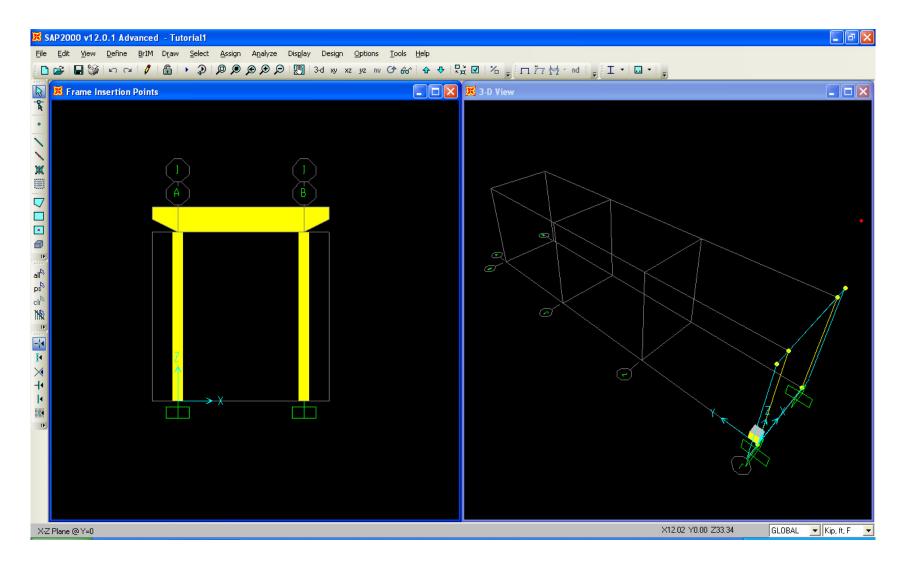
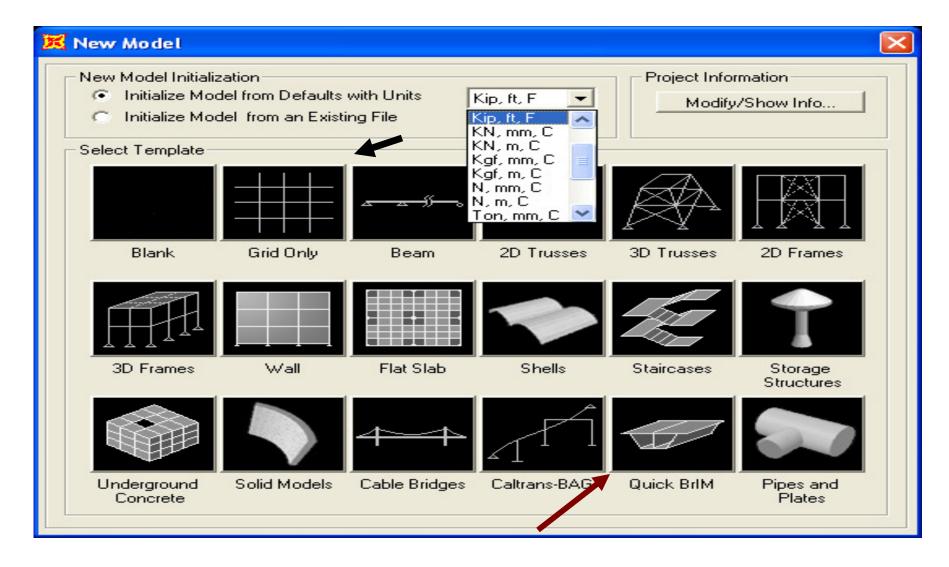
## SAP2000 Getting Started Tutorial

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



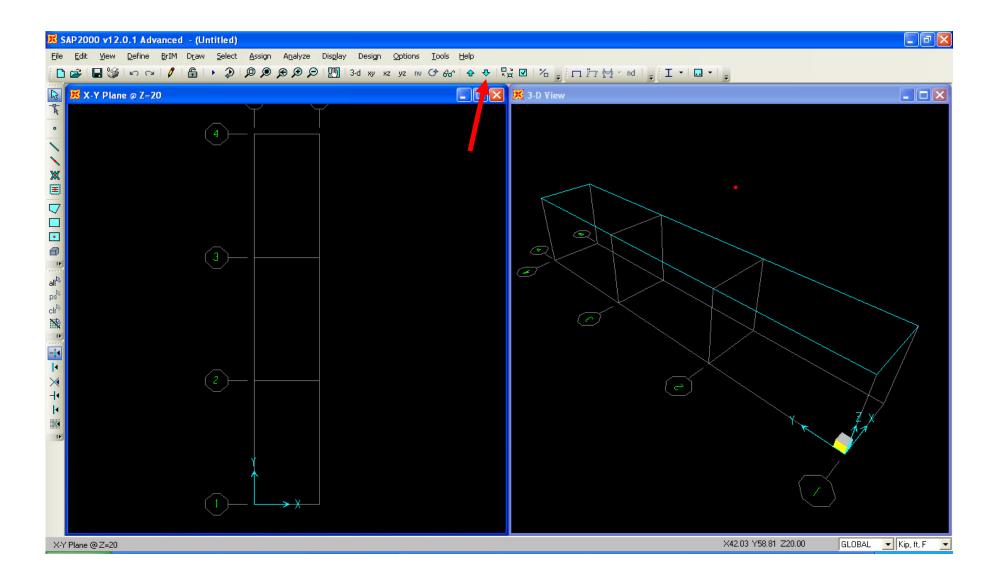
Click File>New menu or button/icon 🗈 to bring up the initial SAP2000 screen where you select units and choose whether you want to work with grids or one of the templates. For this tutorial, we will work with Grids, but you will probably want to explore the Quick BrIM options for modeling of simple bridges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



