
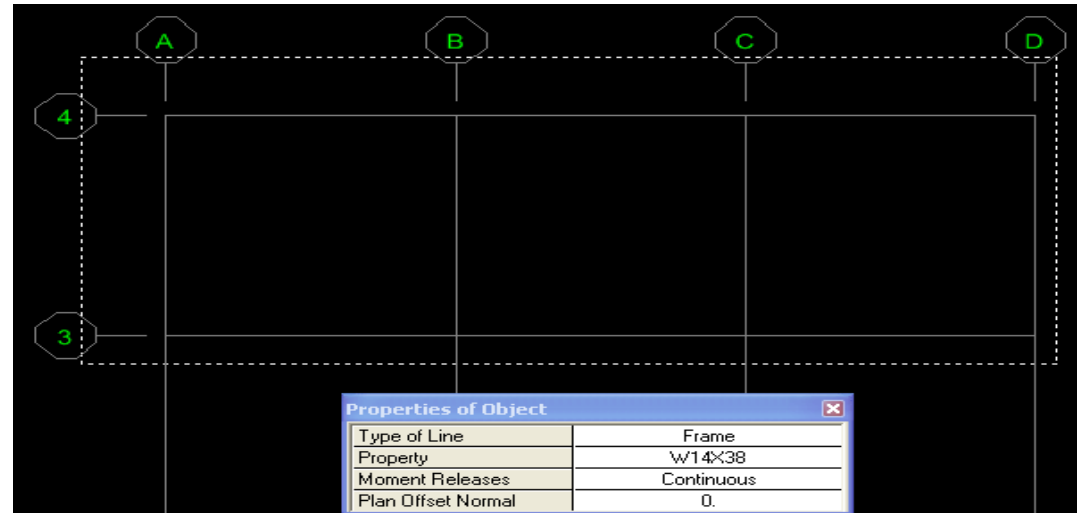
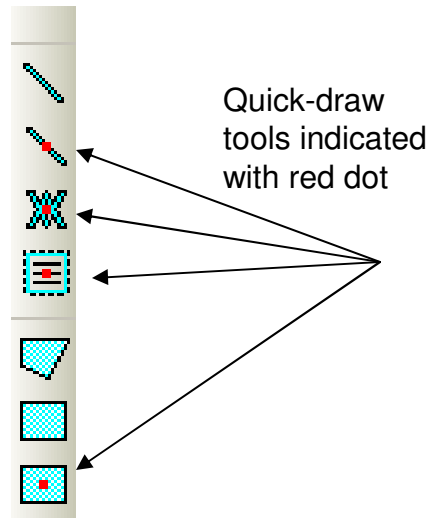
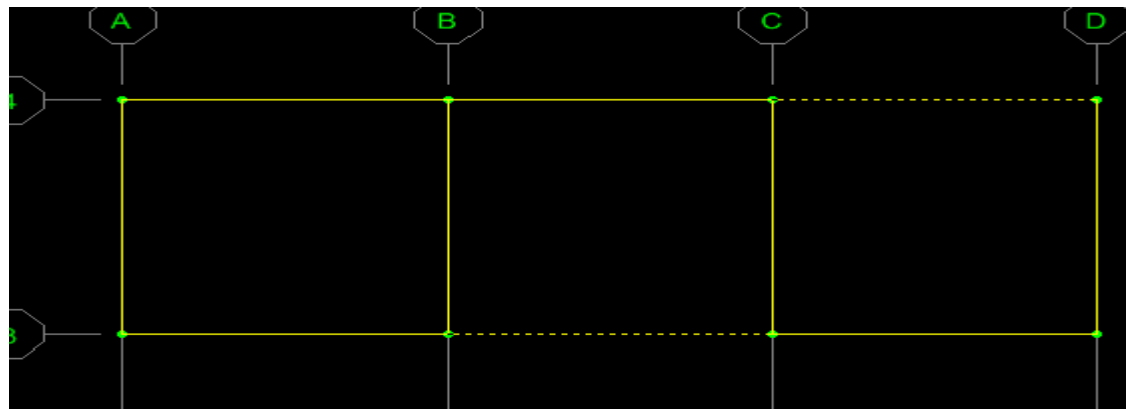


**Quick-draw tools** enables users to window around visible gridlines to draw, or draw by a single mouse click on or between visible gridlines. In ETABS, the quick-draw tools are referred to as draw “at click” The quick-draw tools have the benefit of enabling the user to quickly draw multiple elements without having to mouse-click each element 2 or more times to draw. The issue to be aware of with use of quick-draw tools is that they divide elements at each visible grid intersection

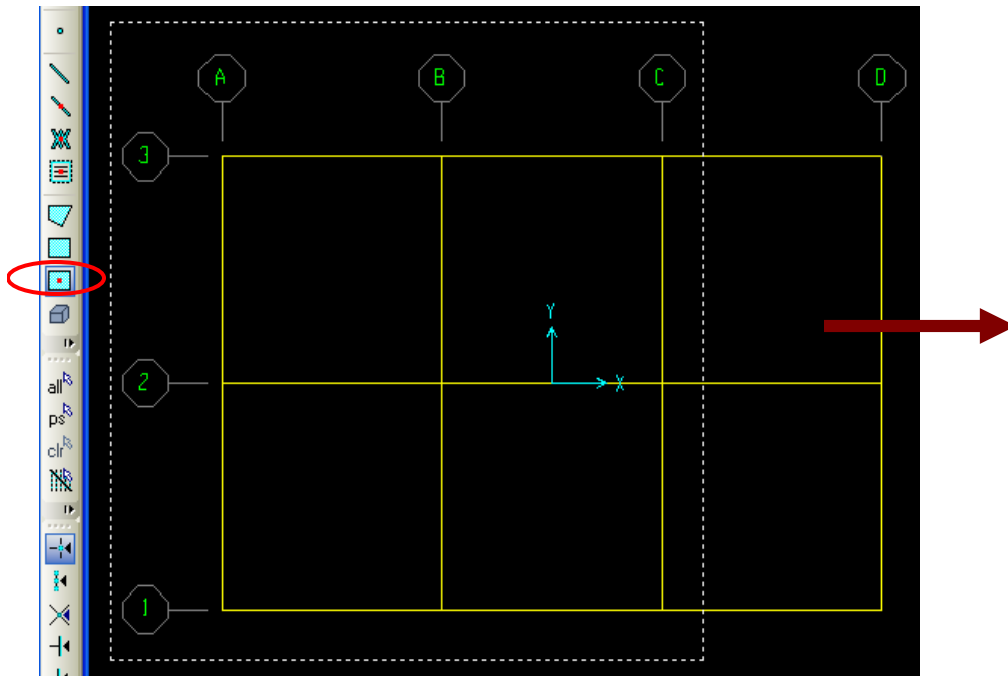
Click this tool  then window around the area where you want to add elements



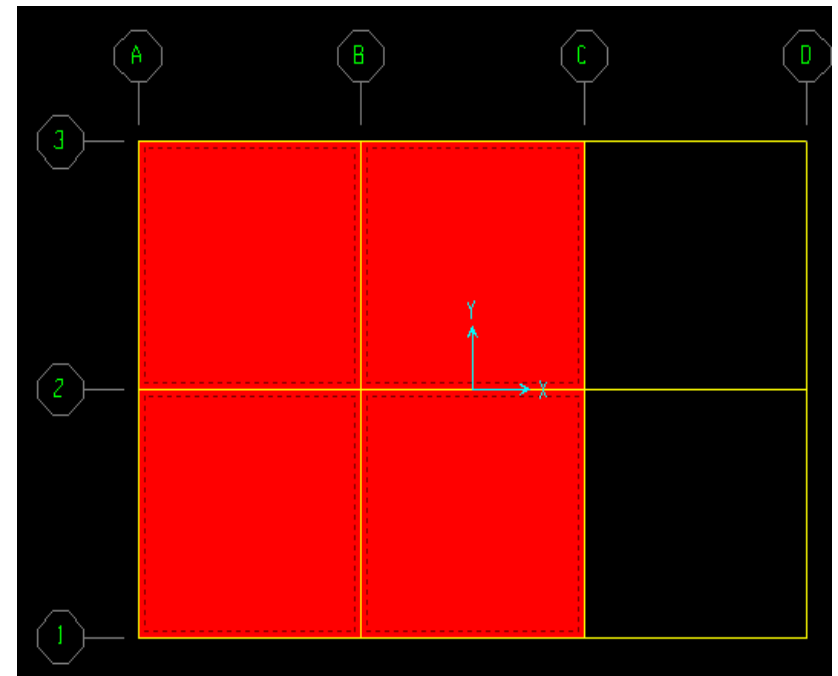
Activating joints to be visible and selecting a couple of members (dotted lines) makes clear that members are divided at grid intersections



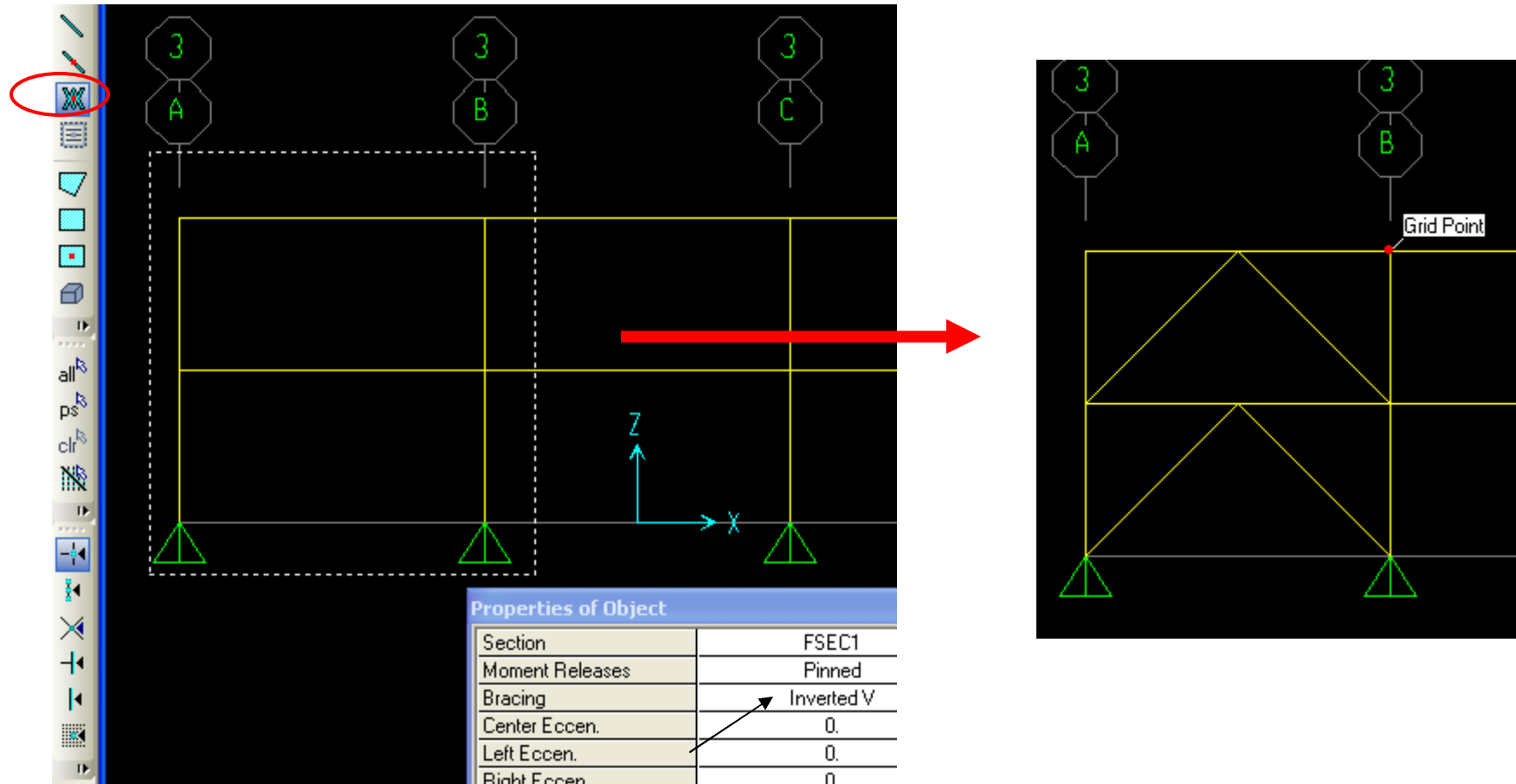
Using SAP2000 or ETABS, use of the quick-draw area element enables users to quickly model elements by windowing around a selected area to draw separate area elements divided at each visible gridline. Fill option was activated using Set Display option button ☒ (aka 'Set building view options' in ETABS)




4 areas in this example

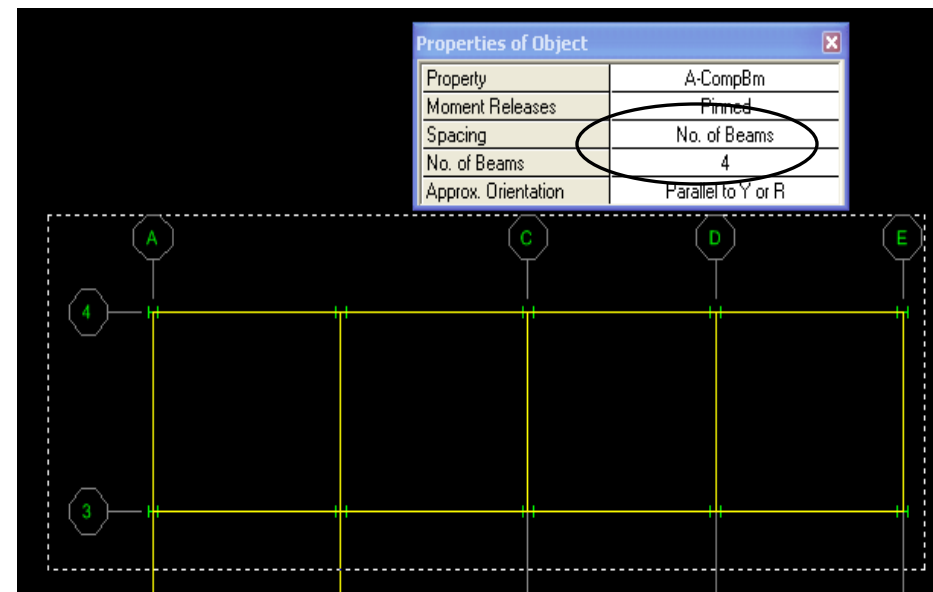
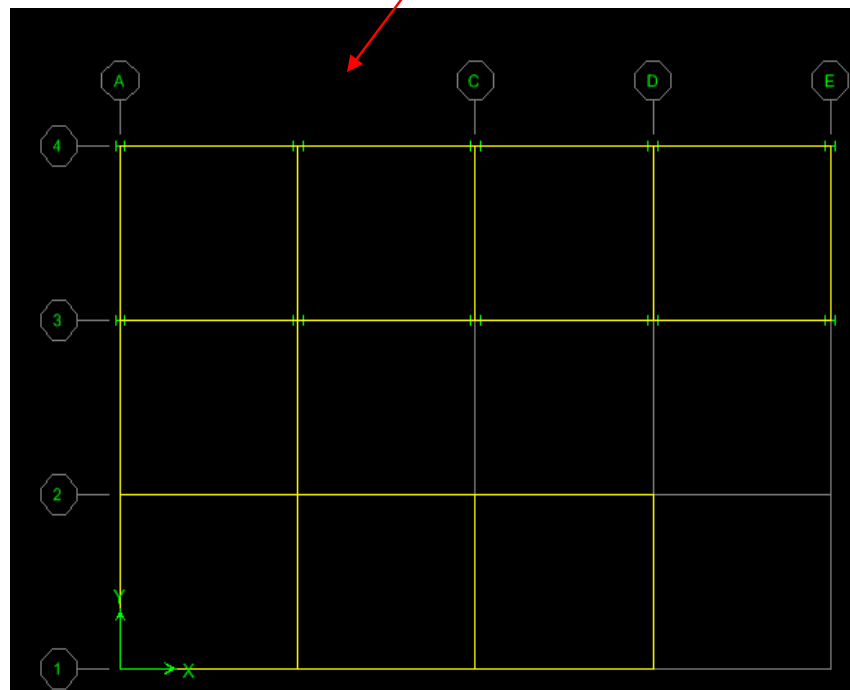


Quick-draw bracing in elevation view using Inverted V bracing option. Bracing is added between each visible gridline within the windowed area

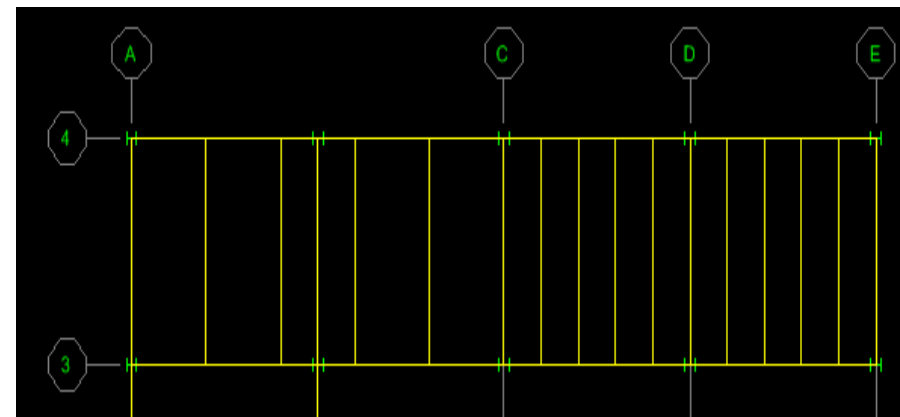


If you double-click a gridline in either SAP2000 or ETABS, you can modify grids, including the option to “Hide” from visibility. Note what happens when we try quick-draw modeling secondary beams  based on 4 beam setting shown below right. Because the spacing between visible gridlines varies in this example because the B grid is hidden, the spacing of the beams vary too. There is another option on this draw tool to draw based on Max spacing. You can use this technique to model bracing or any other quick-draw option. In V14 of SAP2000, the hiding of gridlines is not recognized by the quick-draw tools (temporary bug).


