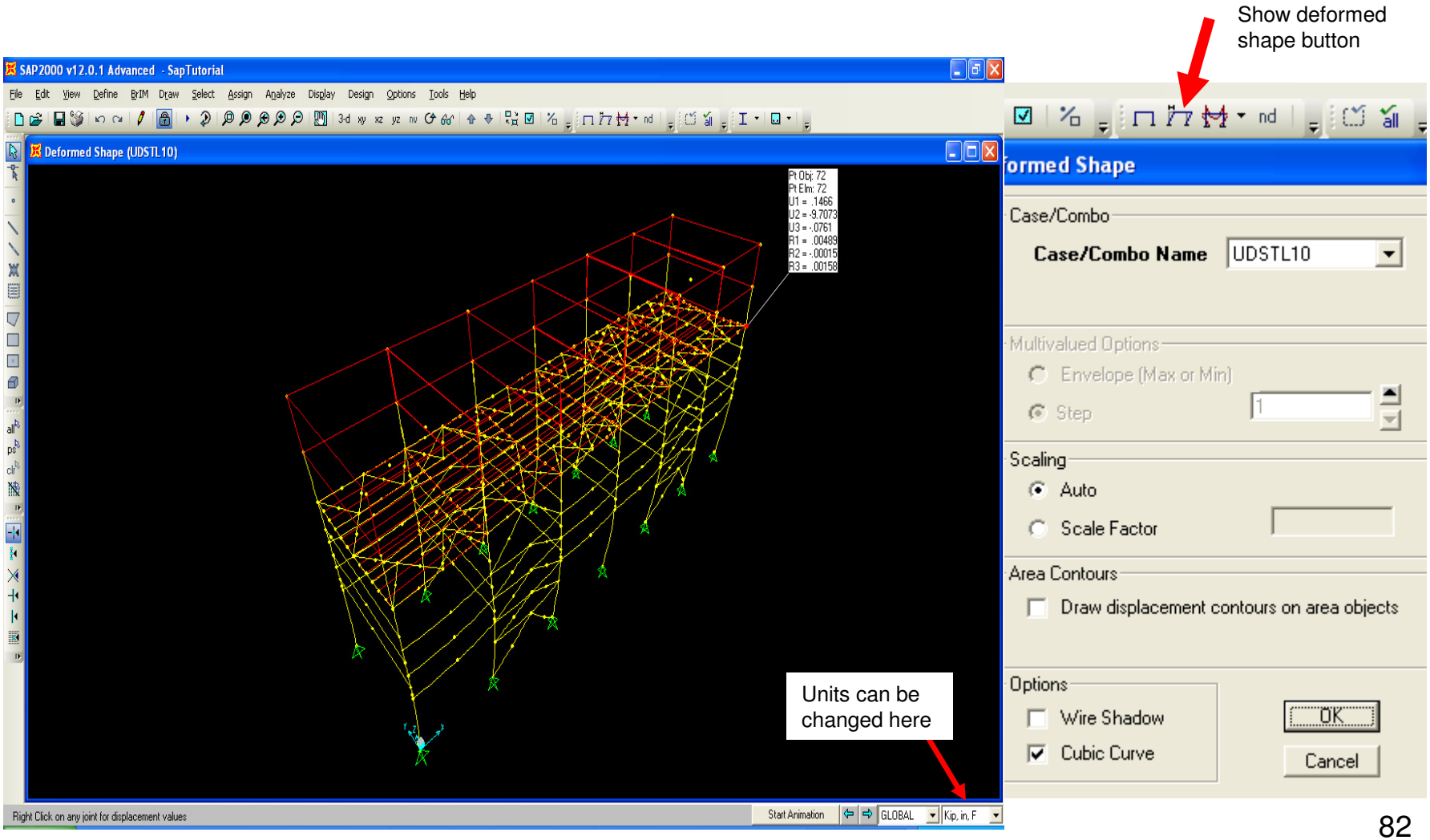


After analysis and design sections all match, you will see color coded graphical results based on percentage of design code allowable. Right click any frame to check results. SAP2000 runs all design combos along every 24” of every frame and defaults to display the worst case load combo and worst case location along the frame. Click Details for more info. You can see that SAP2000 automatically applied reduction factors to EA and EI per the AISC code for the Direct Analysis Method.