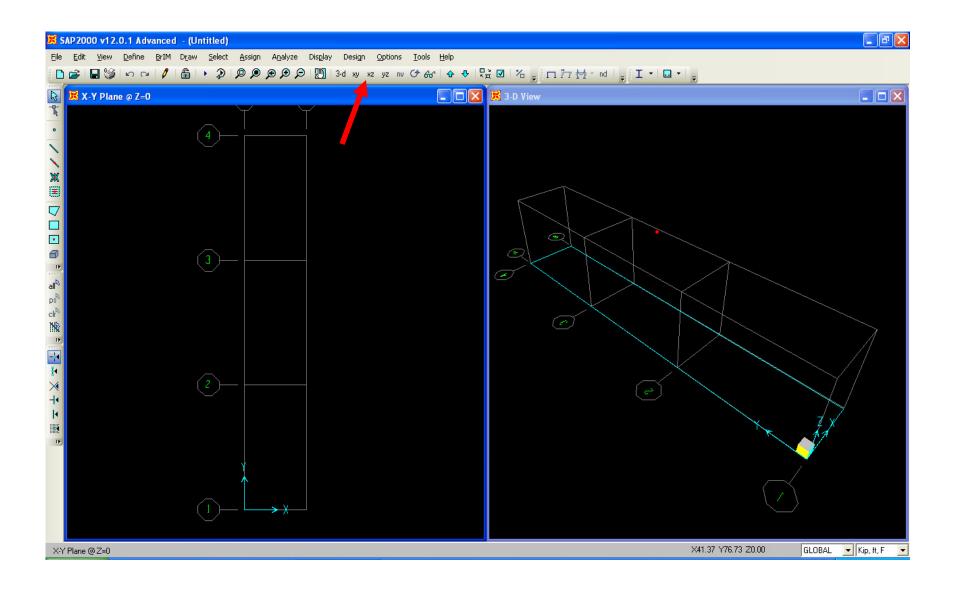
This screen enables you to define equally spaced grids. Later you will learn how to modify and insert irregular spaced gridlines. Please type Number of Grid Lines and Grid Spacing as shown. Note how although we are in Kip-ft units, you can input with architectural units (180" below) at anytime and SAP2000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

Quick Grid Lines					
Cartesian	Cartesian Cylindrical				
Coordinate System	Coordinate System Name				
GLOBAL					
Number of Grid Line	28				
× direction	2				
Y direction	4				
Z direction	2				
- Grid Spacing	Grid Spacing				
× direction	180'				
Y direction	30				
Z direction	20				
First Grid Line Local	tion				
× direction	0.				
Y direction	0.				
Z direction	0.				
ОК	Cancel				

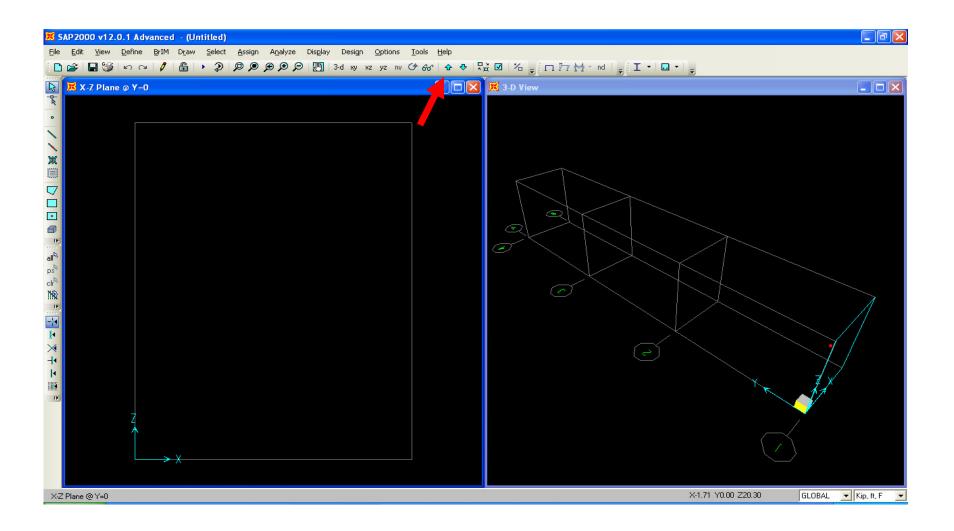
Below you will see the default split-screen view. Note that the planar view on the left (X-Y plan view) is highlighted in blue on the 3D model on the right. As an exercise, click the down arrow as shown below



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



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



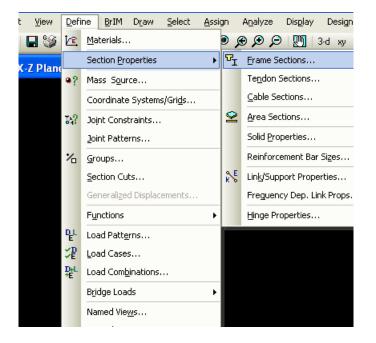
Double click any gridline to display the Define grid system data dialogue. Here you can add, modify or delete gridlines at any time. Grids can be added out of numerical sequence and SAP2000 will automatically reorder them. Select rows and use Edit menu to delete. The 'Glue to Grid Lines' option, when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