X Grid Data					
	Grid ID	Ordinate	Line Type	Visibility	Bubble I
1	A	0.	Primary	Show	Top
2	B	288.	Primary	Hide	Top
3	C	576.	Primary	Show	Top
4	D	864.	Primary	Show	Top
5	E	1152.	Primary	Show	Top
6					
7					
8					

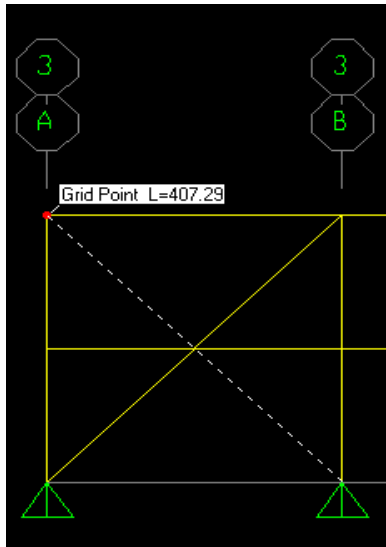



Secondary beams added

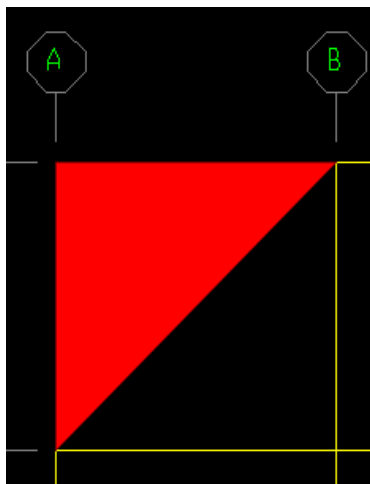


**Other draw tools** - So when do you use the draw tools which don't have the red dots? Whenever you want to draw elements that do not coincide with or divide at visible gridlines or intersections with other elements. Draw frame tool can be used to draw beams, bracing, or columns. Below are some examples

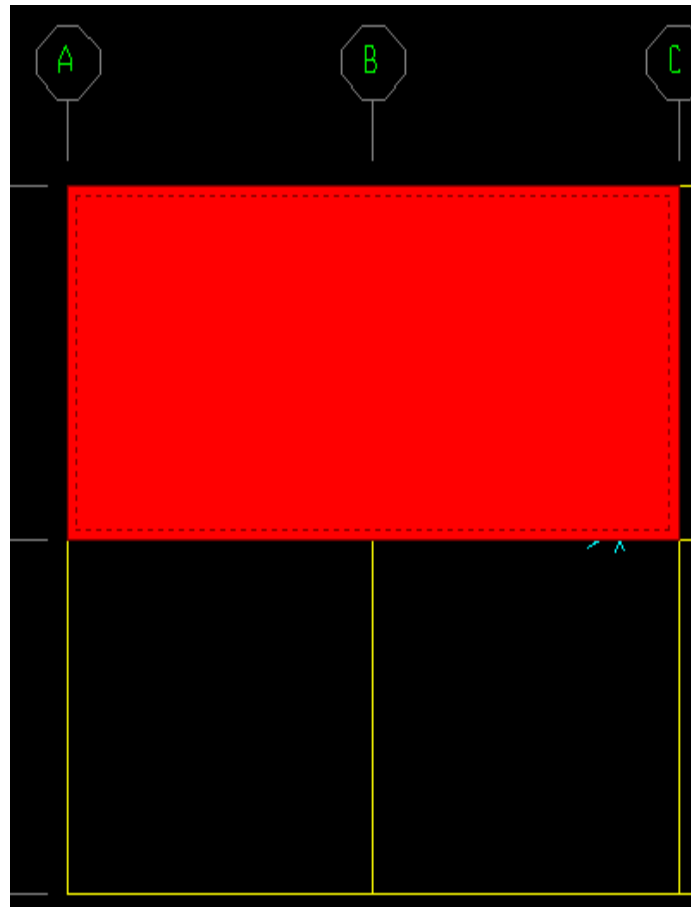
Bracing that spans multiple levels using draw frame tool 




Using Draw areas tool 

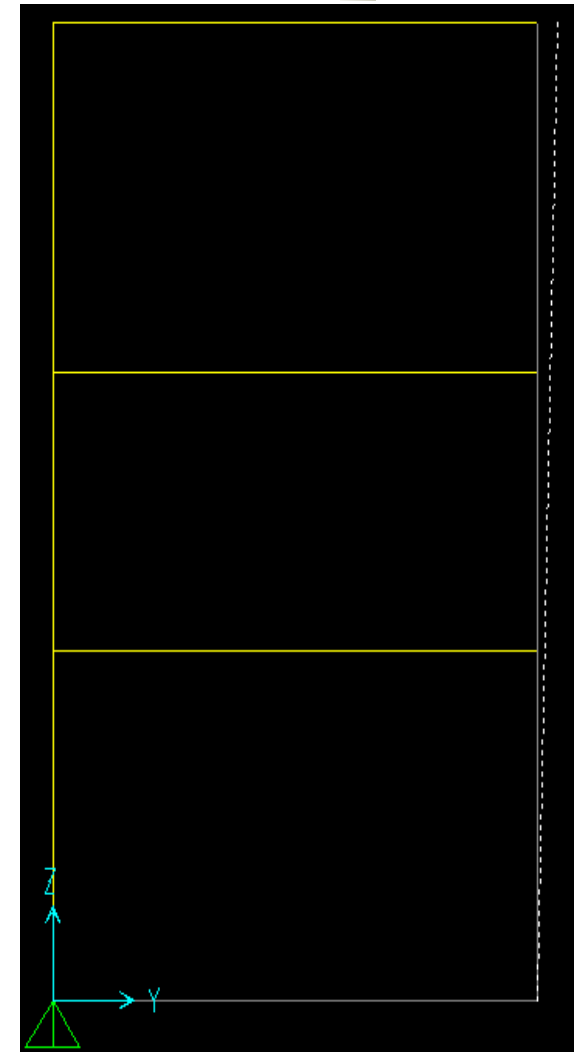


Areas that span multiple gridlines so as not to distribute load to intermediate beams or bracing



Draw rectangular area tool lets you drag from one corner to another 

Columns that span multiple levels or beams that span multiple grids 



In this example, the columns will automatically mesh internally to account for the beams, but they will be reformulated to report and design as 1 column

**Area finite element types** - Shell type elements have both in-plane and out-of-plane stiffness which transfers both shear and moments. By default, shells transfer load two-way only at meshed joint locations. Membrane elements have in-plane stiffness and transfer shear load (not moments) as one-way uniform distributed load to beams. Plate elements have out-of-plane stiffness only. Since shell and membrane are predominantly used, we'll limit discussion to those two types. In SAP2000 and ETABS, be aware that membrane type area elements are auto-meshed by default but shell type area elements are not. In ETABS, the "Deck" type area element is a membrane. A thick shell element includes transverse shear effects, which is typically an issue only in applications such as deep mat foundations. In most applications involving slabs, mat foundations, walls or tanks/vessels, thin shell is applicable. Membrane is used to model shear walls if users want to avoid taking credit for wall out-of-plane stiffness, and when the load transfer from wall to floor is shear load only without moments. Membrane is also used to model floors which distribute shear load one-way without taking credit for the floor resisting moments.

**Shell Section Data**

Section Name: ASEC1

Section Notes: Modify/Show...

Display Color: [Green]

Type:

- ☒ Shell - Thin
- ☐ Shell - Thick
- ☐ Plate - Thin
- ☐ Plate Thick
- ☐ Membrane
- ☐ Shell - Layered/Nonlinear

Modify/Show Layer Definition...

Material:

Material Name: + 4000Psi

Material Angle: 0.

Thickness:

Membrane: 1.

Bending: 1.

Concrete Shell Section Design Parameters:

Modify/Show Shell Design Parameters...

Stiffness Modifiers: Set Modifiers...

Temp Dependent Properties: Thermal Properties...

OK Cancel

SAP2000 area type definition

**Define Wall/Slab/Deck Sections**

Sections:

- DECK1
- PLANK1
- SLAB1
- WALL1

Click to:

Add New Deck

Modify/Show Section...

Delete Section

OK Cancel

With ETABS, you must work with different object types. Deck object is always membrane. Other area objects give you a choice

**Wall/Slab Section**

Section Name: SLAB1

Material: CONC

Thickness:

Membrane: 1.

Bending: 1.

Type:

- ☒ Shell
- ☐ Membrane
- ☐ Plate

☐ Thick Plate

Load Distribution:

☒ Use Special One-Way Load Distribution

Set Modifiers... Display Color: [Green]

OK Cancel

ETABS slab object lets you choose whether shell or membrane, and also gives choice of One-way load distribution, an option not currently available with SAP2000 shell type

**Area meshing** - Users have the option to mesh shell area elements to make them distribute load one-way as demonstrated below left using Edit>Edit areas in SAP2000 (Edit>Mesh areas in ETABS). In addition, users can use the modifier option to reduce stiffness in any direction. The modifier option can be used to specify reduced section for cracking, and/or reduced membrane stiffness if the engineer does not want to take credit for the slab or deck in-plane resistance to lateral loads.

**Divide Selected Areas**

☒ Divide Area Into This Number of Objects (Quads and Triangles Only)

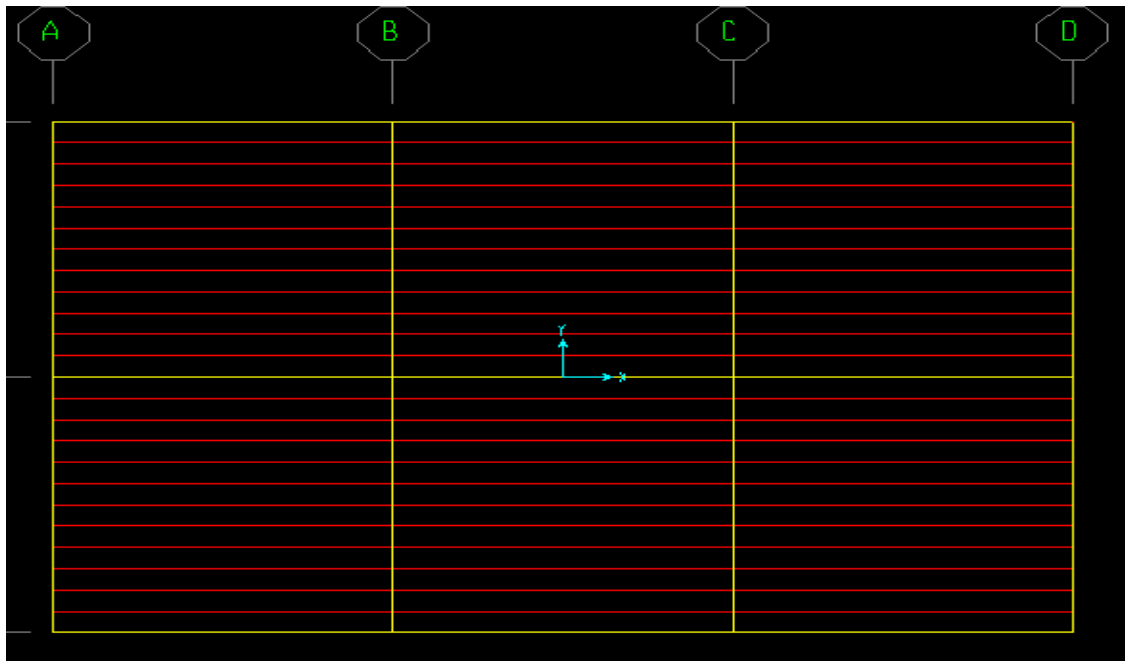
Along Edge from Point 1 to 2

Along Edge from Point 1 to 3

☐ Divide Area Into Objects of This Maximum Size (Quads and Triangles Only)

Along Edge from Point 1 to 2

Meshing as long strips forces 1-way load distribution



Reduced bending stiffness of shell to make it behave like a membrane. Shell will still transfer load only at meshed joint locations.

**Property/Stiffness Modification Factors**

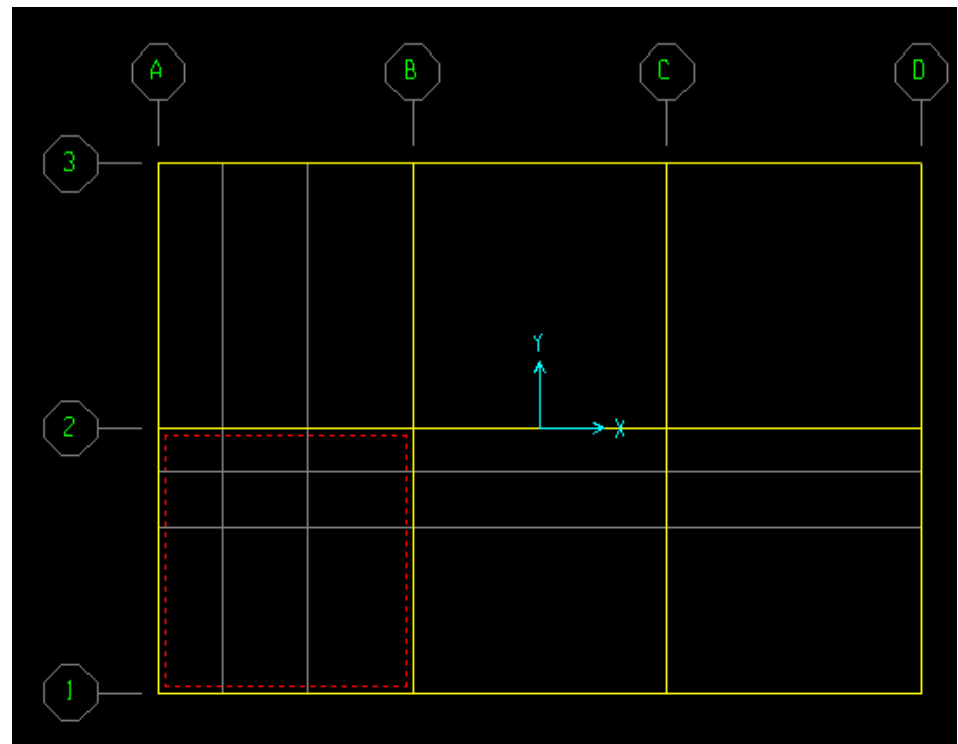
Property/Stiffness Modifiers for Analysis

Membrane f11 Modifier	<input type="text" value="1"/>
Membrane f22 Modifier	<input type="text" value="1"/>
Membrane f12 Modifier	<input type="text" value="1"/>
Bending m11 Modifier	<input type="text" value=".001"/>
Bending m22 Modifier	<input type="text" value=".001"/>
Bending m12 Modifier	<input type="text" value=".001"/>
Shear v13 Modifier	<input type="text" value="1"/>
Shear v23 Modifier	<input type="text" value="1"/>
Mass Modifier	<input type="text" value="1"/>
Weight Modifier	<input type="text" value="1"/>

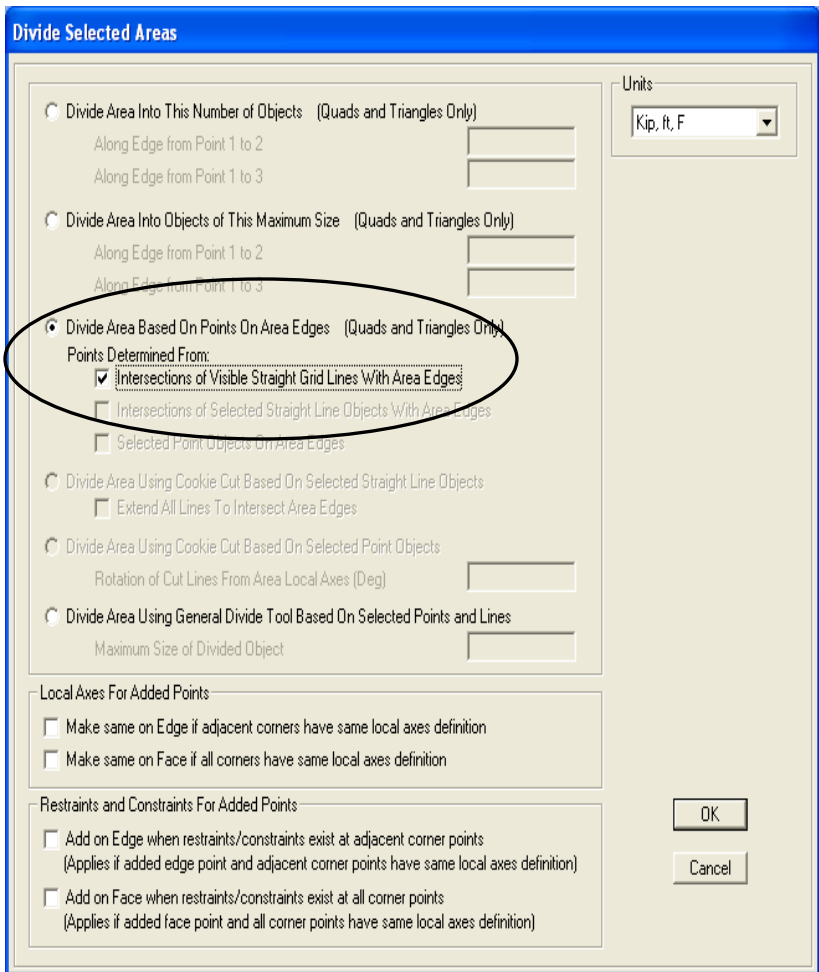
OK Cancel

SAP2000 and ETABS offer several meshing tools. In addition to the 4X4, 2X8, etc. type of meshing, users can add gridlines or draw lines to help them more easily mesh area elements. In the example below, gridlines are added and then Edit>Edit areas to mesh by Intersections of Visible gridlines as shown below right.