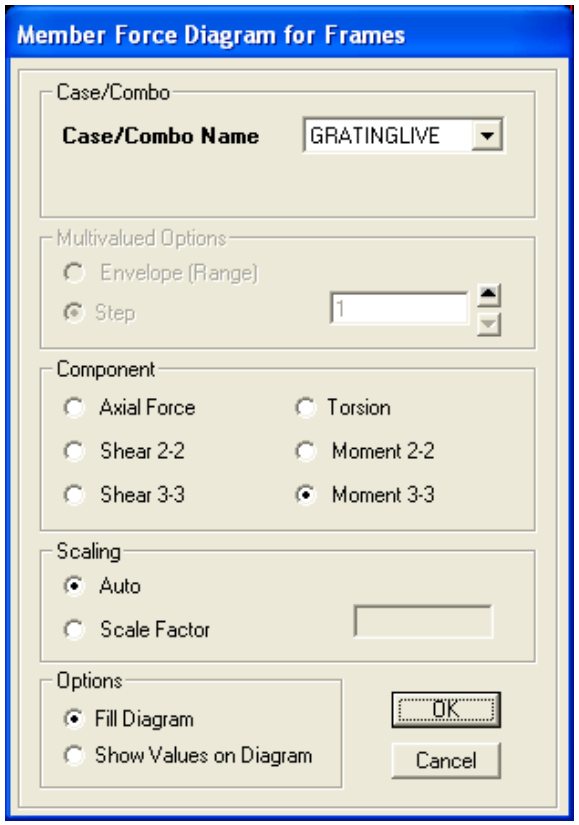


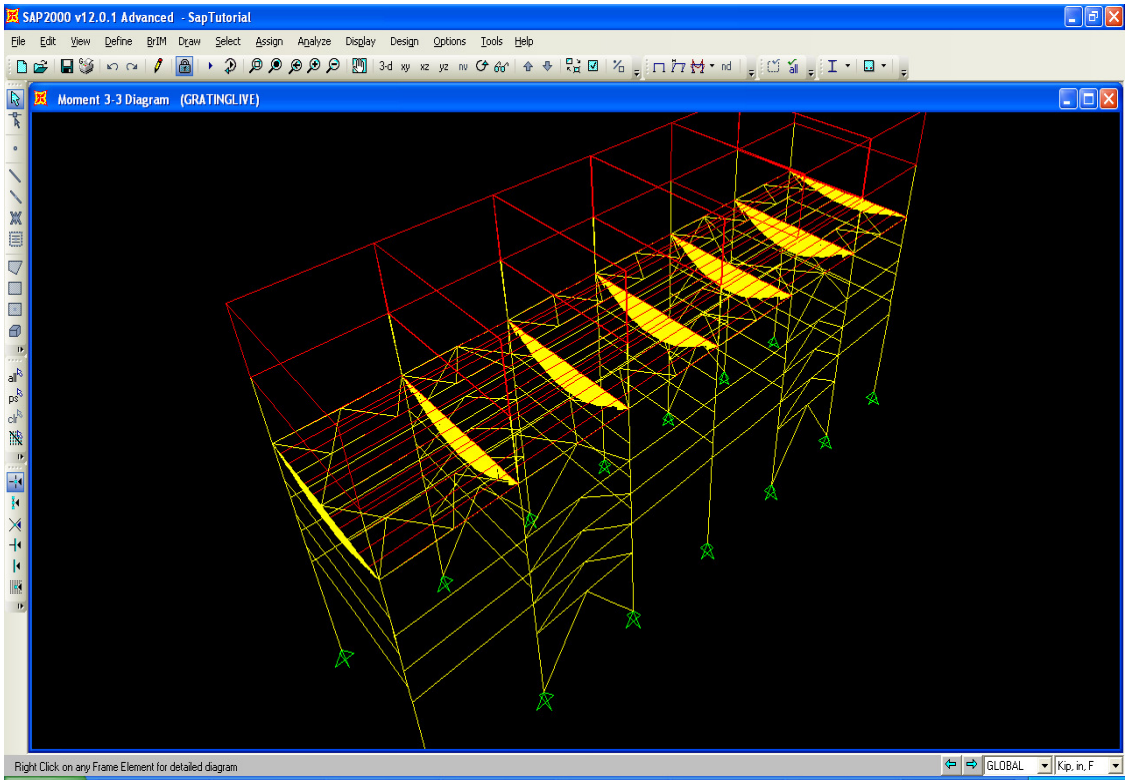
Analysis and design section match

In the design code check, load combo UDSTL10 was the worst case load combo for the frame we just checked, so we can spot check deflections by clicking the ‘show deformed shape’ button and select UDSTL10 combo. Change units to Kip-in and right click individual joints to check deflections or click ‘Start animation’ in the bottom right to watch the structure animate under that applied load combination. Press F4 to exit deflected shape



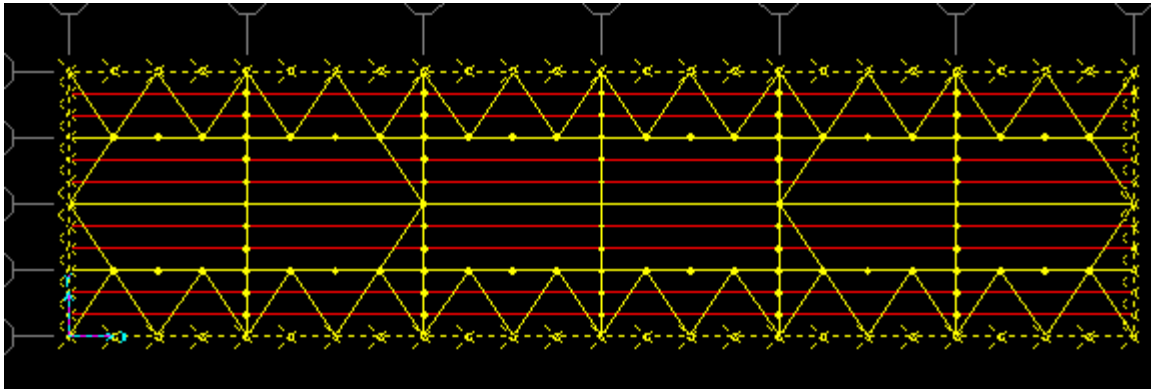
Next, let's do a spot check on load transfer from the grating to the beams. Click the 'Show forces/stresses' for frames button and choose GRATINGLIVE case Moment 3-3 (major). Moments show one-way load distribution to girders. Right click beams and bracing for detailed spot-checks to see that the grating did not distribute load to the bracing underneath, just like we wanted. Press F4 to exit moment diagrams



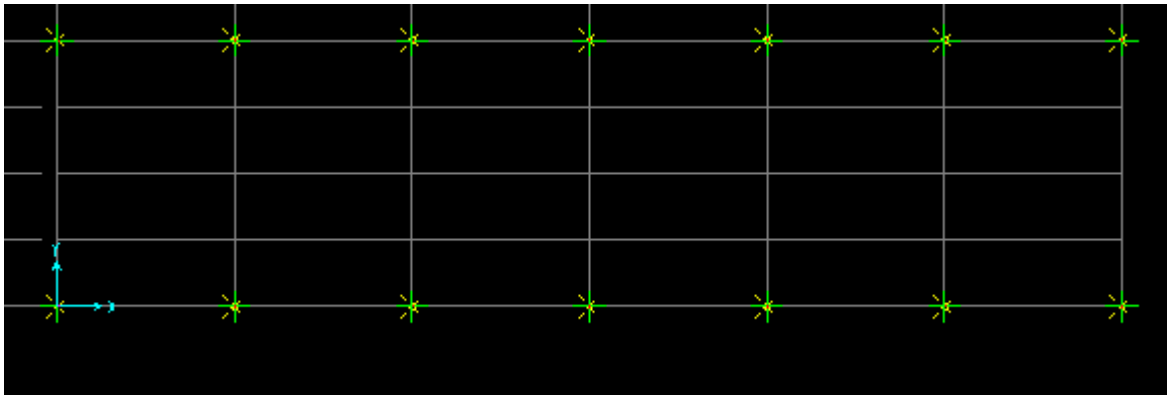


To minimize the size of the output reports, we can select just the joints and frames of interest to be included. Switch to xy plan view and use up/down arrow keys to go to the top level and select edge joints as shown by windowing around them in order to minimize output for max deflection checks. Next, use arrow keys to go to the base and window select the restraints.