					Units		Grid Lines
System	Name	GI	LOBAL		Kip,	ft, F 💌	Quick Start
Grid Da	ta						
	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Grid Color 🔺	
1	А	0.	Primary	Show	End		
2	В	15.	Primary	Show	End		
3							
4							
5							
6							
7							
8							
' Grid Da	ta						Display Grids as
	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Grid Color 🔺	Ordinates C Spacing
1	1	0.	Primary	Show	Start		
2	2	24.	Primary	Show	Start		
3	3	48.	Primary	Show	Start		Hide All Grid Lines
4	4	72.	Primary	Show	Start		Glue to Grid Lines
5							
6							5 JUL 01 175
7							Bubble Size 4.75
8						-	
Grid Da	ta						
	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.		Reset to Default Color
1	Z1	0.	Primary	Show	End		
2	 Z2	20.	Primary	Show	End		Reorder Ordinates
3							
4							
5							
6							
7							Cancel

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

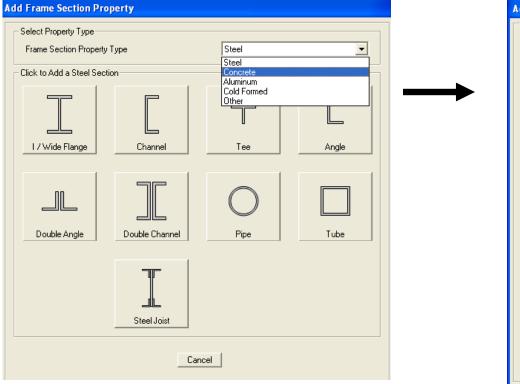
Define Materials	
Quick Material Definition	Click to: Add New Material Quick Add New Material Add Copy of Material
Material Type Concrete	Modify/Show Material Delete Material
Specification <u>(*c 5000 psi</u>	Cancel

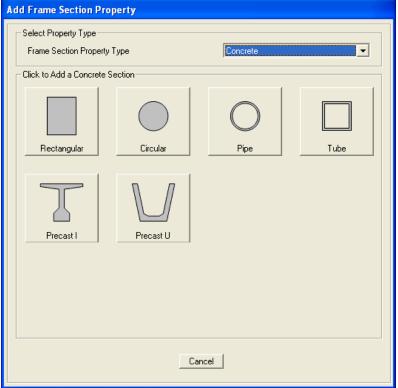
Next we will define frame sections. Go to Define>Section properties>Frame Sections. In this dialogue, you would click 'Import New Property' only if you want to import standard steel sections from a library (AISC and international libraries of sections). For nonstandard steel sections such as Plate Girders,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



Properties Find this property:	Click to:
	Add New Property
	Add Copy of Property
	Modify/Show Property
	Delete Property

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





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

Rectangular Section		
Section Name	3x3	
Section Notes		Modify/Show Notes
Properties	Property Modifiers	Material
Section Properties	Set Modifiers	+ 5000Psi 💌
Dimensions		
Depth (t3)	3.	
Width (t2)	3.	3.
		Display Color
Concrete Reinforceme		ancel
L	OK CA	

einforcement Data			
Rebar Material			
Longitudinal Bars	+ A615Gr60	•	
Confinement Bars (Ties)	+ A615Gr60	•	
Design Type			
Column (P-M2-M3 Design)	)		
🔿 Beam (M3 Design Only)			•
Reinforcement Configuration	- Confinement	You can ty	pe 3″
Rectangular	Ties	and tab to	
C Circular	C Spiral	change co	ver
Clear Cover for Confinement B Number of Longit Bars Along 3 Number of Longit Bars Along 2 Longitudinal Bar Size Confinement Bars Confinement Bars Size Longitudinal Spacing of Confir Number of Confinement Bars in Number of Confinement Bars in Check/Design C Reinforcement to be Chec	ars 3" B-dir Face 3 2-dir Face 3 + #8 + #4 hement Bars 0.5 n 3-dir 3 n 2-dir 3		

Commonly used to apply reduction factors to El for cracked sections

Frame Property/Stiffness Modification Factors				
Property/Stiffness Modifiers for Analysis				
Cross-section (axial) Area				
Shear Area in 2 direction				
Shear Area in 3 direction				
Torsional Constant				
Moment of Inertia about 2 axis				
Moment of Inertia about 3 axis				
Mass 1				
Weight 1				
OK Cancel				

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

Display Color

•

Frame Properties	Rectangular Section
Properties     Find this property:     Import New Property      Add New Property      Add Copy of Property      Modify/Show Property      Delete Property	Section Name     3x1CapBM       Section Notes     Modify/Show Notes       Properties     Property Modifiers       Section Properties     Set Modifiers       Dimensions     + 4000Psi       Depth (t3)     3       Width (t2)     1       Display Cok
OK Cancel	Concrete Reinforcement

Reinforcement Data				
Rebar Material				
Longitudinal Bars	+ A615Gr60 -			
Confinement Bars (Ties)	+ A615Gr60 -			
Design Type				
C Column (P-M2-M3 Design)				
Beam (M3 Design Only)	←			
Concrete Cover to Longitudinal	Rebar Center			
Тор	0.2083			
Bottom	0.2083			
Reinforcement Overrides for Ductile Beams				
	eft Right			
Top 0.	0.			
Bottom 0.	0.			
	J			
ОК	OK Cancel			