Add gridlines to help you mesh precisely where needed. Select area(s) to be meshed

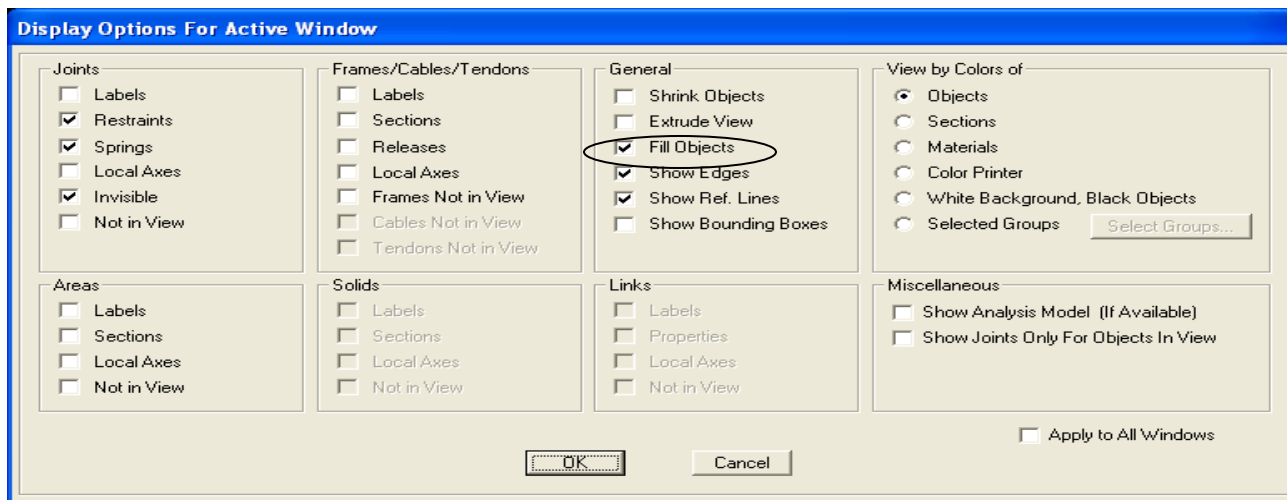


Edit menu>Edit areas in SAP2000. Similar options in ETABS

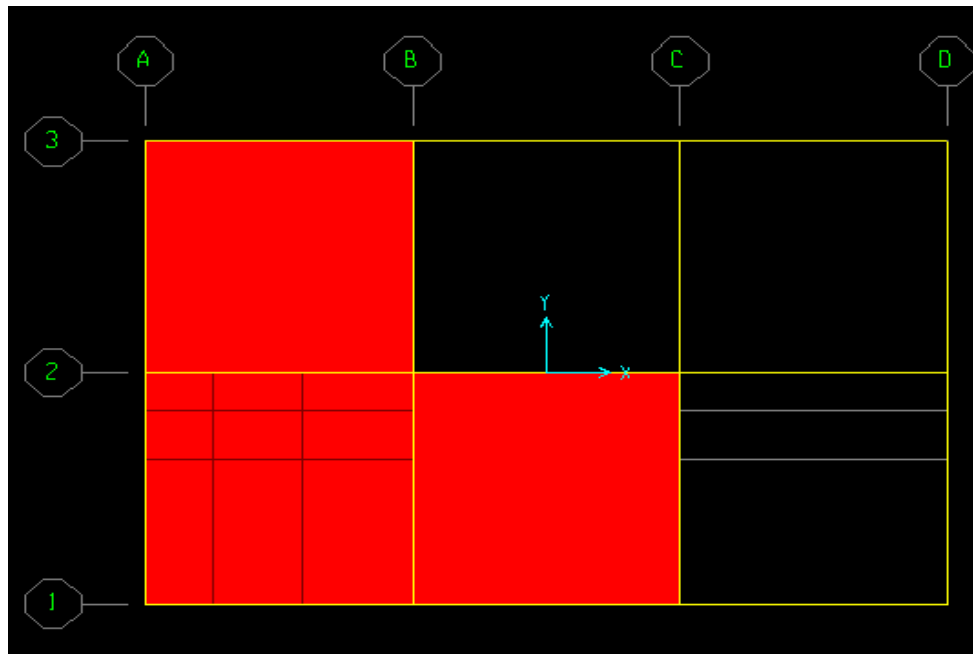




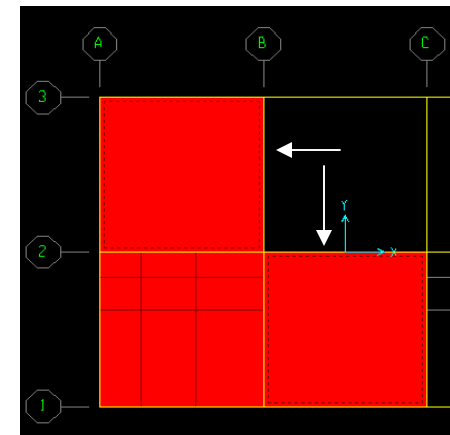
Use Set View options ☒ to activate Fill Objects to view mesh. Similar options in SAP2000, ETABS and SAFE



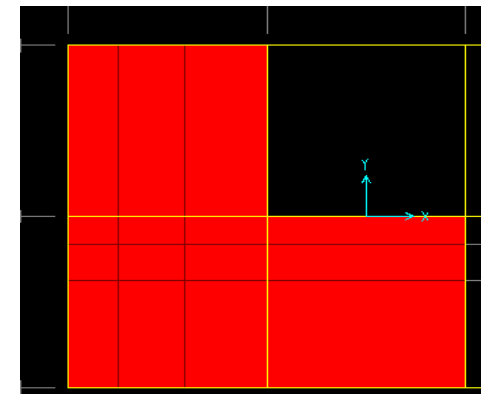
In this example below, only 1 shell area is meshed. Joints are automatically added to perimeter beam objects, but adjacent area shells need to be meshed in order to transfer loads at mesh points



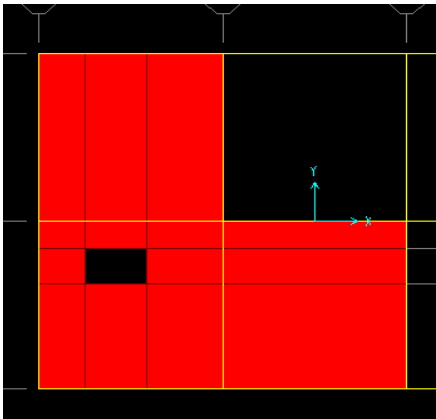
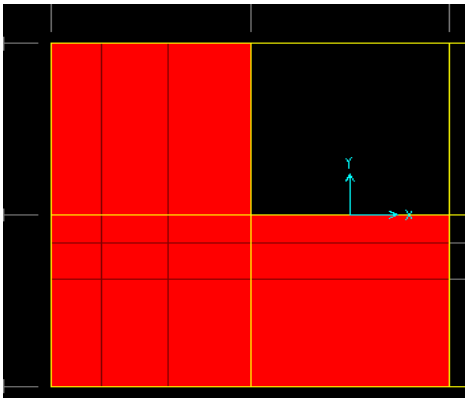
Select 2 adjacent shell areas



Use Edit menu again to mesh by gridlines



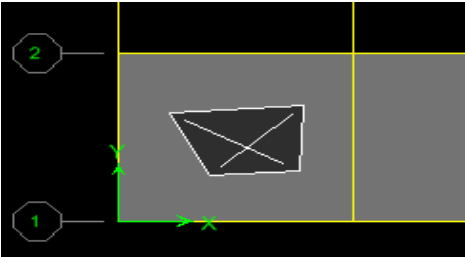
In order to assign different loads to different areas, to create openings, to assign a joint load or support, or transfer floor load to a beam or wall or column, you need to mesh for connectivity. One exception is with floor openings in ETABS. Floor openings in ETABS are drawn as an area with property “Opening” as shown below. In SAP, floor openings are created by meshing and deleting areas. Wall openings are created by mesh and delete in both SAP2000 and ETABS



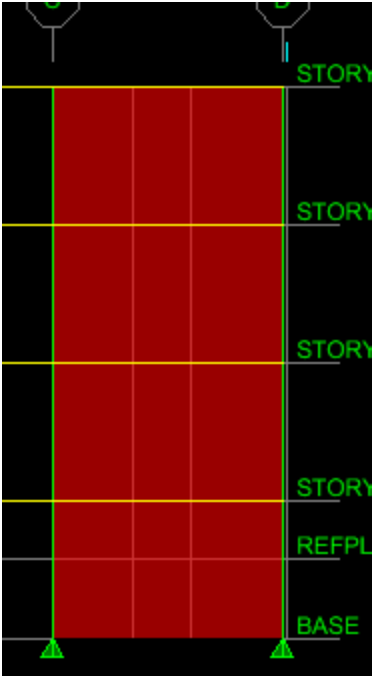
Select a meshed area and press Del key on keyboard to create opening

With ETABS, floor openings are drawn with this property

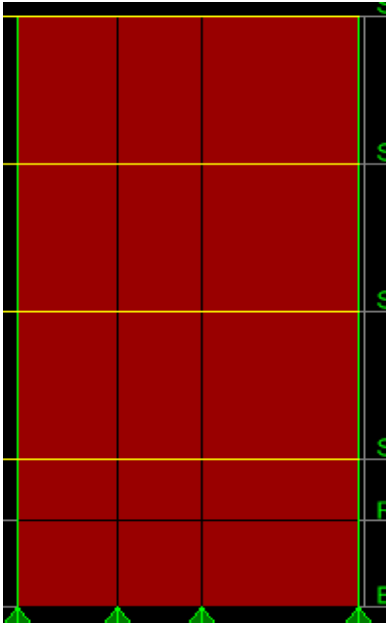
Properties of Object	
Property	OPENING
Local Axis	0.
Drawing Control	None <space bar>



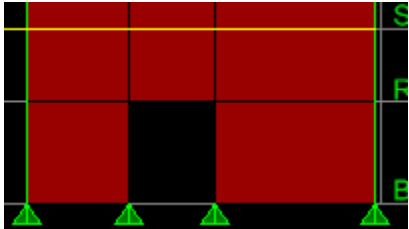
With ETABS, use Edit menu to add reference planes and reference lines to help mesh vertical shear walls by gridline. With SAP2000, just add gridlines



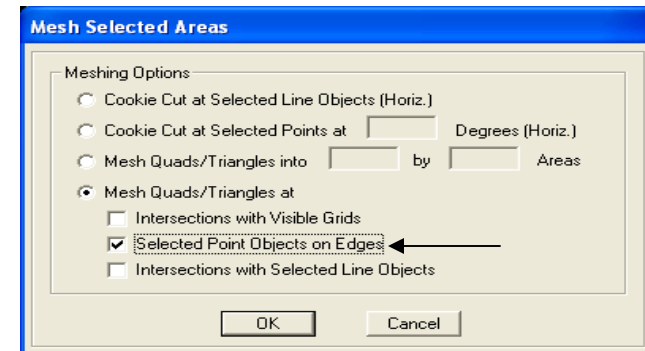
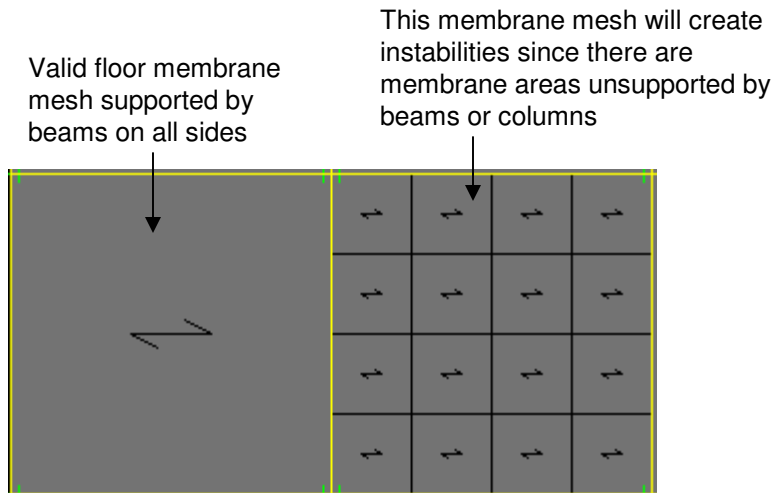
Mesh by gridlines



Select and delete for openings

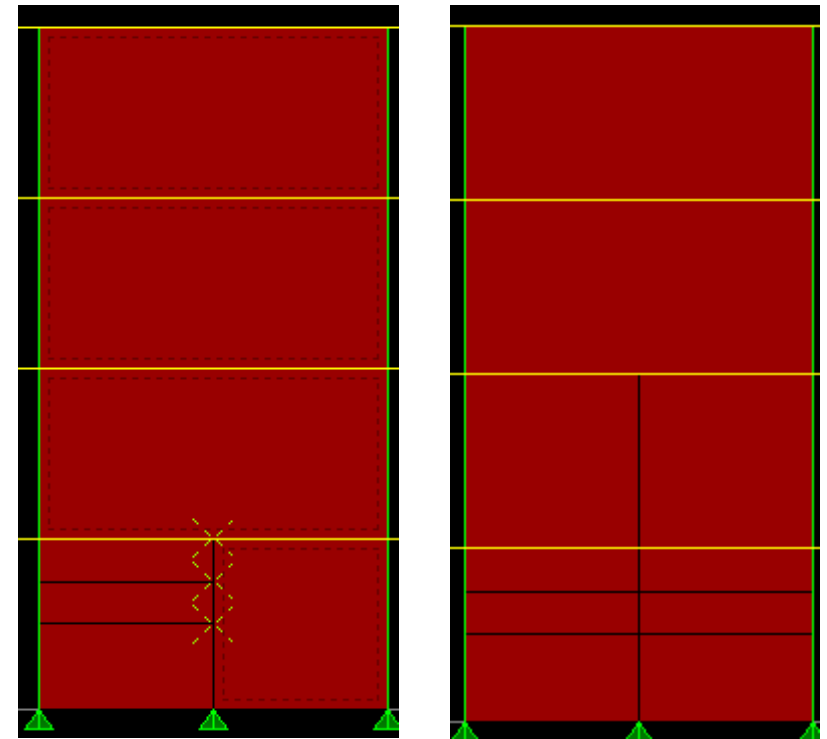
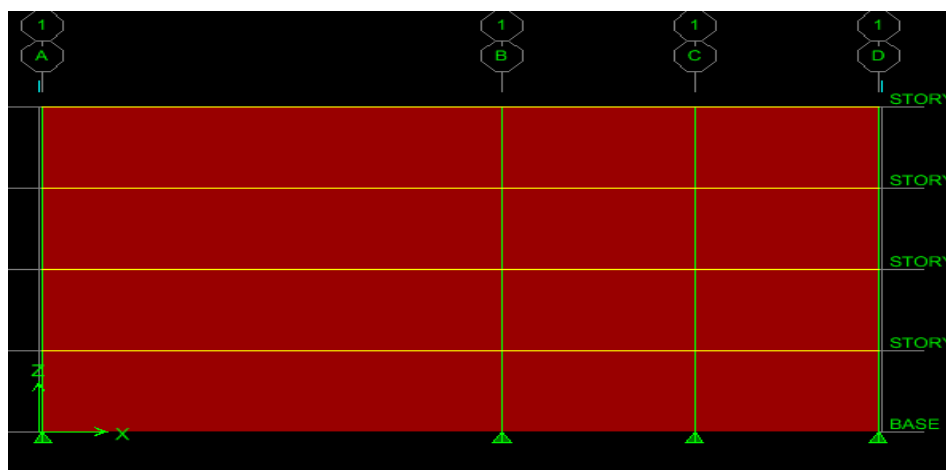


One of the most common meshing errors involves membrane type areas. Because membrane areas have no out-of-plane stiffness, each floor membrane mesh area needs to be supported on all corners by beams, columns, or support/restraint. Wall membrane elements can take in-plane load, but not load normal to the element face.