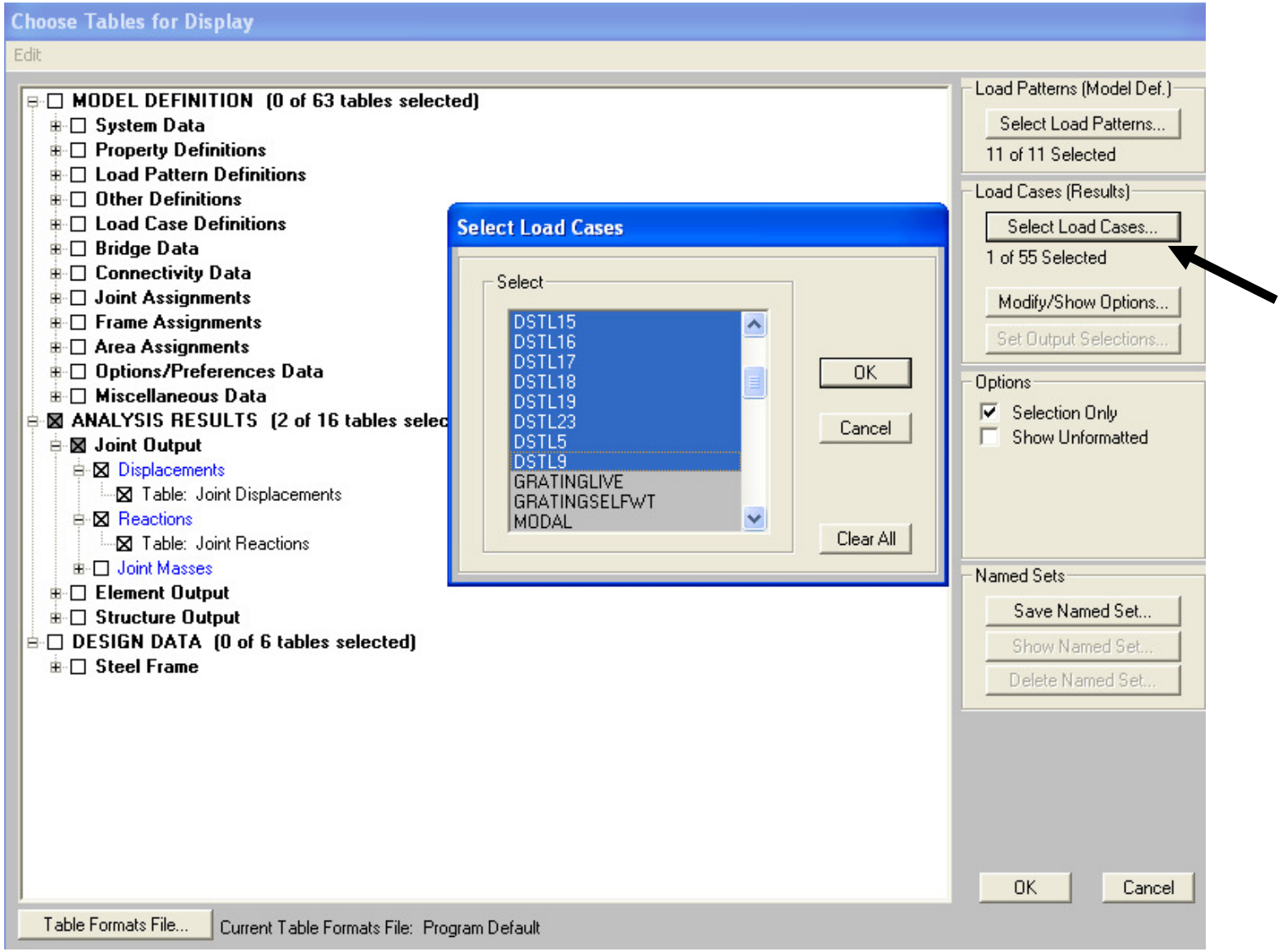
Top level edges selected in plan view



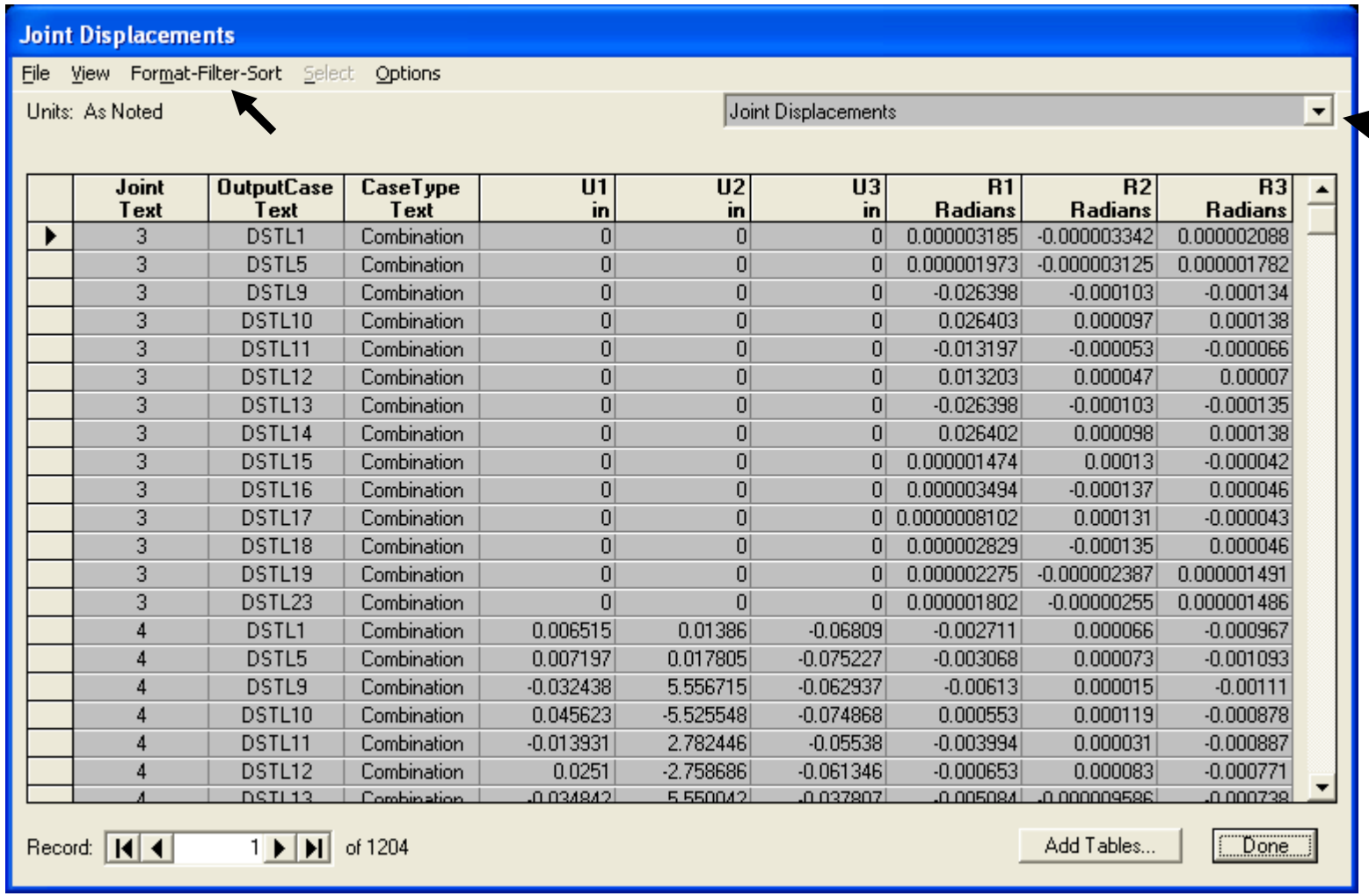
Base level selected restraints



Use Display menu>Show tables to generate output tables. The output in this example will be limited to the selected joints, and further limited by selected load cases and reports