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

Rectangular Section				
Section Name	3x1Cap	ъВМ		
Section Notes		Modify/Show Notes		
Properties Section Properties	Property Modifiers	Material + 4000Psi •		
Dimensions				
Depth (t3)	3			
Width (12)	1	3		
Display Color Display Color OK Cancel				

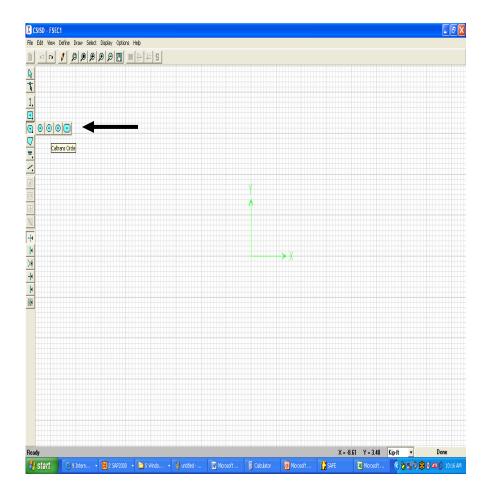
Rectangular Section				
Section Name		1.5X1Ca	рВМ	
Section Notes			Modify/Show Notes	
Properties Section Properties	Property Mod		Material + 4000Psi	•
Dimensions				
Depth (t3)	1.5			
Width (t2)	1.		3<	
			Display Color	
Concrete Reinforceme	ent			
	OK	Car	ncel	

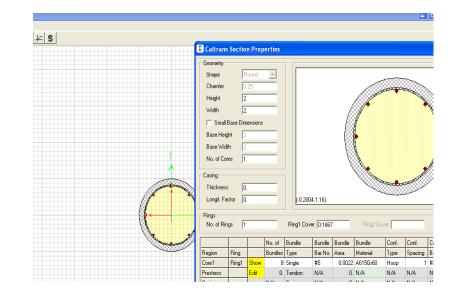
Next, to review SAP2000's section designer, click 'Add new property' again with Frame section property type 'Other', click Section Designer, then in the SD Section data dialogue click the 'Section Designer' button

SD Section Data

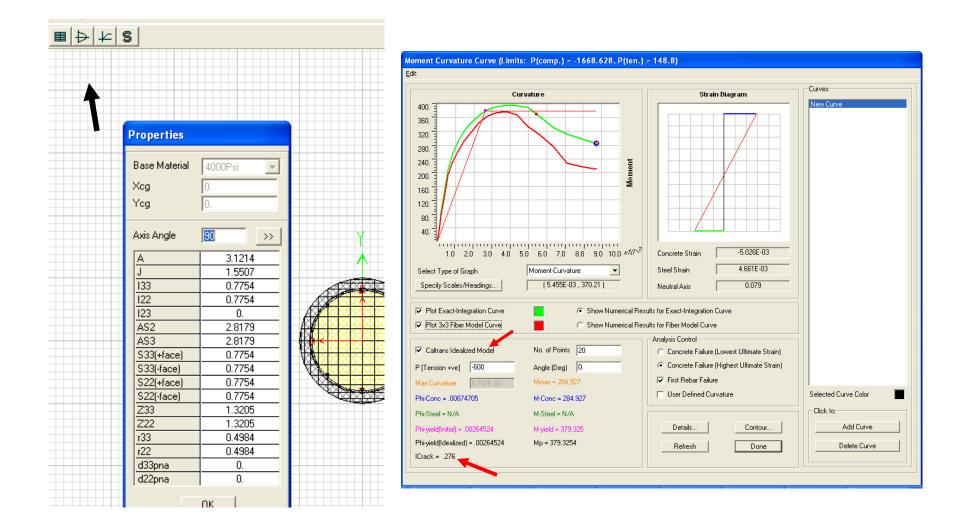
Add Frame Section Property Select Property Type	Section Name FSEC1 Section Notes Modify/Show Notes
Frame Section Property Type	Base Material + 4000Psi  Design Type  No Check/Design  General Steel Section  Concrete Column  Concrete Column Check/Design  Reinforcement to be Checked  Reinforcement to be Designed
Cancel	Define/Edit/Show Section Section Designer Section Properties Properties Display Color