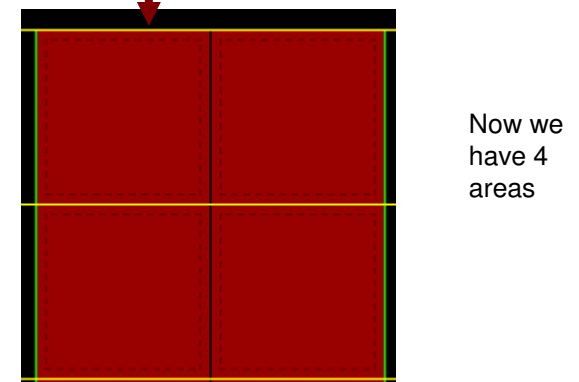
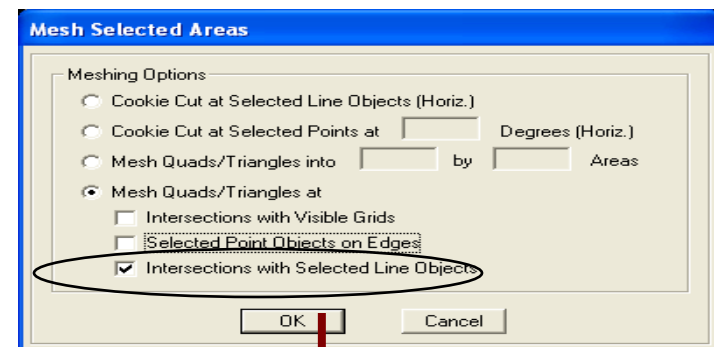
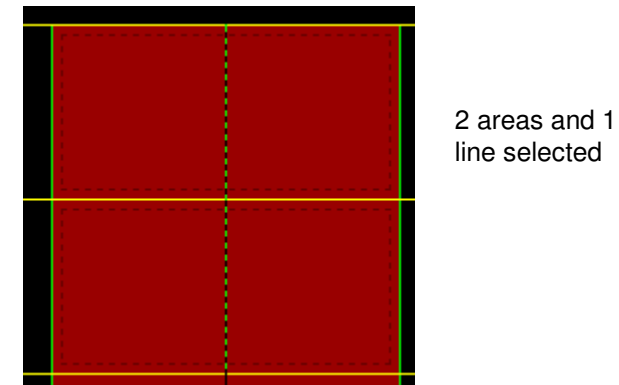
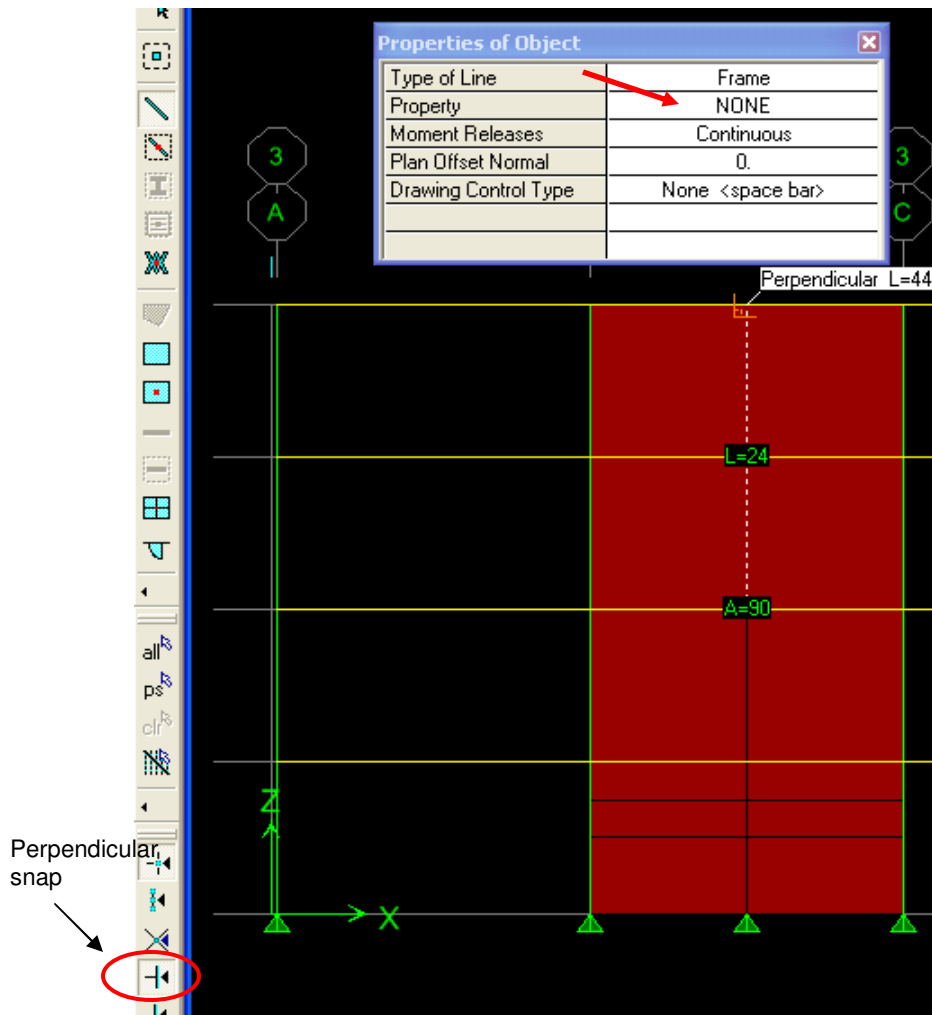


Wall mesh needs to be carried through adjacent elements. Select joints and areas to mesh by "Selected Point objects on Edges"

In this shear wall example, gravity loads are lumped at each support location. Long unsupported lengths of walls can act like beams. Mesh every 6 – 8 ft.' or so (engineering judgment) and add supports along the base to more realistically model support conditions at the base of walls

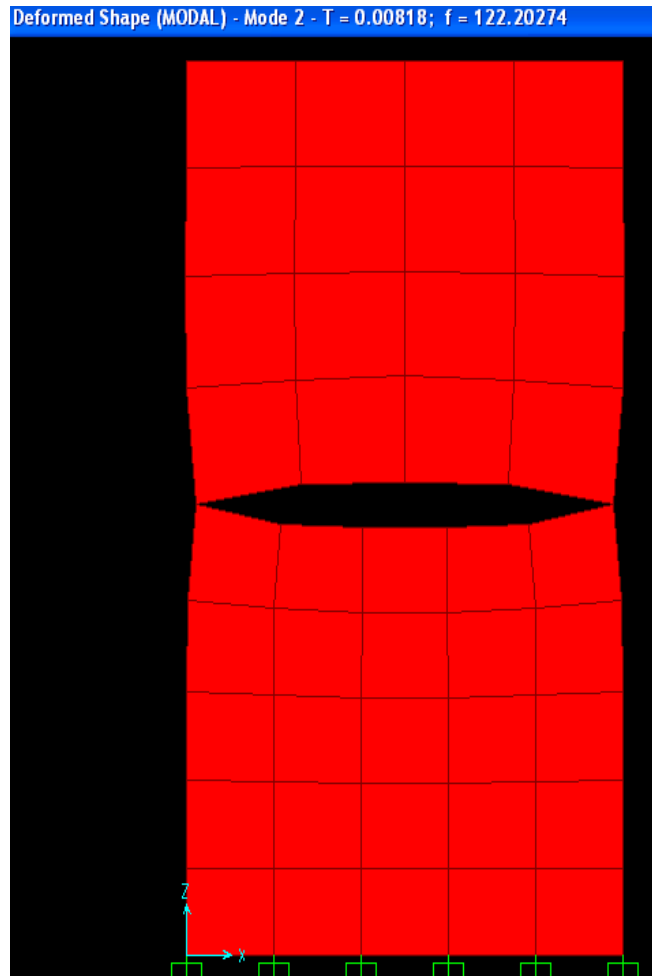
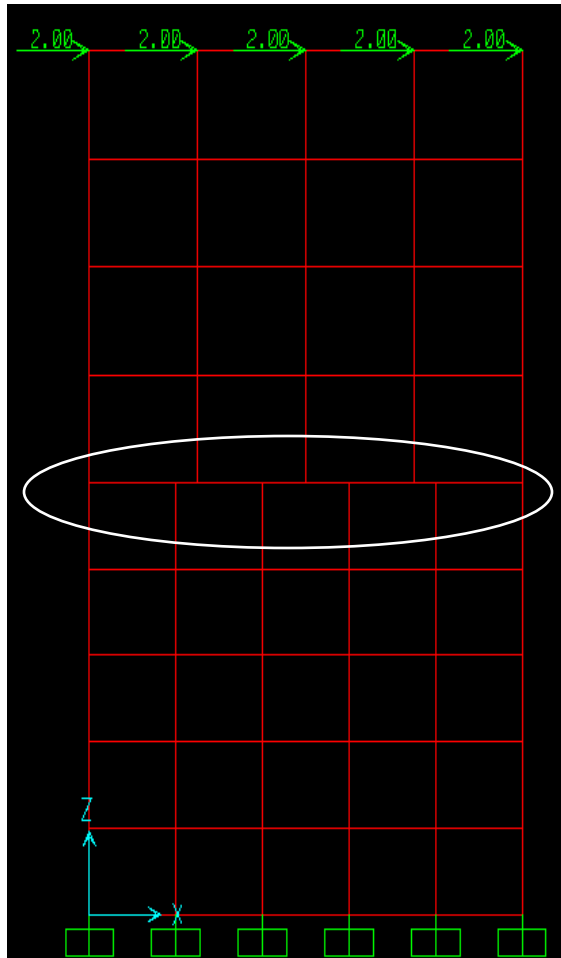


In the wall example in the previous slide, it appears that ETABS only meshed adjacent elements. This may be a bug, but there's an easy workaround. Activate Snap to perpendicular directions button, then Draw line (Draw frame/cable in SAP2000) with Property "None" and draw the line to continue the mesh through adjacent elements. Select the areas and the None property line and use Edit menu mesh option to mesh by "Intersections with selected line objects".

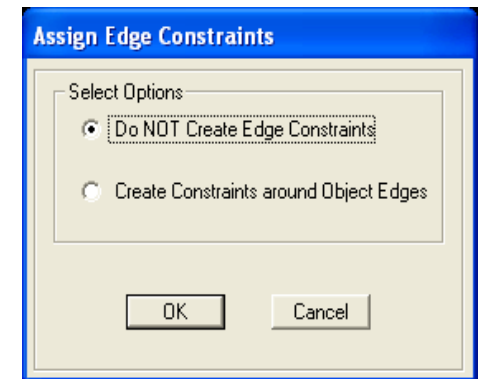


One important issue to note with regards to meshing is the application of the line constraint (called 'edge constraint' in SAP2000). The line constraint "zips" together adjacent elements which do not share common mesh points using a displacement interpolation numerical technique. By default, the line constraint is activated in ETABS, but in SAP2000 the edge constraint must be assigned. In the example below, we have a wall with a mismatched mesh. Assign>Area to activate or deactivate line/edge constraints. If joints of adjacent elements are reasonably close together, use of these constraints can be a huge time saver compared to manually adjusting the meshes, and it can save users from having to model transitions using less accurate triangular elements

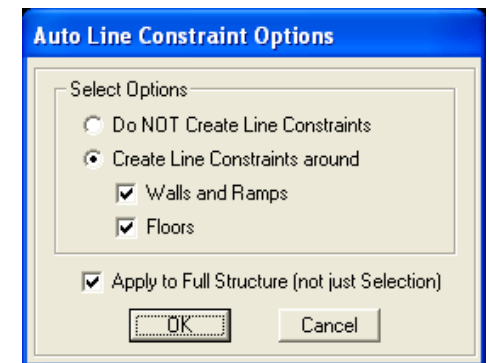
Edge/Line constraint would solve this problem



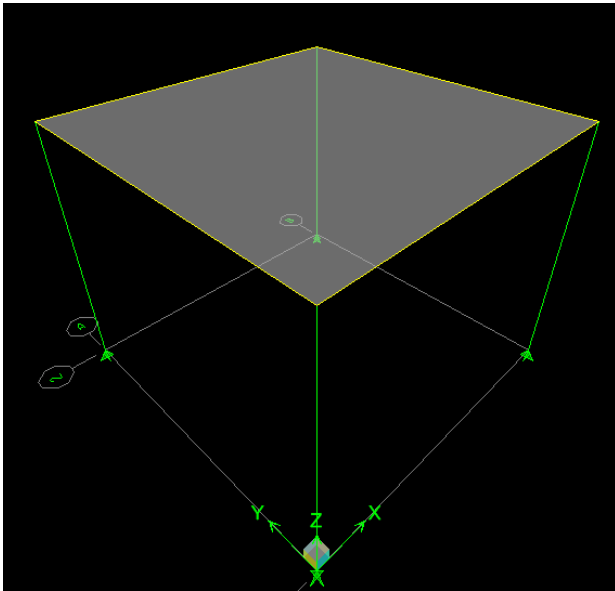
SAP2000 default



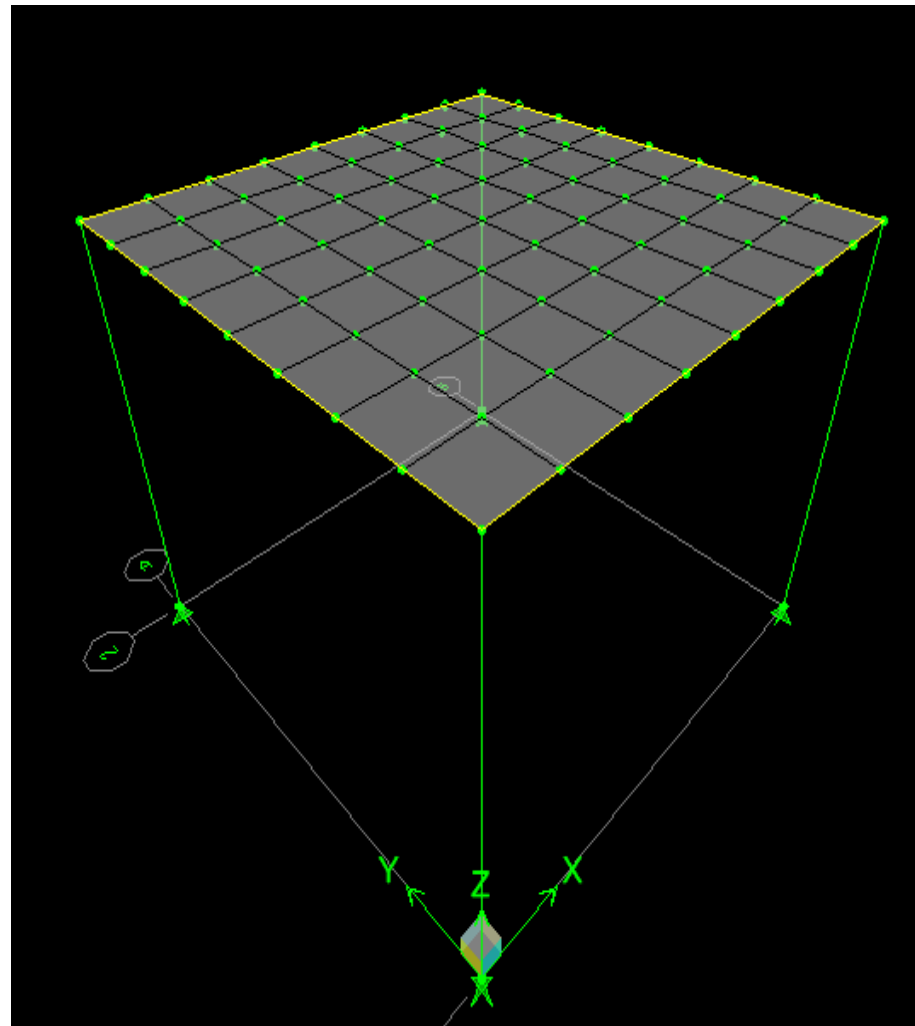
ETABS default



By default, ETABS and SAP2000 automatically add joints to beam/frame elements for connectivity whenever adjacent area elements are meshed