chosen. Typically avoid choosing modal for analysis results except for reporting modal dynamic analysis results

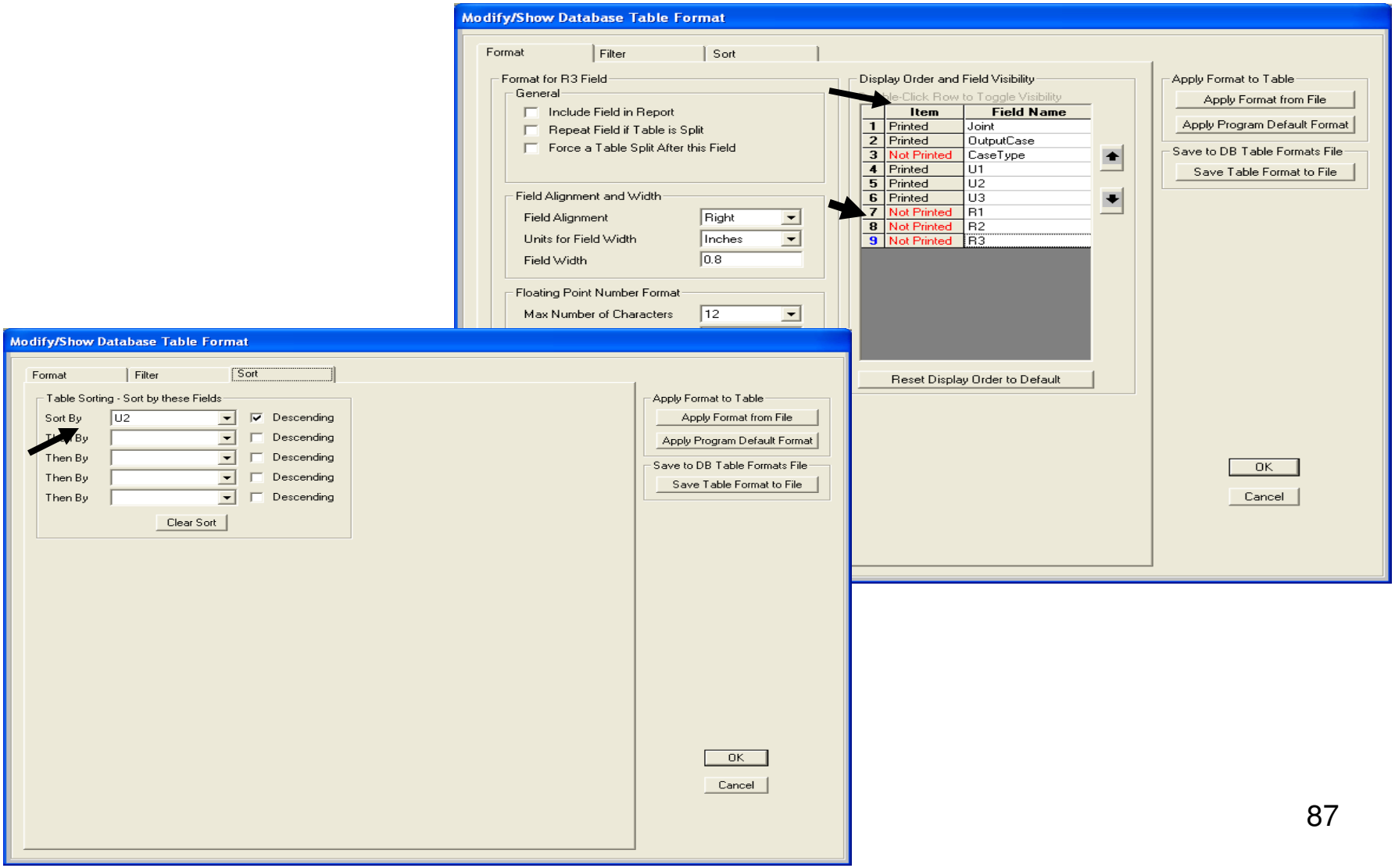


This is the default report format. As an example, let's clean it up by clicking Format-Filter-Sort and formatting it first by eliminating unwanted data which may clutter the report.