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





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

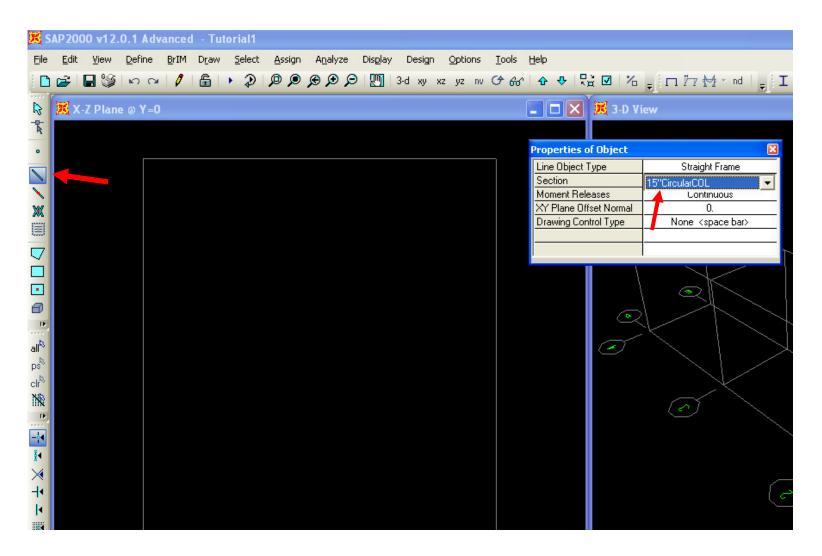


Click 'Add new property' button once more, but this time click Concrete 'Circular' type, name it 15"CircularCOL, type 15" and then press tab to have SAP2000 convert to current ft. units. Click OK twice to accept and save defined sections. Click the save button and name the model *Tutorial1* 

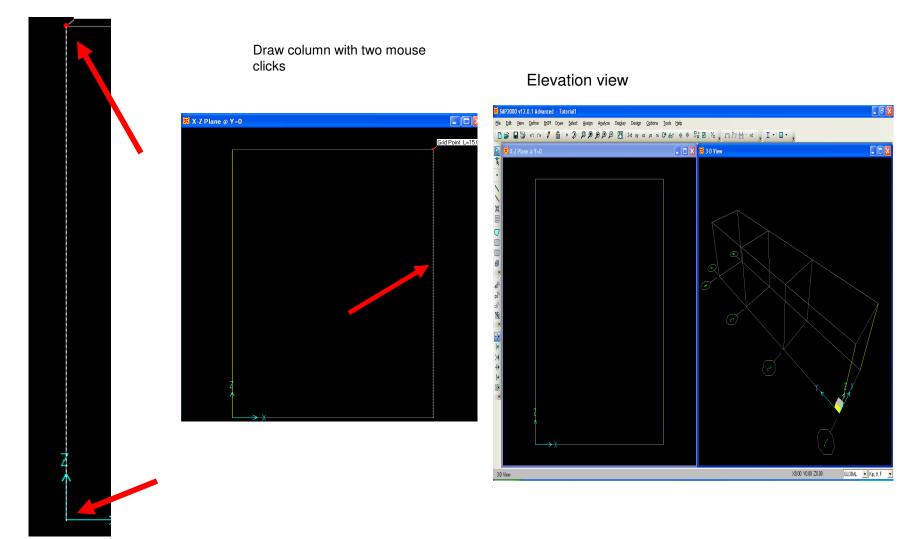
Frame Properties	
Properties Find this property: 1.5x1CapBM	Click to: Import New Property
1.5X1CapBM 3x1CapBM 3x3	Add New Property Add Copy of Property Modify/Show Property
	Delete Property
DK	Cancel

Section Name	15"Circ	ularCOL
Section Notes		Modify/Show Notes
Properties Section Properties	Property Modifiers Set Modifiers	Material + 4000Psi
Dimensions Diameter (t3)	15'	
Concrete Reinforcemer	ıt	Display Color

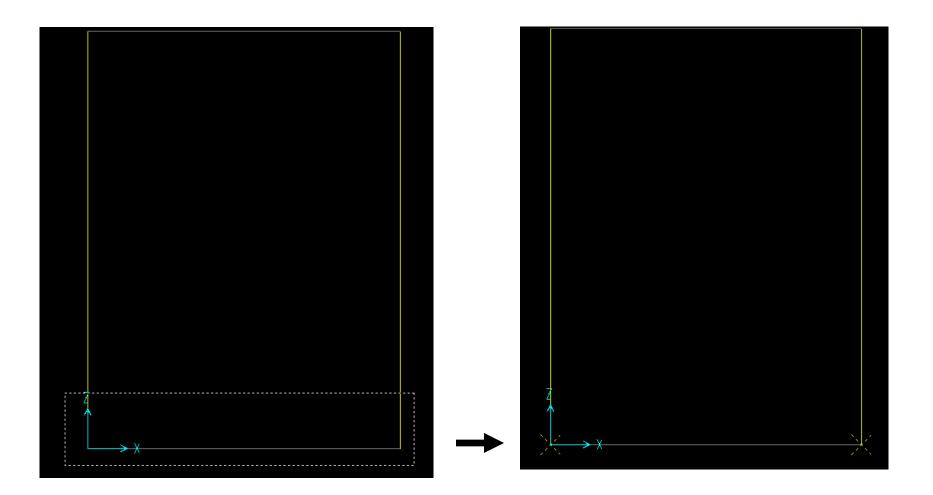
You should still have a split screen view as shown below with X-Z planar view on the left. If not, click your left window to make it active, then click the xz button to change view and up/down arrow keys if you need to adjust further. Next, click 'Draw frame/cable element' button as shown and click the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



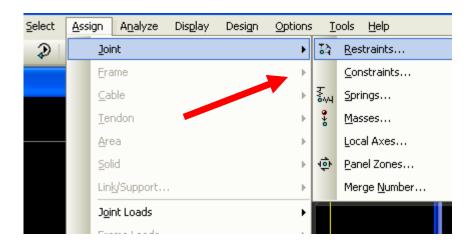
In X-Z view window, click once on the bottom left grid intersection and then click up to the top left grid intersection in order to draw the column. Next, right mouse click to lift your cursor to draw another column on the right side. Click once on the bottom right intersection and complete the column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