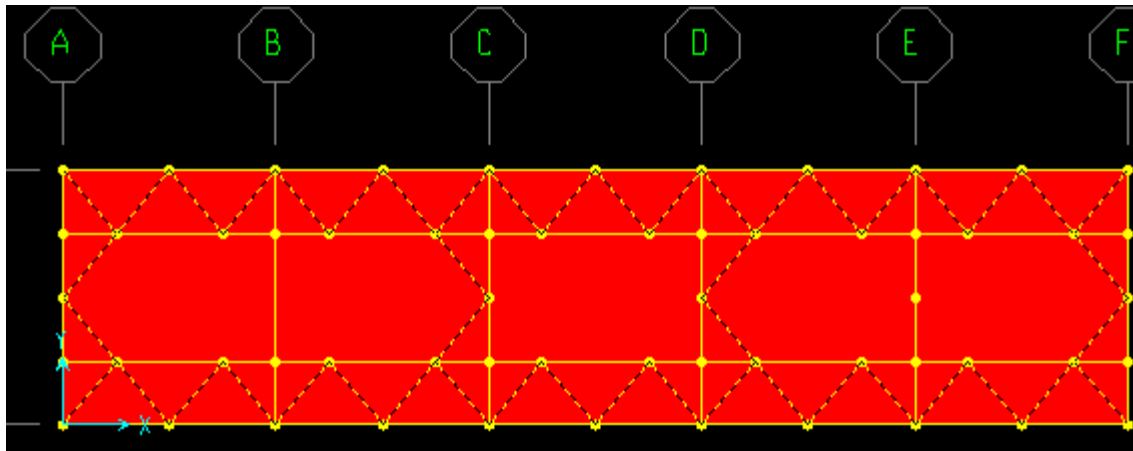


By default, joints are automatically added to adjacent beam elements whenever adjacent area elements are meshed

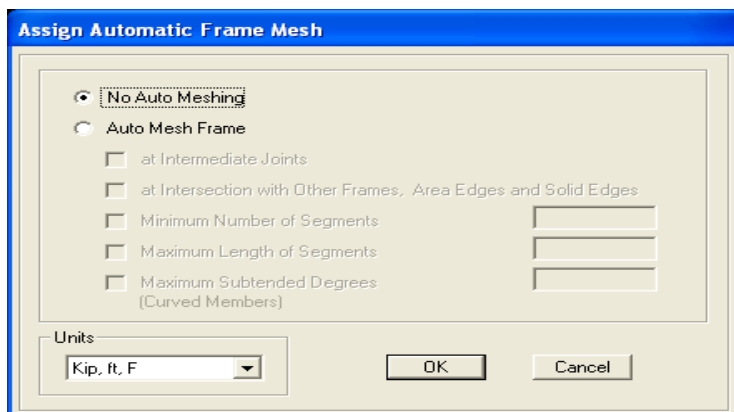


In order to have more control over which beam/frame members are transferred load when an area is meshed, users have the option to de-activate selected frames from taking load from area elements. Select frame elements that you do not want to take load and Assign>Frame to specify no automeshing

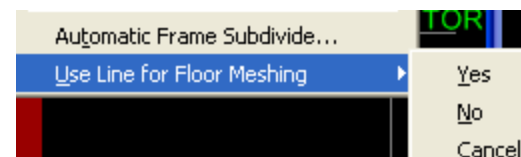
Bracing is selected, then assigned not to automesh with floor



### SAP2000

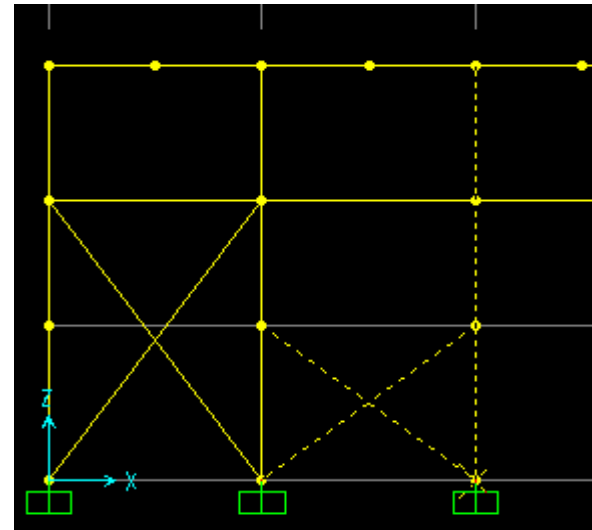
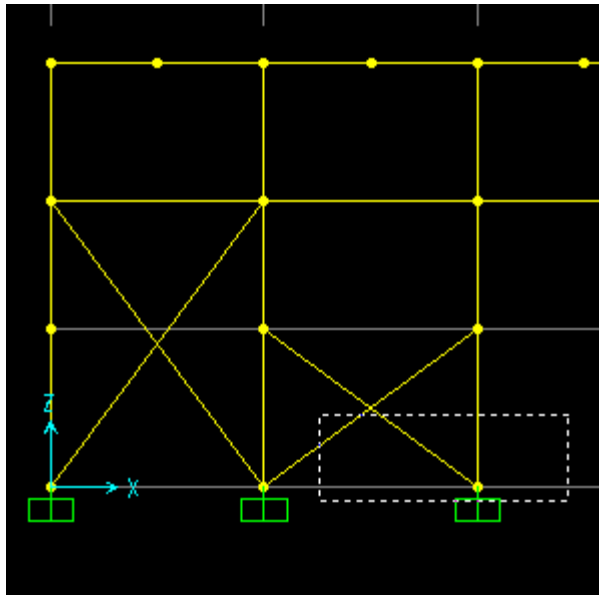


### ETABS

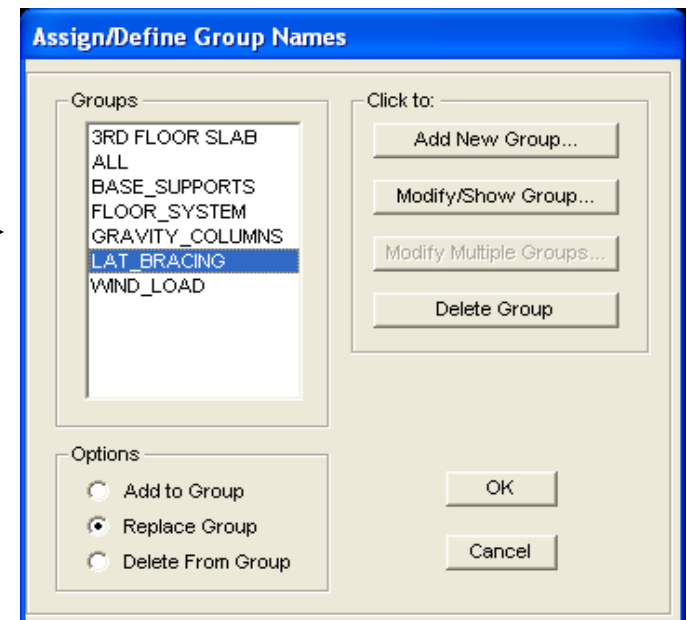


**Select option** menus are currently quite different between ETABS and SAP2000. For that reason, we'll start by reviewing select options used by both programs. Holding down the left mouse key while moving the mouse right-to-left will select every element which is intersected, including those within the window. This enables users to utilize the width of the window to control what is selected

Hold down left mouse key and drag right to left



Assign selected joints, frames, and/or areas to groups. Later you can select by group to view, assign, or tailor output results



Familiarize yourself with these 3 select options on your toolbar. Select all, previous selection (particularly useful when making multiple assignments to a selection), and clear selection. You'll frequently need these options



Select menu options are fairly self-explanatory, except that SAP2000's ability to select by tables opens a number of possibilities. In the following example, we will select only joints  $\geq 40$  ft in the Z (vertical) direction which have load assignments from specified load patterns. First, we use the Select menu>Select>by Coordinate specification as shown.

**Select By Specified Coordinate Range**

Coordinate System: GLOBAL Units: Kip, ft, F

Select All Objects That Are:

- ☒ Entirely Inside The Specified Selection Volume
- ☐ Entirely Or Partially Inside The Specified Selection Volume

Select These Object Types:

- ☒ Point ☒ Line ☒ Area ☒ Solid ☒ Link

X Coordinate Limits:

- ☒ Not Limited
- ☐ Between Two Values
- ☐ Outside Of Two Values
- ☐ At A Single Value

Y Coordinate Limits:

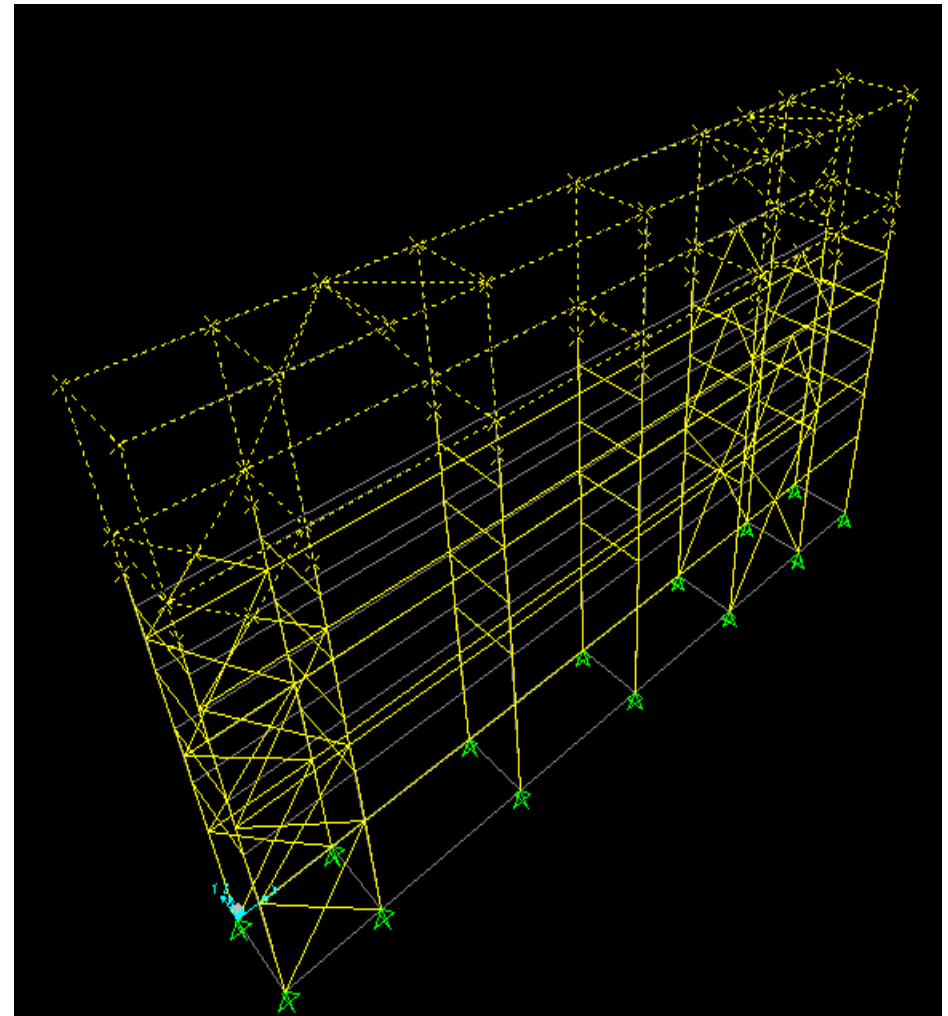
- ☒ Not Limited
- ☐ Between Two Values
- ☐ Outside Of Two Values
- ☐ At A Single Value

Z Coordinate Limits:

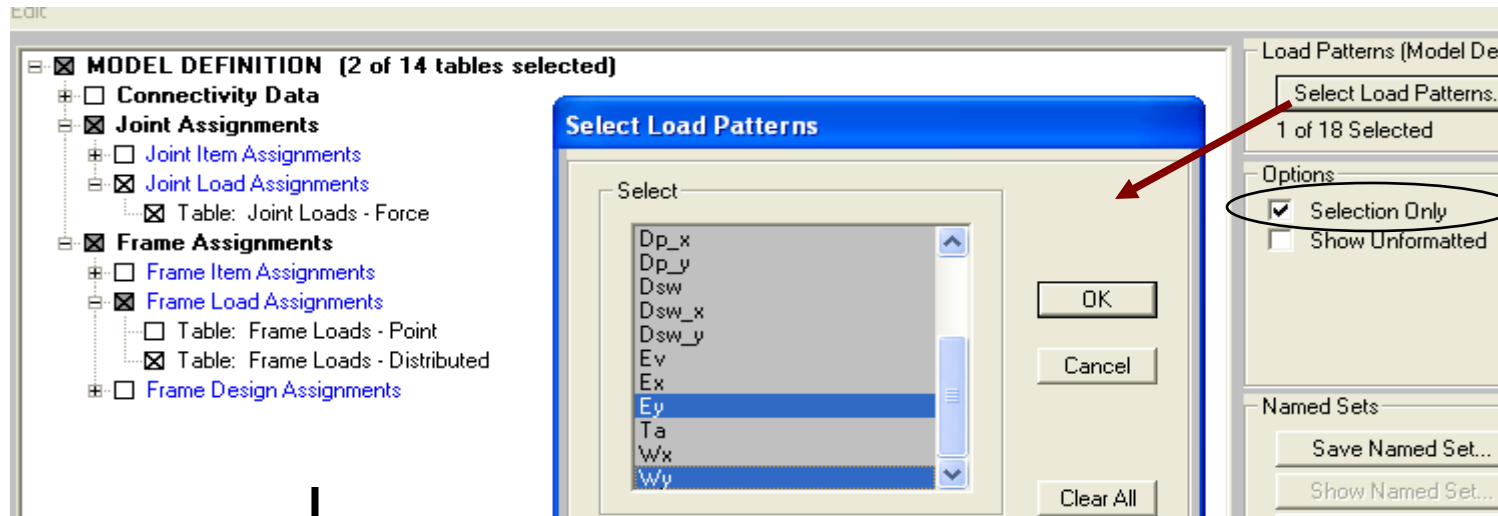
- ☐ Not Limited
- ☐ Between Two Values
- ☐ Outside Of Two Values
- ☒ At A Single Value

Operator:  $\geq$  Value: 40.

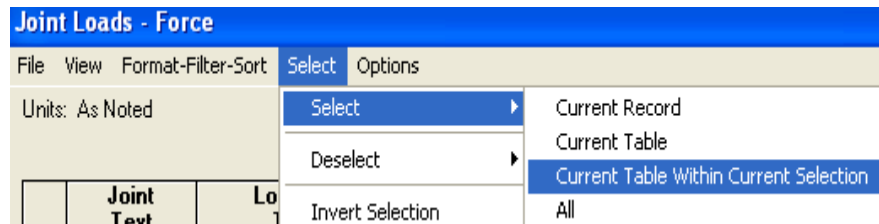
OK Cancel



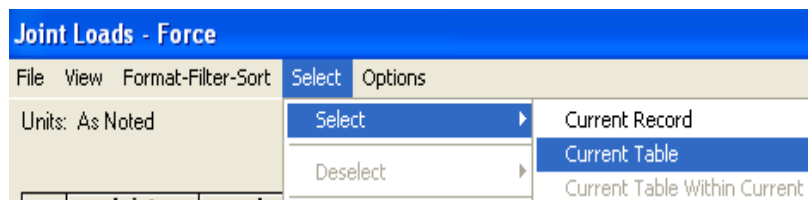
Next, Select using tables to narrow down the selection further by choosing only joints which have load assignments from Ey and Wy load patterns which we highlighted. We'll ignore the distributed loads for this example



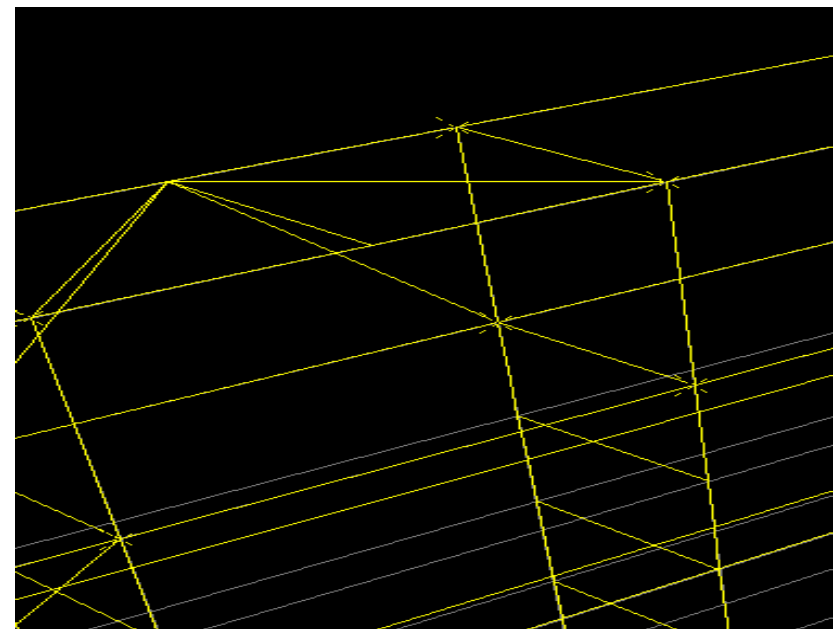
Within Joint loads table, we select Current table within current selection



Next, go back again to the Select menu and select Current table



Joints now selected are only those equal to or above 40 ft which have Ey or Wy load patterns assigned to them



**Working with gridlines** - In SAP2000, you start by defining perimeter grids, then double click any gridline to open the Define grids dialogue in order to add, delete, or modify gridlines and spacing in detail. Grids are quite useful in that they automatically generate planar views, and you can use grids to snap when drawing frames or areas. Also, grids can be added to help with area meshing since SAP2000 and ETABS offer the option to mesh by gridlines. If the “Glue to Grid Lines” option is checked, elements modeled within a gridline that is modified will stretch or shrink with the change in grid spacing.