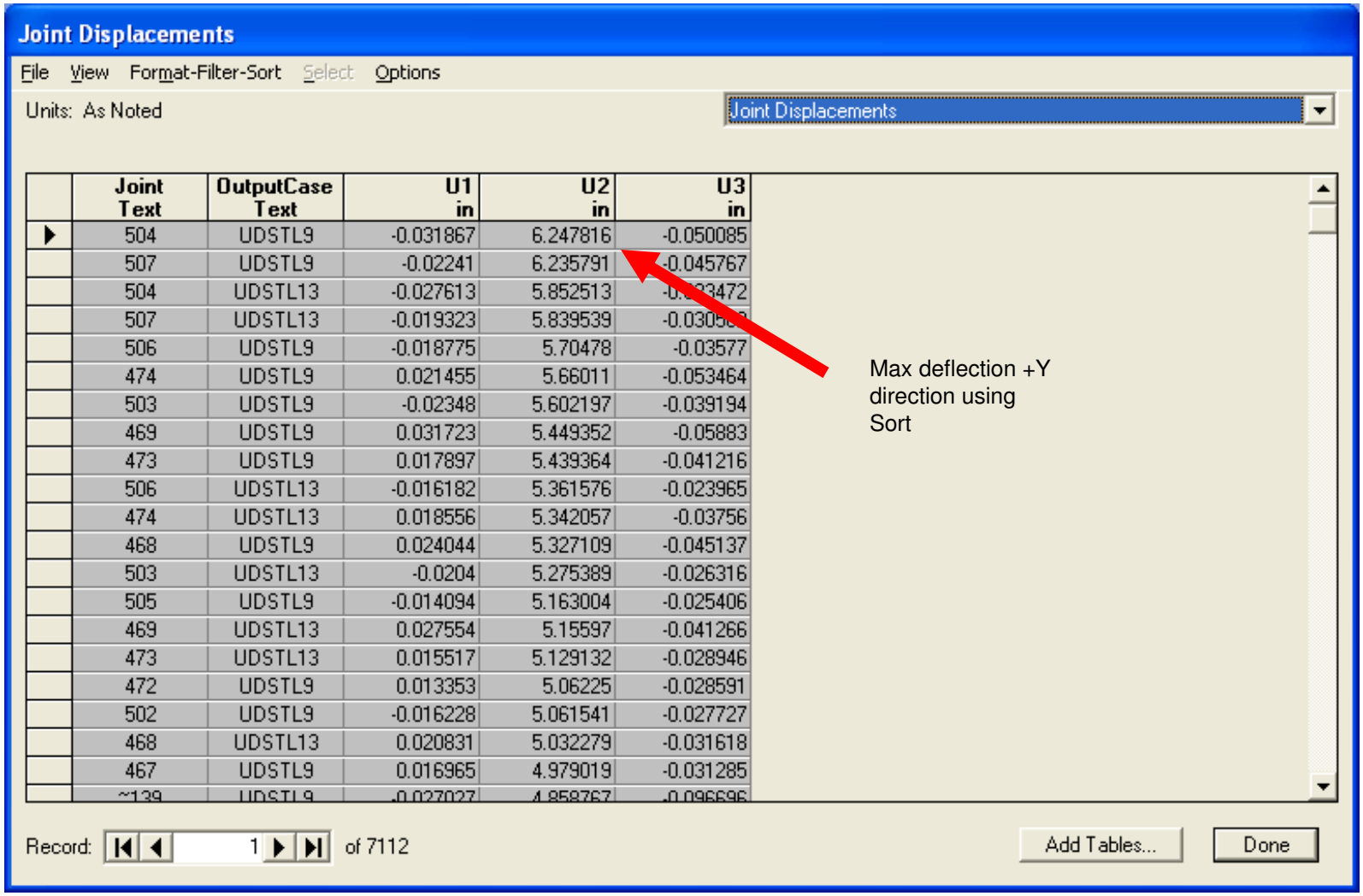


	Joint Text	OutputCase Text	CaseType Text	U1 in	U2 in	U3 in	R1 Radians	R2 Radians	R3 Radians
▶	3	DSTL1	Combination	0	0	0	0.000003185	-0.000003342	0.000002088
	3	DSTL5	Combination	0	0	0	0.000001973	-0.000003125	0.000001782
	3	DSTL9	Combination	0	0	0	-0.026398	-0.000103	-0.000134
	3	DSTL10	Combination	0	0	0	0.026403	0.000097	0.000138
	3	DSTL11	Combination	0	0	0	-0.013197	-0.000053	-0.000066
	3	DSTL12	Combination	0	0	0	0.013203	0.000047	0.00007
	3	DSTL13	Combination	0	0	0	-0.026398	-0.000103	-0.000135
	3	DSTL14	Combination	0	0	0	0.026402	0.000098	0.000138
	3	DSTL15	Combination	0	0	0	0.000001474	0.00013	-0.000042
	3	DSTL16	Combination	0	0	0	0.000003494	-0.000137	0.000046
	3	DSTL17	Combination	0	0	0	0.0000008102	0.000131	-0.000043
	3	DSTL18	Combination	0	0	0	0.000002829	-0.000135	0.000046
	3	DSTL19	Combination	0	0	0	0.000002275	-0.000002387	0.000001491
	3	DSTL23	Combination	0	0	0	0.000001802	-0.00000255	0.000001486
	4	DSTL1	Combination	0.006515	0.01386	-0.06809	-0.002711	0.000066	-0.000967
	4	DSTL5	Combination	0.007197	0.017805	-0.075227	-0.003068	0.000073	-0.001093
	4	DSTL9	Combination	-0.032438	5.556715	-0.062937	-0.00613	0.000015	-0.00111
	4	DSTL10	Combination	0.045623	-5.525548	-0.074868	0.000553	0.000119	-0.000878
	4	DSTL11	Combination	-0.013931	2.782446	-0.05538	-0.003994	0.000031	-0.000887
	4	DSTL12	Combination	0.0251	-2.758686	-0.061346	-0.000653	0.000083	-0.000771
	4	DSTL13	Combination	-0.034842	5.550042	-0.037807	-0.005094	-0.000009586	-0.000738