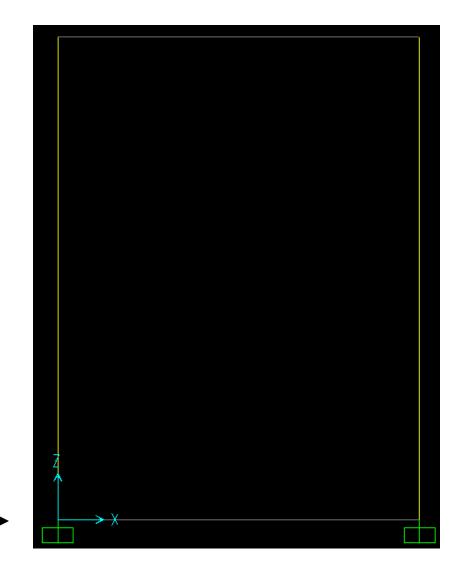
Holding your left mouse key down, drag your mouse to window around the base of the two columns in order to select bottom joints as shown



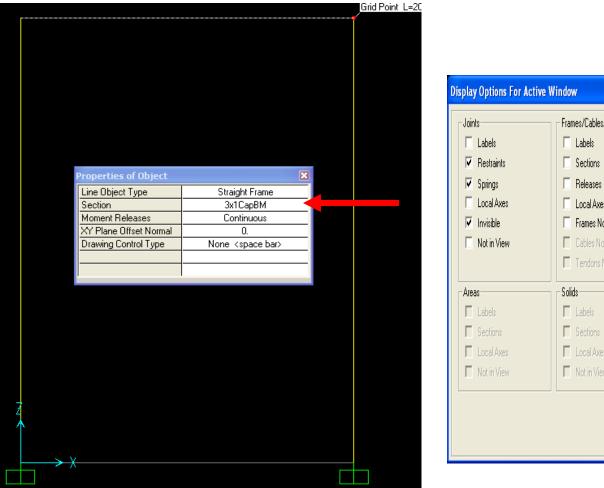
Next, use the Assign menu>Joint>Restraints to add fixed restraints as shown. Press OK



Joint Restraints
Restraints in Joint Local Directions
▼ Translation 1 ▼ Rotation about 1
▼ Translation 2 ▼ Rotation about 2
🔽 Translation 3 🔽 Rotation about 3
Fast Restraints
<u> </u>

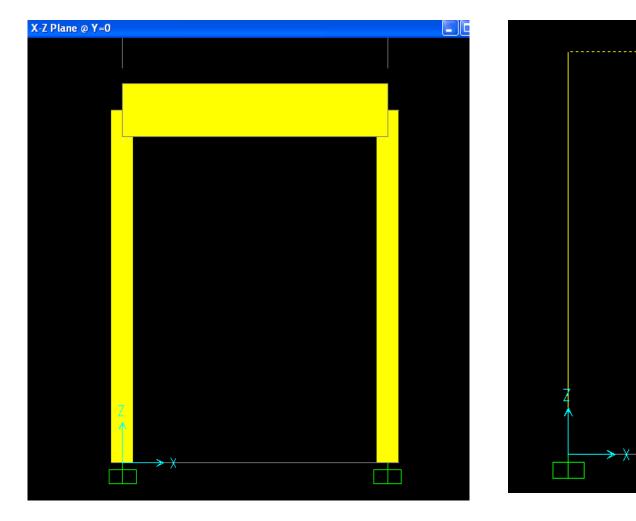


Click 'Draw frame/cable element' button, change Section property to 3x1CapBM and click twice to draw the cap beam as shown. Click 'Set Display options' button I and checkbox 'Extrude view' option and press OK



Joints	Frames/Cables/Tendons	General	View by Colors of
Labels	Labels	Shrink Objects	<ul> <li>Objects</li> </ul>
Restraints	Sections	Extrude View	C Sections
Springs	Releases	Fill Objects	C Materials
🗌 Local Axes	Local Axes	Show Edges	C Color Printer
🗹 Invisible	Frames Not in View	Show Ref. Lines	O White Background, Black Objects
Not in View	🔲 Cables Not in View	Show Bounding Boxes	C Selected Groups Select Groups
	Tendons Not in View		
Areas	Solids	Links	Miscellaneous
🗖 Labels	🗖 Labels	Labels	🔲 Show Analysis Model (If Available)
Sections	E Sections	Properties	🔲 Show Joints Only For Objects In View
🗖 Local Axes	🗖 Local Axes	Local Axes	
Not in View	Not in View	Not in View	