**Quick Grid Lines**

Cartesian Cylindrical

Coordinate System Name  
GLOBAL

Number of Grid Lines

X direction 4

Y direction 4

Z direction 5

Grid Spacing

X direction 24.

Y direction 24.

Z direction 12.

First Grid Line Location

X direction 0.

Y direction 0.

Z direction 0.

OK Cancel

Here you can add, modify, and delete gridlines

**Define Grid System Data**

Edit Format

System Name GLOBAL Units Kip, ft, F

Grid Lines Quick Start...

X Grid Data

	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Bubble Loc.
1	A	0.	Primary	Show	End	
2	B	24.	Primary	Show	End	
3	C	48.	Primary	Show	End	
4	D	72.	Primary	Show	End	
5						
6						
7						
8						

Y Grid Data

	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Bubble Loc.
1	1	0.	Primary	Show	Start	
2	2	24.	Primary	Show	Start	
3	3	48.	Primary	Show	Start	
4	4	72.	Primary	Show	Start	
5						
6						
7						
8						

Z Grid Data

	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Bubble Loc.
1	Z1	0.	Primary	Show	End	
2	Z2	12.	Primary	Show	End	
3	Z3	24.	Primary	Show	End	
4	Z4	36.	Primary	Show	End	
5	Z5	48.	Primary	Show	End	
6						
7						
8						

Display Grids as  
☒ Ordinates ☐ Spacing

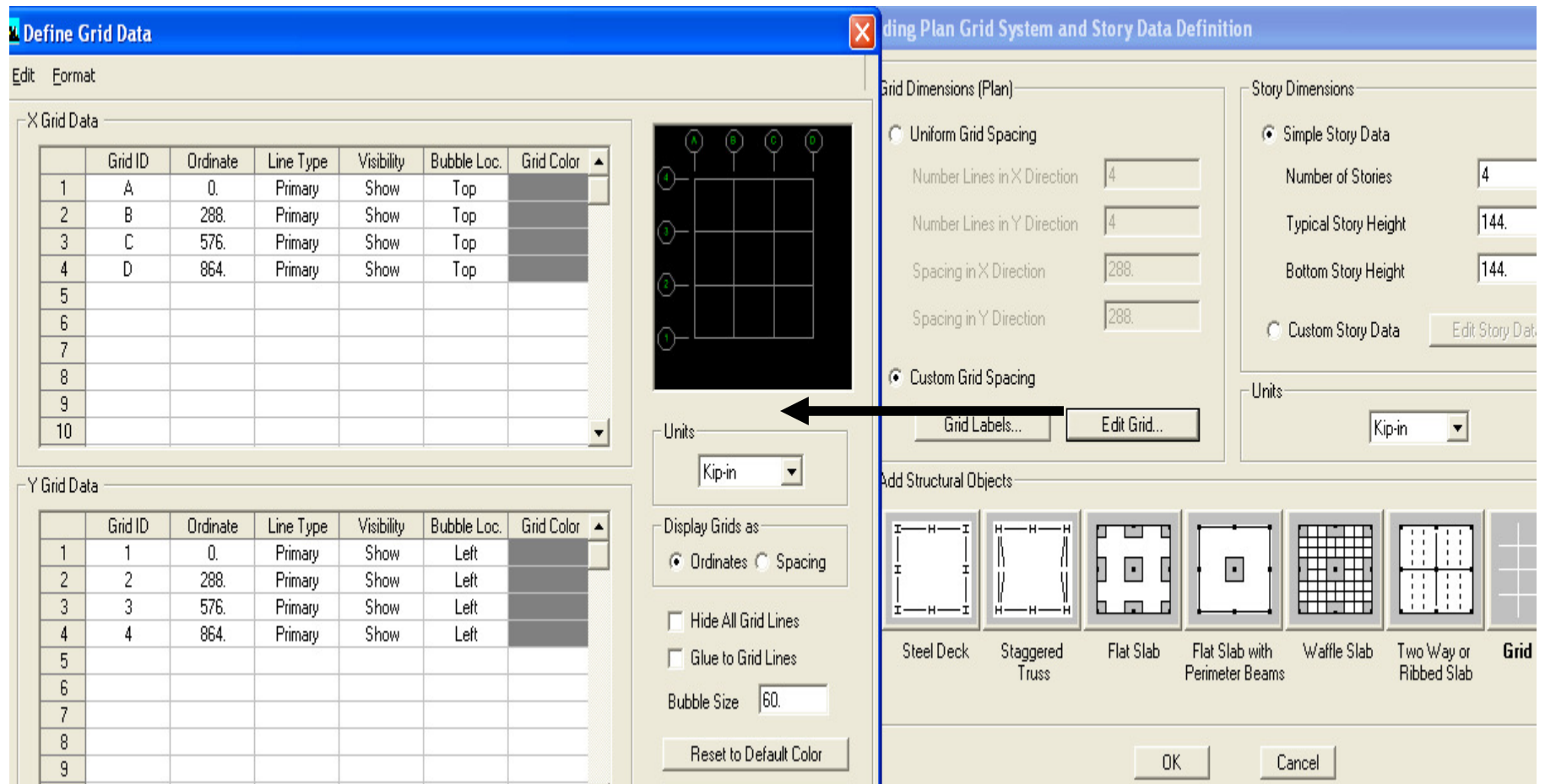
☐ Hide All Grid Lines  
☐ Glue to Grid Lines

Bubble Size 5.

Reset to Default Color  
Reorder Ordinates

OK Cancel

With ETABS, you can define your grids in detail as soon as you start a new model.



Story levels add gridlines at each story elevation. If you want to add gridlines between story levels in ETABS, use Edit>Reference planes and reference lines. With SAP2000, you would simply add a Z gridline to accomplish this.

## Define story levels

[illegible]

**Edit Reference Planes**

Location of Reference Plane (Horiz.)

Z-Ord

7.
7.

Add

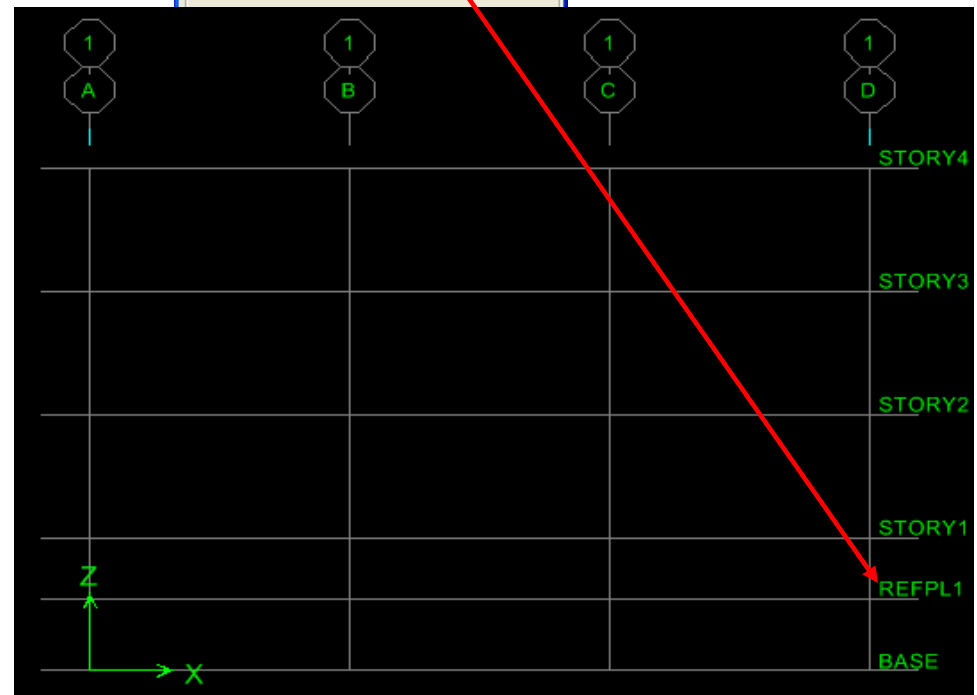
Modify

Delete

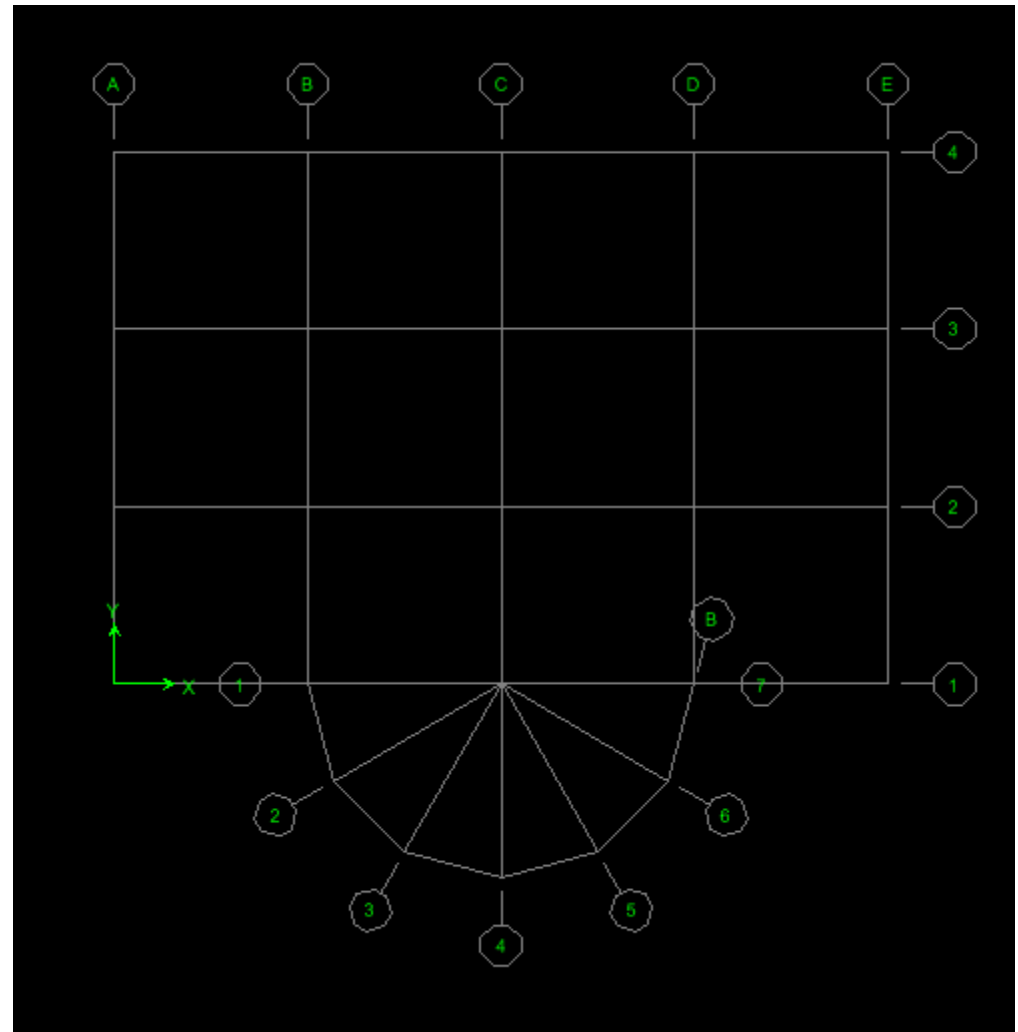
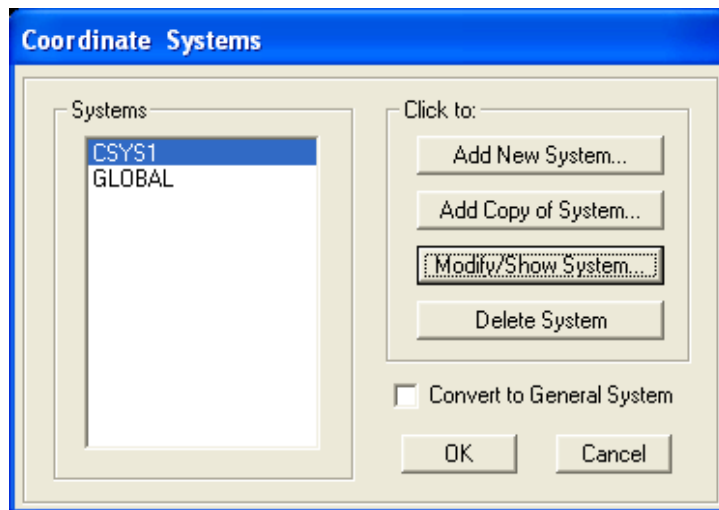
Delete All