Click within the Item column to toggle certain items to be ‘Not printed’ (ie. not displayed) in red and press OK. Next, go back to Format-Filter-Sort and use Sort by U2 direction (U1, U2, U3 = joint axes X, Y, and Z unless local axis was rotated) with Descending check boxed to review max deflection positive Y direction. Uncheck Descending to review max deflection in negative direction Y assuming there are negative direction deflections

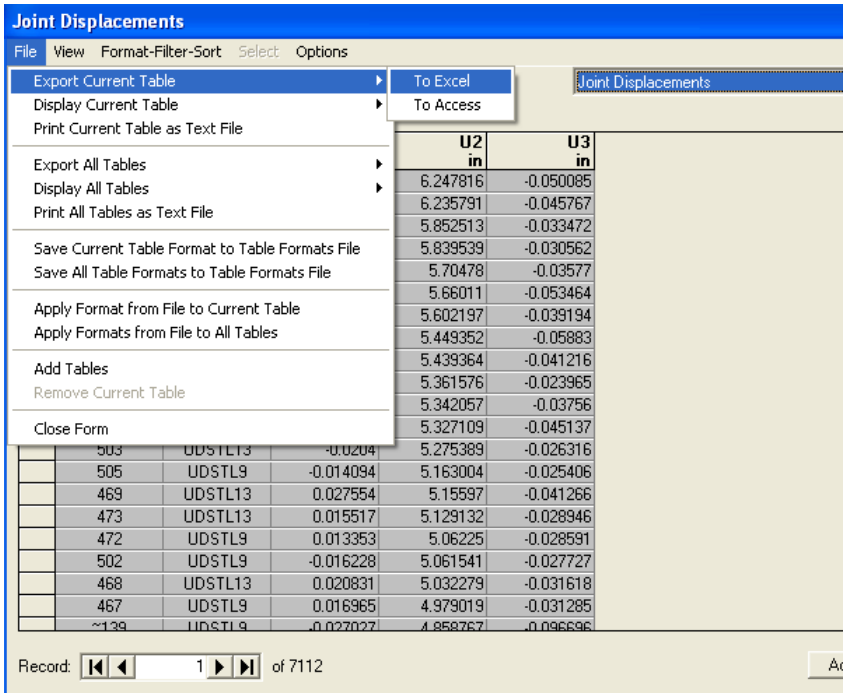


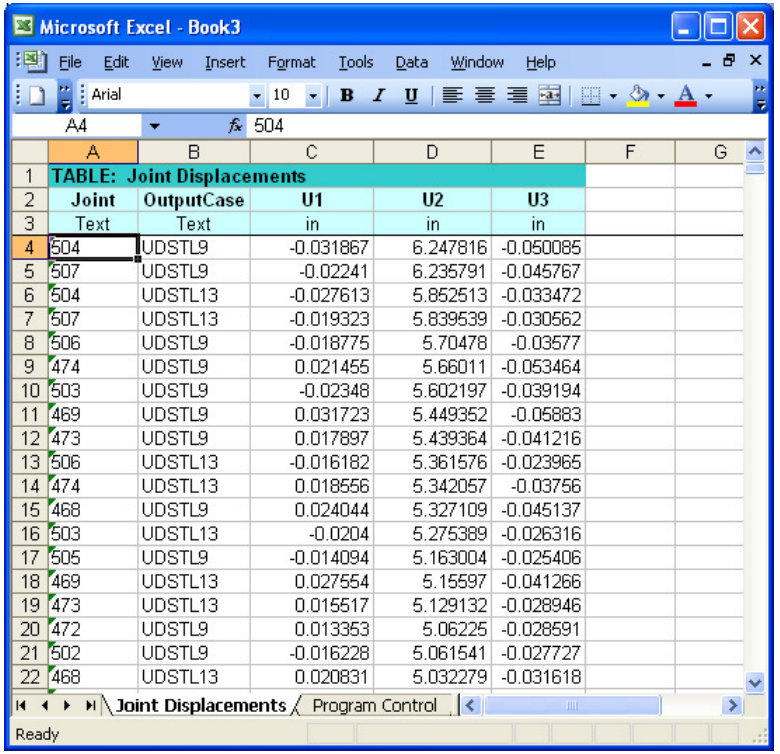
Sorted max deflection in the Y (U2) direction in this example. Uncheck the 'Descending' box in the sort dialogue in order to check deflections in – direction.