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

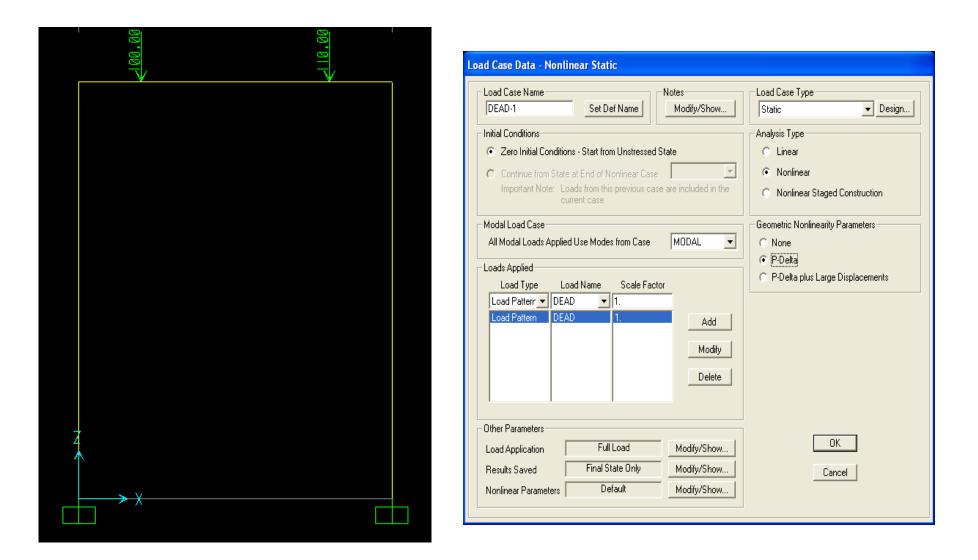


Selected beam displays as dotted line

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

<u>A</u> ssign	A <u>n</u> alyze Dis <u>p</u> lay	Design Options	<u>T</u> ools <u>H</u> elp	
20	oint	د ۱	아 🕜 🕹 🕹 🖓	Forme Delet Londo
Ð	rame	•		Frame Point Loads
⊆	jable			Load Pattern Name Units
Ŀ	endon			+ DEAD V Kip, ft, F V
A	rea			Load Type and Direction
5	olid			Forces     Moments     Add to Existing Loads
Li	ink/Support			Coord Sys GLOBAL   Generative GLOBAL  Generative Genera
Jç	oint Loads	÷.		Direction Gravity  C Delete Existing Loads
Fi	ra <u>m</u> e Loads	۱.	<u>G</u> ravity	Point Loads
C	ia <u>b</u> le Loads	<u></u>	Point	1. 2. 3. 4.
Т	endon <u>L</u> oads		Distributed	Distance 02 .8 1.
A	r <u>e</u> a Loads	•	Temperature	Load 0.  100  110 0.
S	oli <u>d</u> Loads	Þ	<u>S</u> train	Relative Distance from End-I C Absolute Distance from End-I
Li	ink√S <u>u</u> pport Loads	Þ	Defor <u>m</u> ation	
Jo	oint <u>P</u> atterns		Target <u>F</u> orce	OK Cancel
γ А	ssign to <u>G</u> roup		Auto <u>W</u> ave Loading Param	
			<u></u>	

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



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

Case Name	Туре	Status	Action	, Click to: Run/Do Not Run Cas
DEAD MODAL DEAD-1	Linear Static Modal Nonlinear Static	Not Run Not Run Not Run	Run Run Run	Show Case Delete Results for Cas
				Run/Do Not Run All Delete All Results
				Show Load Case Tree.
halysis Monitor D Always Show Never Show	ptions			Model-Alive

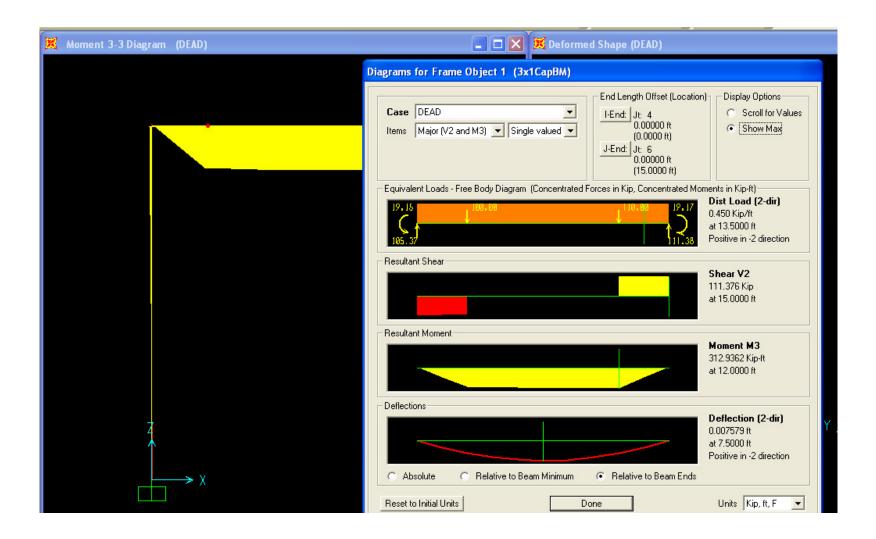


Auto
 Scale Factor

Options

 Fill Diagram
 Show Values on Diagram
 Cancel

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



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

Choose Tables for Display		
Edit		
	Select Load Patterns	Load Patterns (Model Def.) Select Load Patterns 1 of 1 Selected Load Cases (Results) Select Load Cases 3 of 3 Selected Modify/Show Options Set Output Selections Options Selection Only Show Unformatted
		OK Cancel

Using the table form, you can go to the File menu>Export current table and export to Excel or Access. You can utilize the Format-Filter-Sort menu option to format and to sort by min/max values in any specified direction.