Change Units

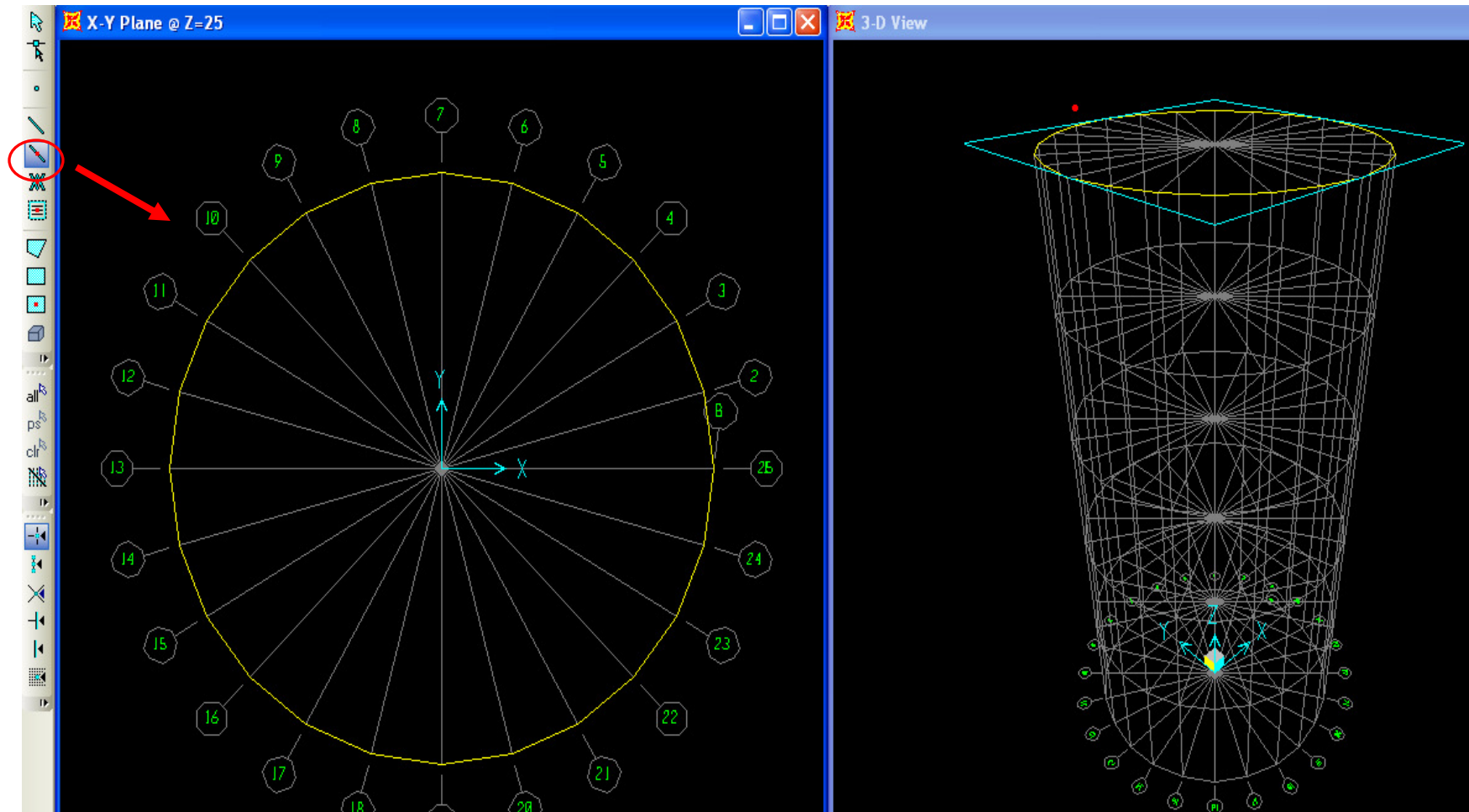
FT




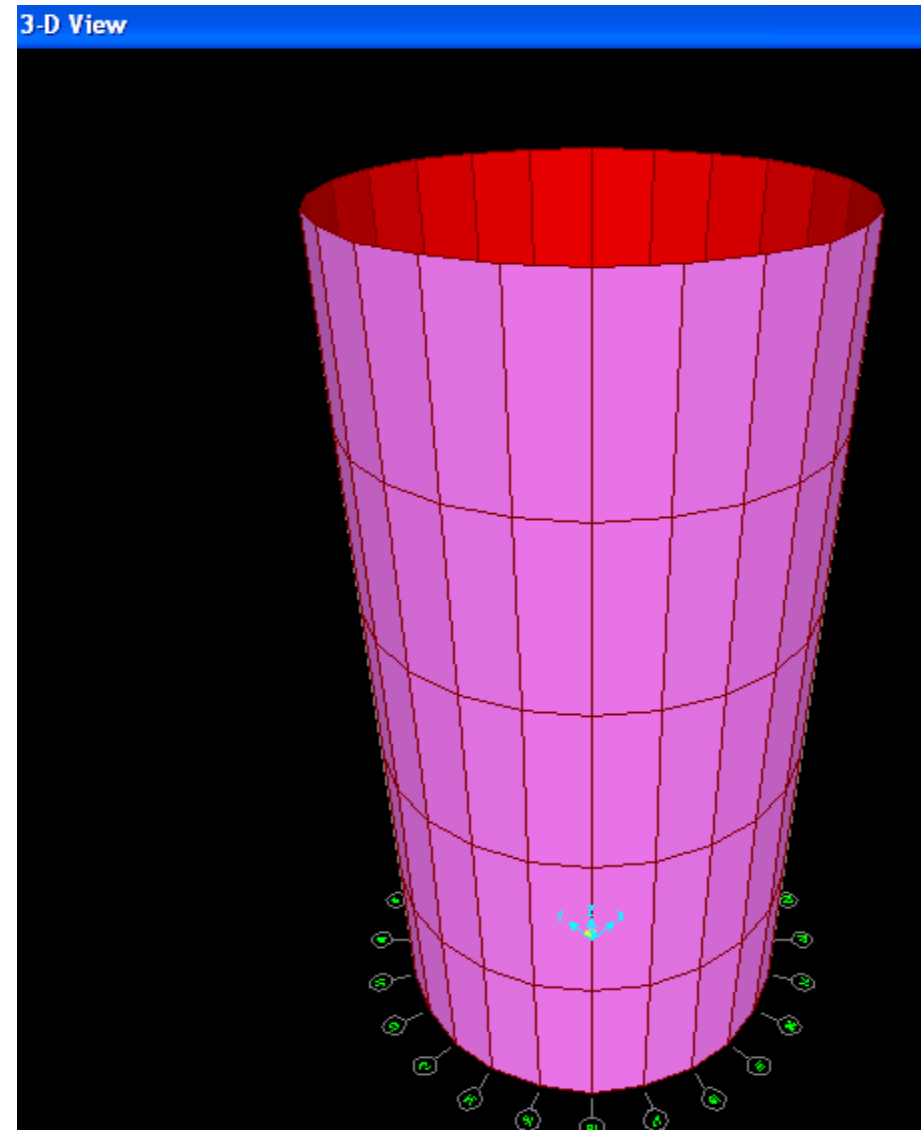
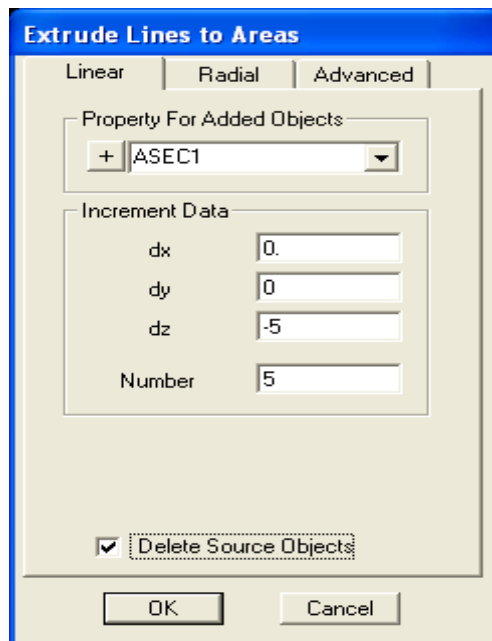
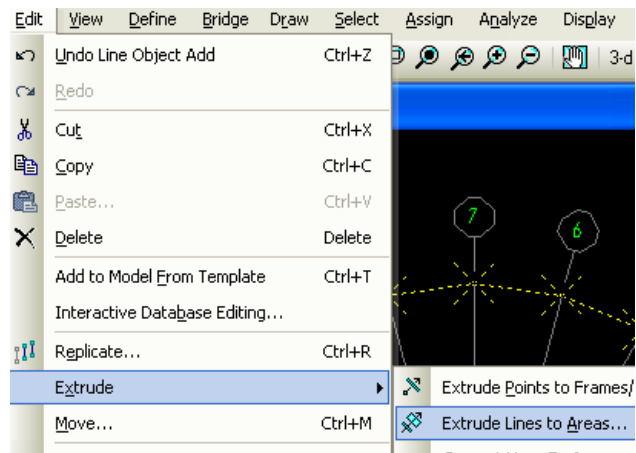
Both ETABS and SAP2000 can have multiple Cartesian and/or cylindrical coordinate systems in one model. The “Convert to General System” option can be used if you need skewed direction gridlines.




This example will demonstrate the extrude feature as a modeling tool, as well as demonstrate a useful cylindrical grid system as shown below, then use the quick-draw frames to add frame elements along the perimeter (section property doesn't matter) at the top level.

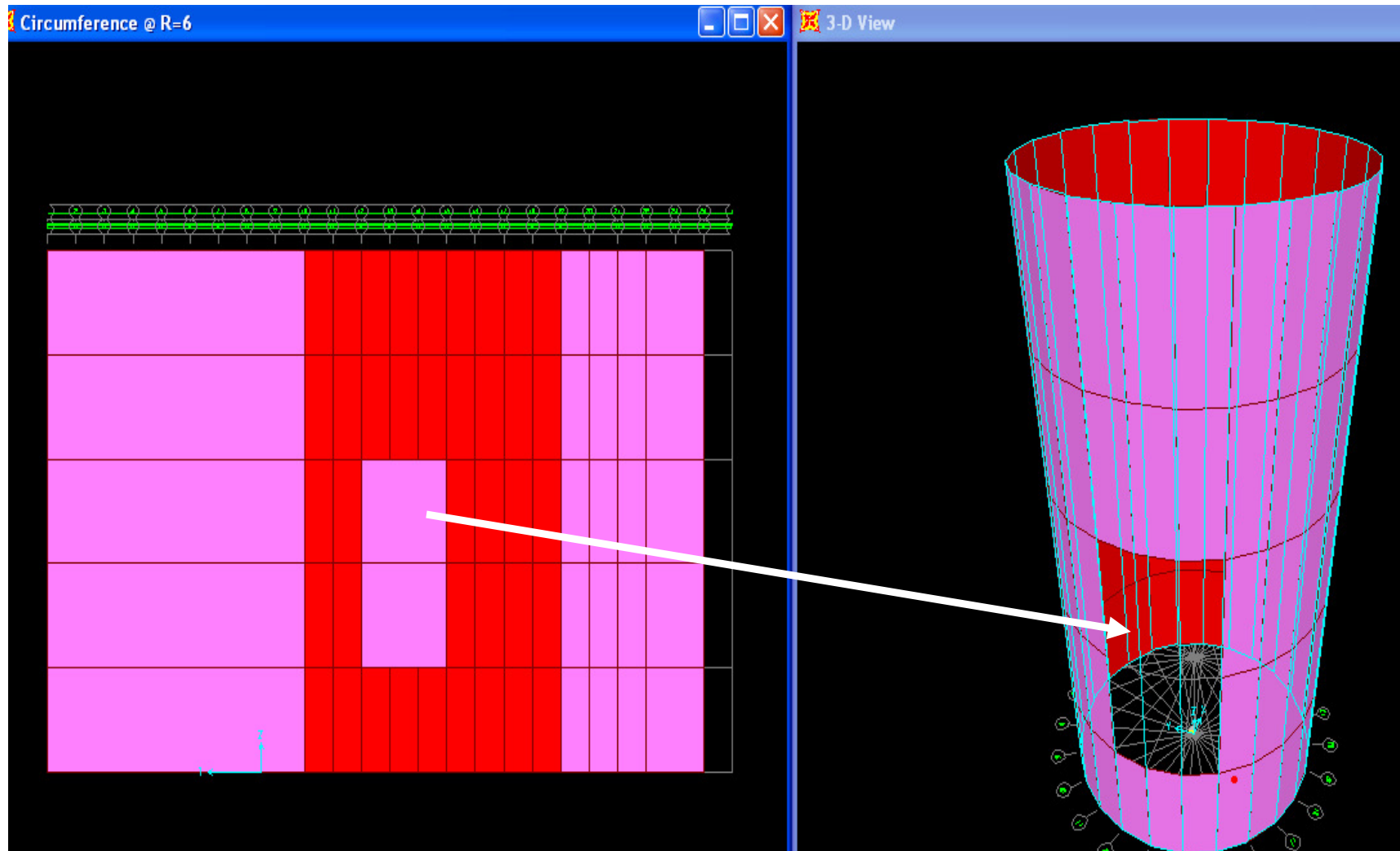


Next, click the 'select all' button  to select frame elements, then Edit>Extrude>Extrude lines to areas. In this example we extrude downward in the -Z direction 5 X 5 ft, deleting the dummy frames source object. This extrusion technique enables us to quickly model a cylinder made of area finite elements

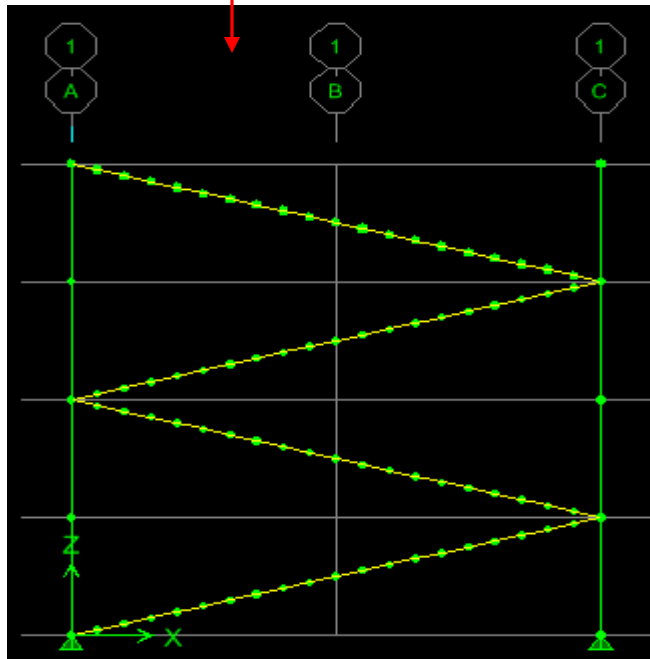
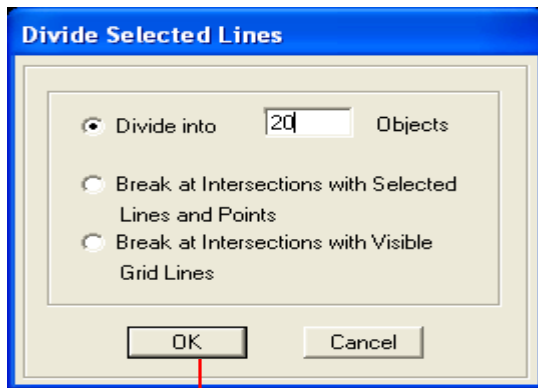




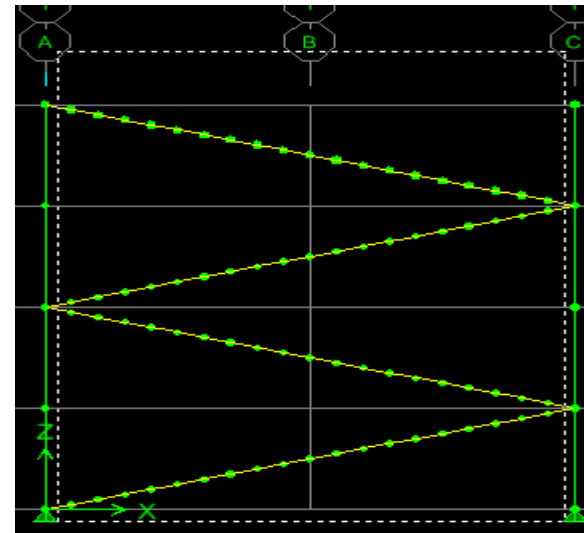
By clicking the tz button  you can unwrap the cylinder for editing as shown below left. In this example, we selected some areas and pressed the Del key on the keyboard to delete them in order to create an opening. If the tz button disappears, use the yz button to unwrap. ETABS does not offer this unwrap feature at this time.



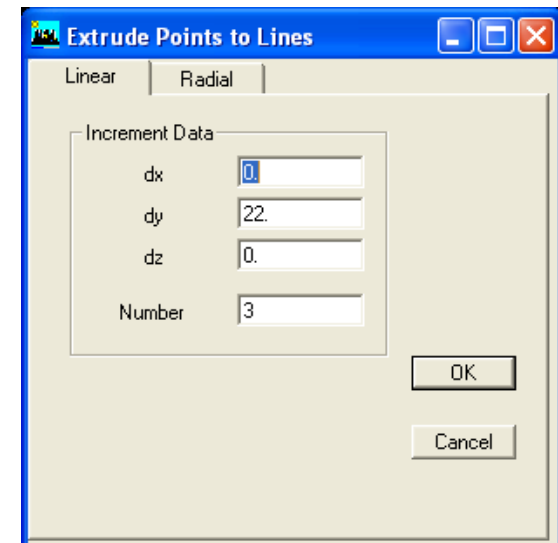
We're going to explore another option for modeling with extrude features which involves extruding points to lines. This modeling technique can be used to draw beams, columns, or braces, but it has a particularly useful application in drawing beams on a slope. In this example we divide selected beams, then select the newly added joints and Edit>Extrude points to lines (in SAP2000 Edit>Extrude>Extrude points to frame/cables).