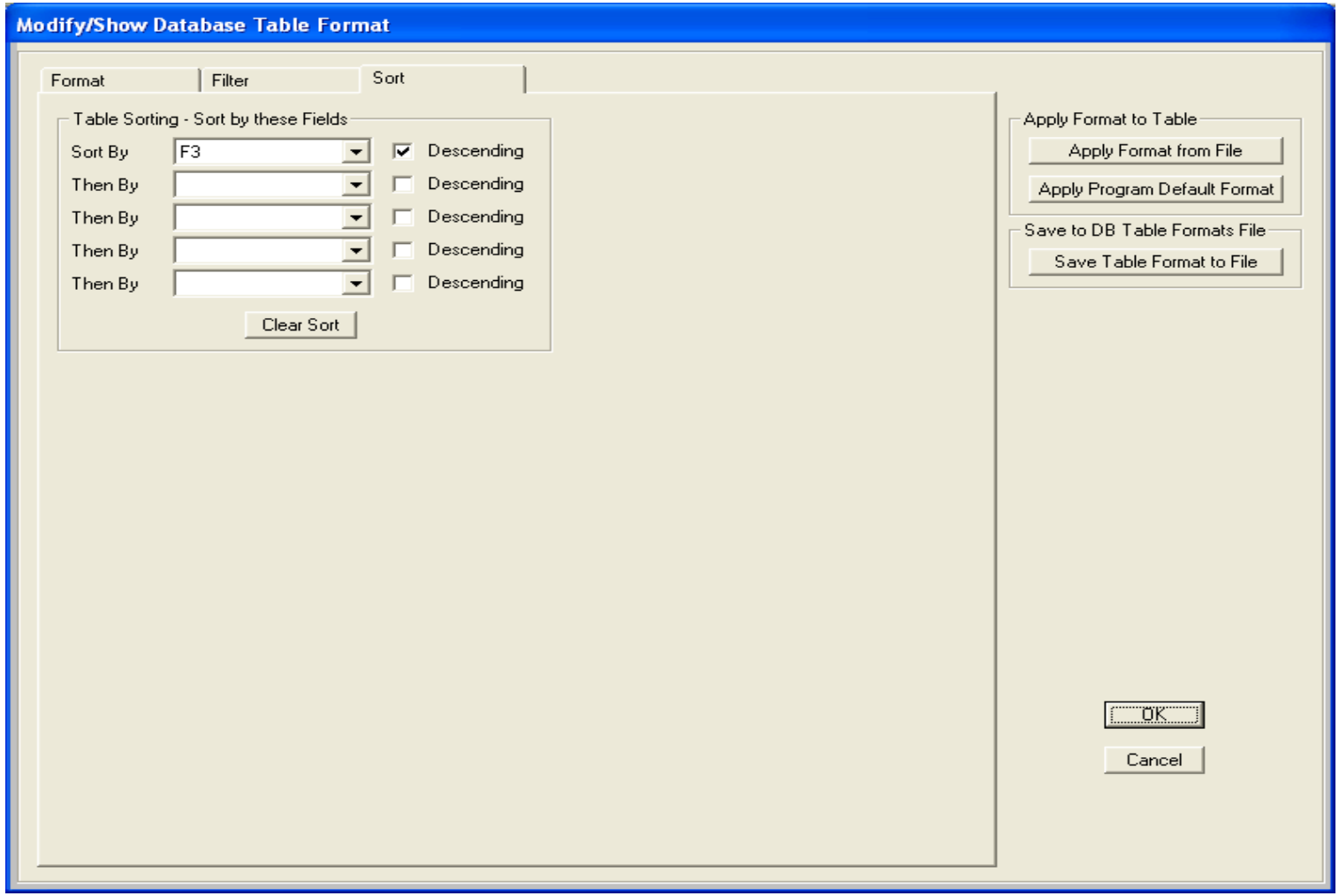
Joint Text	OutputCase Text	U1 in	U2 in	U3 in
504	UDSTL9	-0.031867	6.247816	-0.050085
507	UDSTL9	-0.02241	6.235791	-0.045767
504	UDSTL13	-0.027613	5.852513	-0.[illegible]3472
507	UDSTL13	-0.019323	5.839539	-0.030[illegible]
506	UDSTL9	-0.018775	5.70478	-0.03577
474	UDSTL9	0.021455	5.66011	-0.053464
503	UDSTL9	-0.02348	5.602197	-0.039194
469	UDSTL9	0.031723	5.449352	-0.05883
473	UDSTL9	0.017897	5.439364	-0.041216
506	UDSTL13	-0.016182	5.361576	-0.023965
474	UDSTL13	0.018556	5.342057	-0.03756
468	UDSTL9	0.024044	5.327109	-0.045137
503	UDSTL13	-0.0204	5.275389	-0.026316
505	UDSTL9	-0.014094	5.163004	-0.025406
469	UDSTL13	0.027554	5.15597	-0.041266
473	UDSTL13	0.015517	5.129132	-0.028946
472	UDSTL9	0.013353	5.06225	-0.028591
502	UDSTL9	-0.016228	5.061541	-0.027727
468	UDSTL13	0.020831	5.032279	-0.031618
467	UDSTL9	0.016965	4.979019	-0.031285
~139	UDSTL9	-0.027027	4.858767	-0.096696

Use File menu to automatically export to Excel as shown below without any manual copy & pasting or editing of text files. Every single aspect of the model input and analysis and design results can be automatically exported to Excel or Access or imported from Excel or Access. SAP2000 also comes with a well documented application programming interface (API) which facilitates integrating SAP2000 with 3rd party applications as well as plug-in options. API documentation installs in the SAP2000 directory in a file named *SAP2000_API_Documentation.chm*

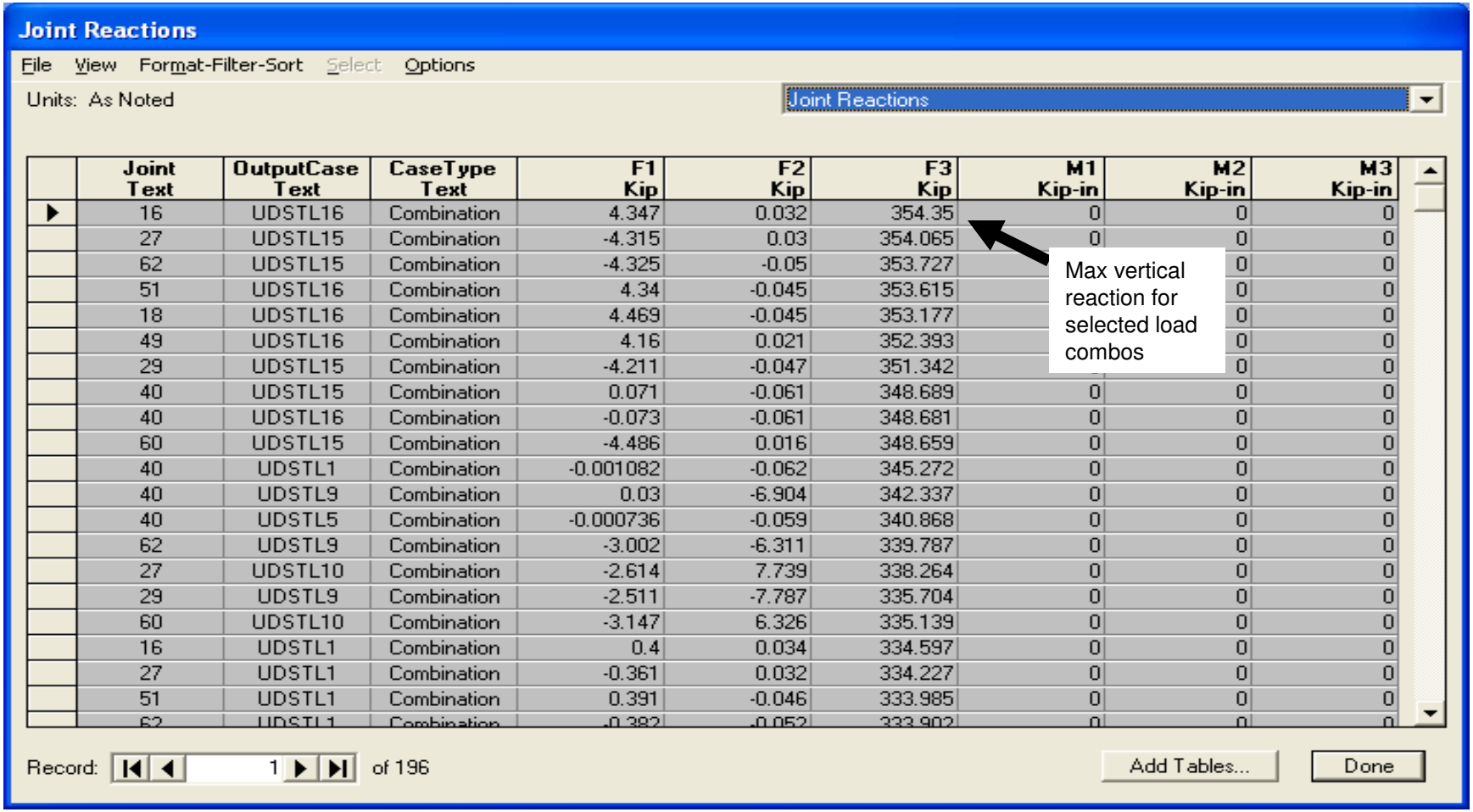




Use the pull-down to switch from joint displacements to joint reactions and Format-Filter-Sort, then Sort for F3 Descending as shown. F1, F2, F3 correspond to global X, Y, and Z unless local axis of joints have been rotated.



Joint reactions now sorted by max vertical reaction. Alternatively use File menu>Create report to generate output reports using SAP2000's default settings for output report generation. You can generate custom output reports and save formats using the Advanced report writer.



Joint Text	OutputCase Text	CaseType Text	F1 Kip	F2 Kip	F3 Kip	M1 Kip-in	M2 Kip-in	M3 Kip-in
16	UDSTL16	Combination	4.347	0.032	354.35	0	0	0
27	UDSTL15	Combination	-4.315	0.03	354.065	0	0	0
62	UDSTL15	Combination	-4.325	-0.05	353.727		0	0
51	UDSTL16	Combination	4.34	-0.045	353.615		0	0
18	UDSTL16	Combination	4.469	-0.045	353.177		0	0
49	UDSTL16	Combination	4.16	0.021	352.393		0	0
29	UDSTL15	Combination	-4.211	-0.047	351.342		0	0
40	UDSTL15	Combination	0.071	-0.061	348.689	0	0	0
40	UDSTL16	Combination	-0.073	-0.061	348.681	0	0	0
60	UDSTL15	Combination	-4.486	0.016	348.659	0	0	0
40	UDSTL1	Combination	-0.001082	-0.062	345.272	0	0	0
40	UDSTL9	Combination	0.03	-6.904	342.337	0	0	0
40	UDSTL5	Combination	-0.000736	-0.059	340.868	0	0	0
62	UDSTL9	Combination	-3.002	-6.311	339.787	0	0	0
27	UDSTL10	Combination	-2.614	7.739	338.264	0	0	0
29	UDSTL9	Combination	-2.511	-7.787	335.704	0	0	0
60	UDSTL10	Combination	-3.147	6.326	335.139	0	0	0
16	UDSTL1	Combination	0.4	0.034	334.597	0	0	0
27	UDSTL1	Combination	-0.361	0.032	334.227	0	0	0
51	UDSTL1	Combination	0.391	-0.046	333.985	0	0	0
62	UDSTL1	Combination	-0.382	-0.052	333.902	0	0	0

Please feel free to contact me with any modeling questions regarding SAP2000, or our SAFE program for design of basemat foundations on soil or piles. Also, feedback on this tutorial would be appreciated, as we can customize similar tutorials to meet your needs. We are very interested in working with you to increase usage of SAP2000 and SAFE wherever you believe it may improve your work process. As you can see from this tutorial, SAP2000 can be effectively used for day-to-day modeling and design applications as well as for advanced nonlinear and dynamic analysis.

We encourage you to check out our design and user manuals which install in the Manuals subdirectory of the SAP2000 installation folder

Watch & Learn video tutorials here: http://www.csiberkeley.com/support_watch&learn.html

Darrell Foster
CSI Houston, TX office
Tel: 713-533-4420
Computers & Structures, Inc.
TECHNOLOGY FOR A BETTER WORLD

darrell@csiberkeley.com
www.csiberkeley.com

CSI is an ISO-9001 certified company.