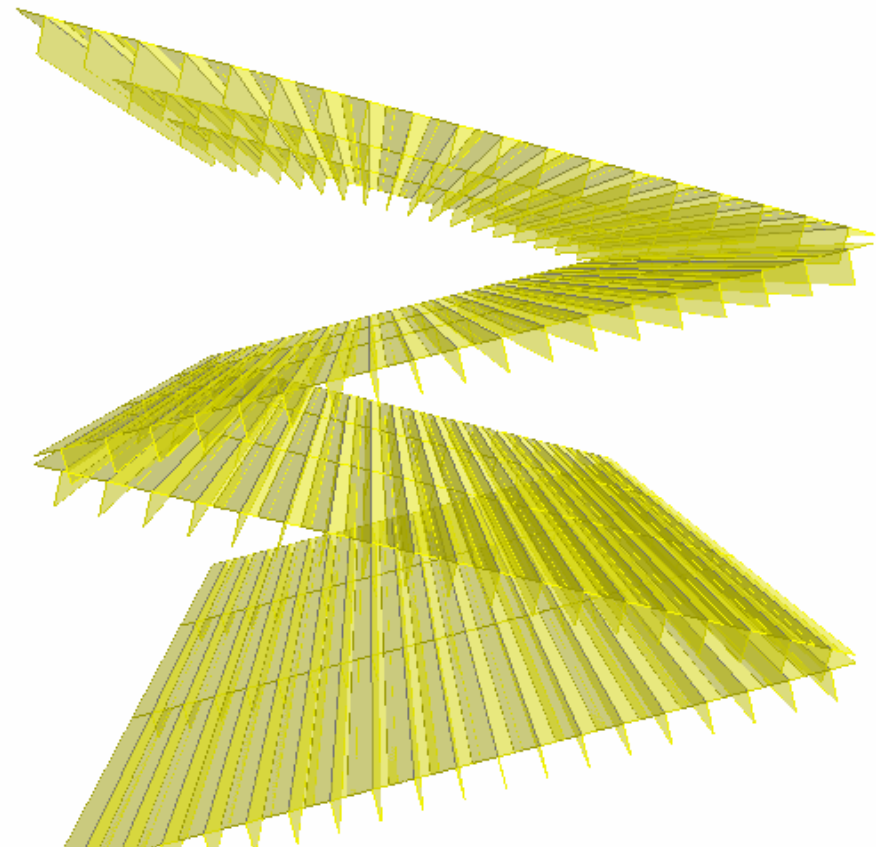
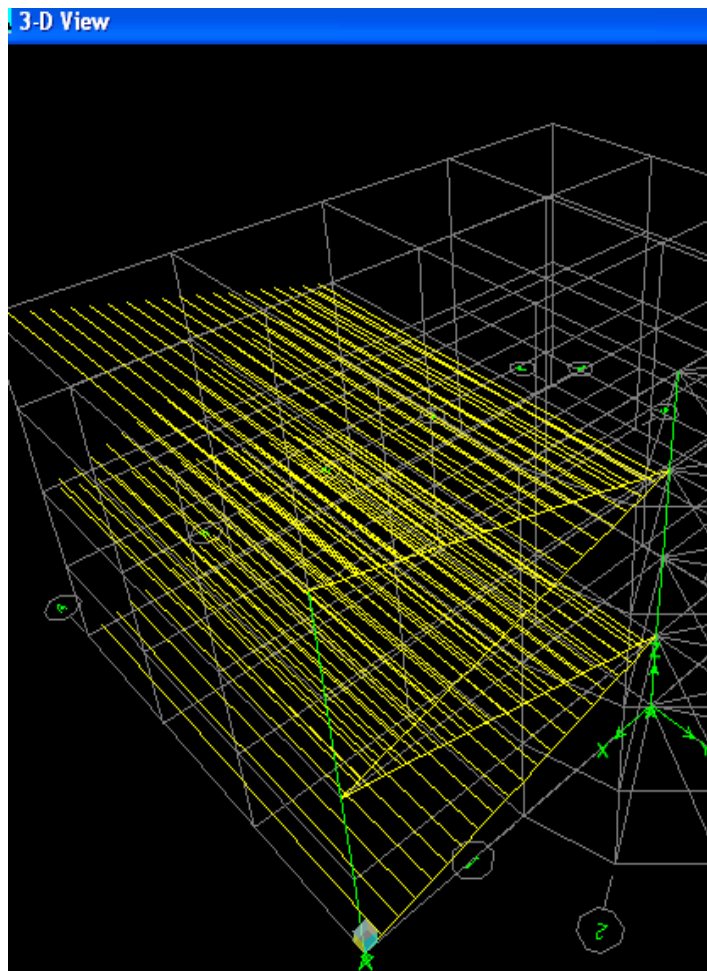
Window  
around  
joints to  
select



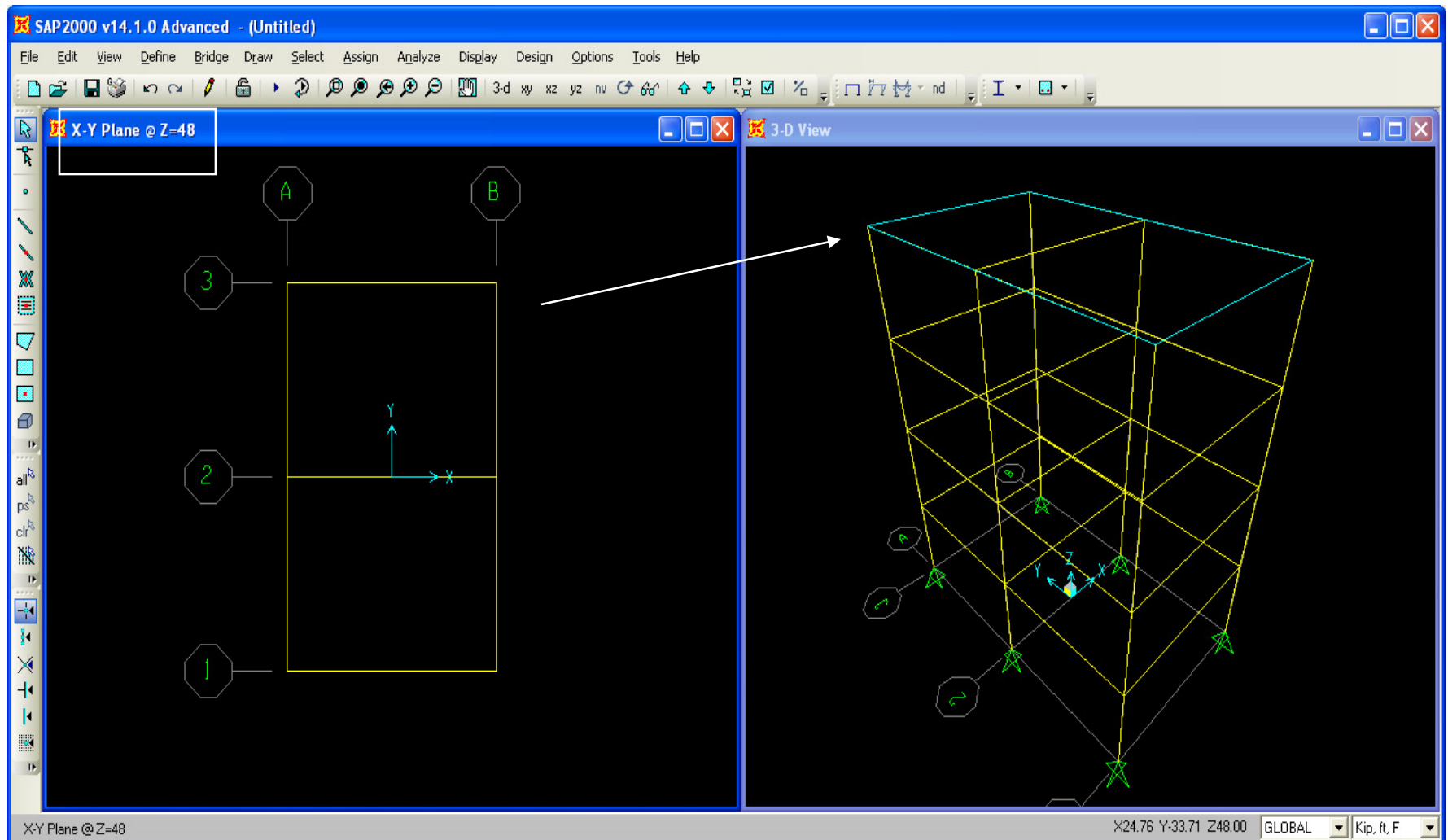
SAP2000 offers  
the option to  
choose the  
section which is  
extruded.  
ETABS does  
not offer that  
option at this  
time



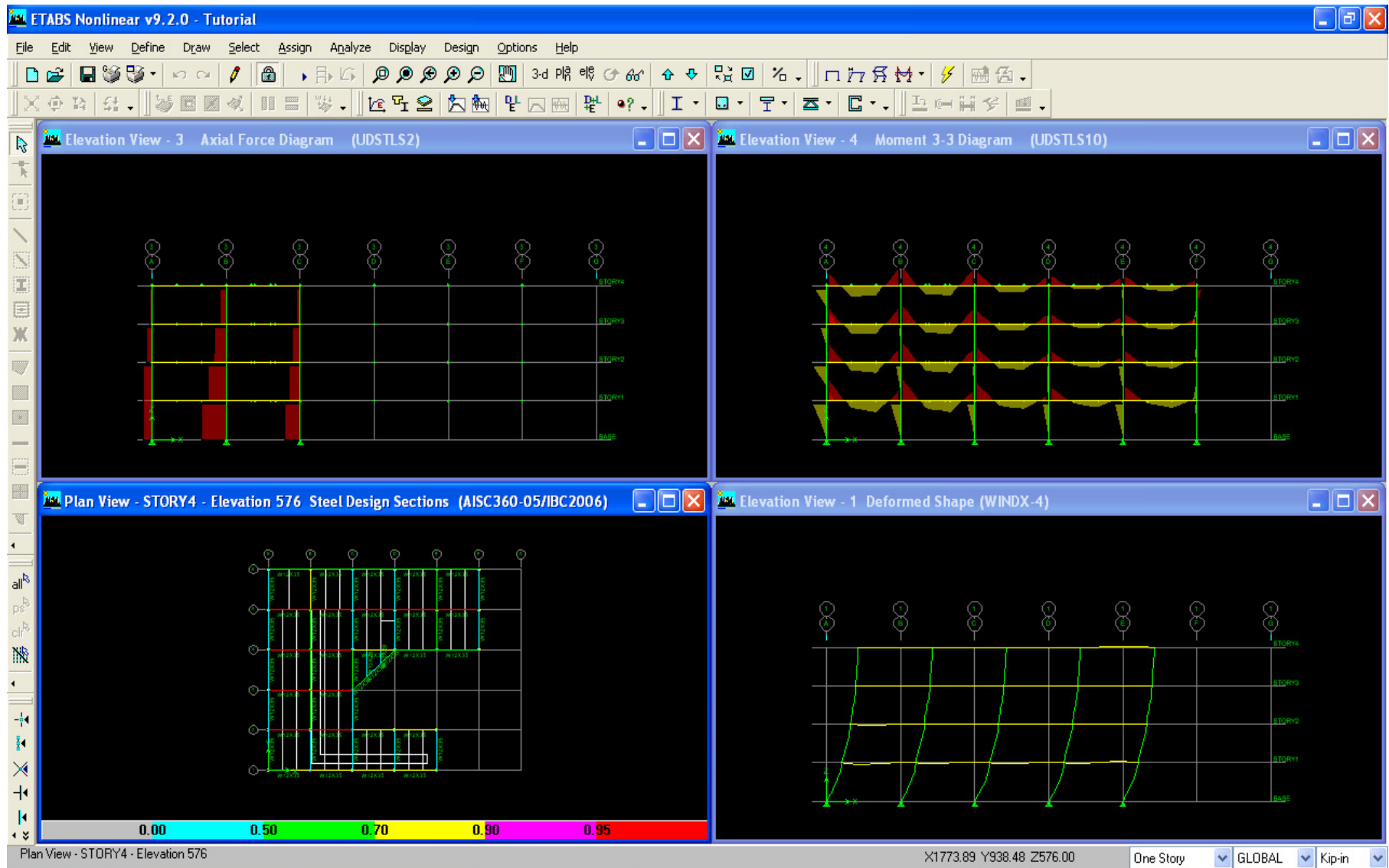
Next after extruding our beam elements, use select options to select the extruded beams in order to rotate local axes and/or assign section. Use arctangent of the slope to determine angle, then Assign>Frame>Local axes and input angle. In this example, the sign of the angle changes at each level from positive to negative because of the different slopes. When working with concrete Tee sections, one peculiar aspect to defining the section is that you must first add from the steel section type and click Tee. Simply change the default material from steel to concrete and input Tee section dimensions.



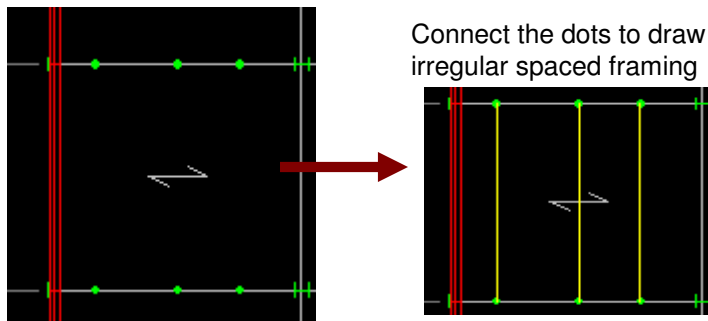
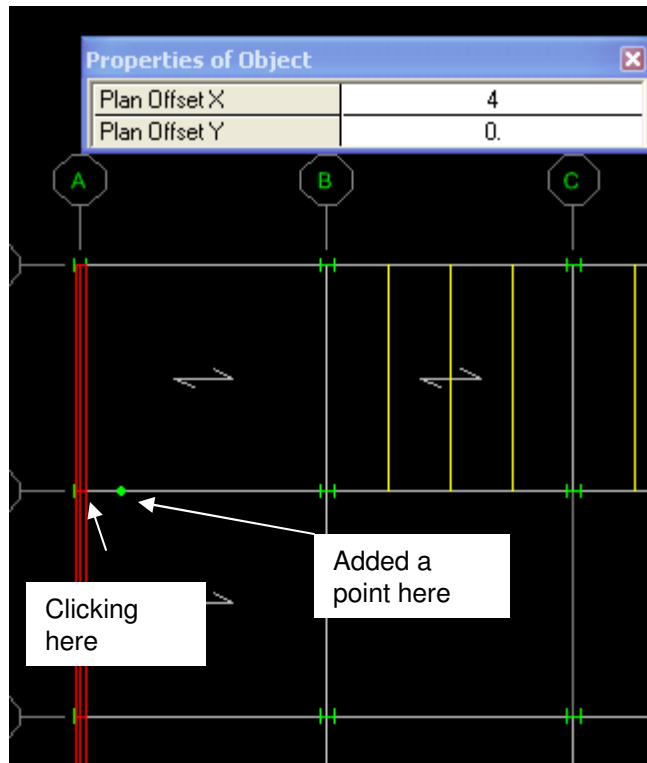
Whenever a planar view is the active window, it's displayed as a blue perimeter outline in the 3D model. In this manner, the windows are linked. In this screenshot, it's an XY plan view at the top level, but using the arrow keys and xy xz yz buttons, users can switch views and scroll through the model.



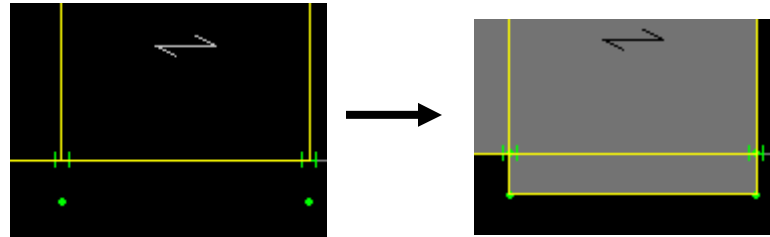
Users can work with up to 4 windows in either ETABS or SAP2000. When drawing sloped beams or ramps, users can begin drawing in one planar view window then complete drawing the element in another window which has a different planar location. When drawing ramps or sloped beams, it helps to work with 3 windows – 1 large 3D view with the other 2 windows in XY plan view but at different elevations.



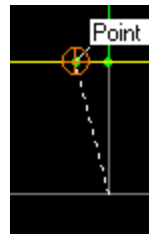
**‘Draw Point objects’ option, aka ‘Draw special joint’** in SAP2000, is a tool which enables users to draw joints offset a user specified distance from wherever they mouse click. It’s a useful tool for modeling irregular framing, or for assigning joint load. When drawing point objects on an existing frame element, ETABS and SAP2000 will automatically mesh that element internally in order to consider loads, but then reformulate the element to report results and design it using its original length with which it was originally drawn.



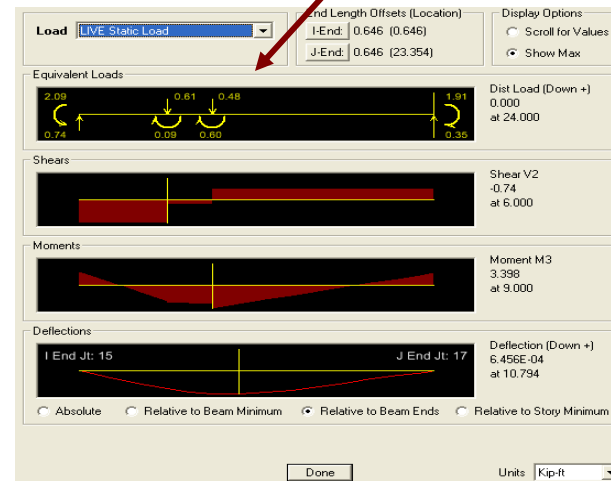
Use point objects to model cantilevered structures



Connect the dots to draw sloped columns or beams



Use it to assign joint loads on existing frame elements



ETABS and SAP2000 automatically mesh this beam to account for joint loads, but report the beam as 1 moment diagram instead of dividing it into multiple smaller pieces