	Displacem	ents -Filter-Sort Seler	1 Ontions							File View Format-Filter-Sort Select Options
	As Noted		<u>s</u> provide		Joint Di	splacements			-	Export Current Table  To Excel
	Joint Text	OutputCase Text	CaseType Text	StepType Text	U1 ft	U2 ft	U3 ft	R1 Radians	R2 Badians	Display Current Table To Access
•	3	DEAD	LinStatic		0	0	0	0	0	Print Current Table as Text File
	3	DEAD-1 DEAD-1	NonStatic NonStatic	Max Min	0	0	0	0	0	StepType
-	4	DEAD	LinStatic	1900	-0.000021	0	-0.003366	0	0.001546	Export All Tables
	4	DEAD-1	NonStatic	Max	-0.000019	0	-0.003366	0	0.001549	
	4	DEAD-1	NonStatic	Min	-0.000019	0	-0.003366	0	0.001549	
	5	DEAD	LinStatic		0	0	0	0	0	-
	5	DEAD-1	NonStatic	Max	0	0	0	0	0	
	5	DEAD-1	NonStatic	Min	0	0	0	0	0	
	6	DEAD	LinStatic		-0.000035	0	-0.003554	0	-0.001552	
	6	DEAD-1	NonStatic	Max	-0.000032	0	-0.003554	0	-0.001555	$\mathbf{V}$
	6	DEAD-1	NonStatic	Min	-0.000032	0	-0.003554	0	-0.001555	
	~1	DEAD	LinStatic		-0.000028	0	-0.011039	0	0.000026	
_	~1	DEAD-1	NonStatic	Max	-0.000025	0	-0.01105	0	0.000026	Microsoft Excel - Book1
	~1	DEAD-1	NonStatic	Min	-0.000025	0	-0.01105	0	0.000026	
										Ele Edit View Insert Format Tools Data Window Help Type a question for help
										Image: Image
	1									I I I I I I I I I I I I I I I I I I I
•										Image: Second state     Image: Second s
	d: <b>  4  4</b>	1 🕨 🕅	of 15					Add Tables	▶ Done	I I I I I I I I I I I I I I I I I I I

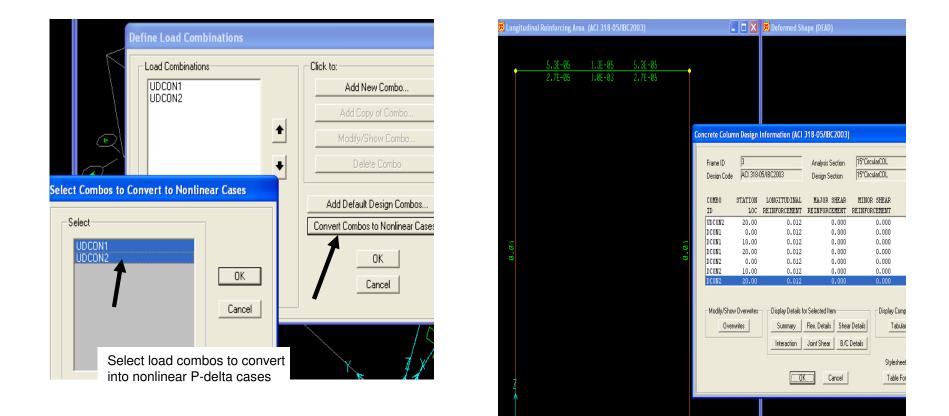
2)	<u>E</u> ile <u>E</u> dit	⊻iew Insert	Format 1	ools <u>D</u> ata	Window H	elp				uestion for hel	
	🞽 🖬 I			- 10	- B I	<u>n</u>   ≣ ∄		\$ % ,	♦.00.00. ♦.00.	E 🛊   🖽 •	• 🖄 • <u>A</u>
	A4	▼ fx	-								
	A	В	C	D	E	F	G	Н		J	K
1		loint Displace									
2	Joint	OutputCase			U1	U2	U3	R1	R2	R3	
3	Text	Text	Text	Text	ft	ft	ft	Radians	Radians	Radians	
	3	DEAD	LinStatic		0	0	0	0	0	0	
	3	DEAD-1	NonStatic	Max	0	0	0	0	0	0	
-	3	DEAD-1	NonStatic	Min	0	0	0	0	0	0	
-	4	DEAD	LinStatic		-0.000021	0	-0.003366	0	0.001546	0	
	4	DEAD-1	NonStatic	Max	-0.000019	0	-0.003366	0	0.001549	0	
	4	DEAD-1	NonStatic	Min	-0.000019	0	-0.003366	0	0.001549	0	
	5	DEAD	LinStatic		0	0	0	0	0	0	
1		DEAD-1	NonStatic	Max	0	0	0	0	0	0	
	5	DEAD-1	NonStatic	Min	0	0	0	0	0	0	
-	6	DEAD	LinStatic		-0.000035	0	-0.003554	0	-0.001552	0	
	6	DEAD-1	NonStatic	Max	-0.000032	0	-0.003554	0	-0.001555	0	
	6	DEAD-1	NonStatic	Min	-0.000032	0	-0.003554	0	-0.001555	0	
~	~1	DEAD	LinStatic		-0.000028	0	-0.011039	0	0.000026	0	
	~1	DEAD-1	NonStatic	Max	-0.000025	0	-0.01105	0	0.000026	0	
8	~1	DEAD-1	NonStatic	Min	-0.000025	0	-0.01105	0	0.000026	0	
9											
0		0									
!1											
2					/ Sheet3 /		<				

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

Concrete Frame Design Preferences for ACI 318-05/IBC2003

<b></b>			Item Description		
	Item	Value	Seismic Design Category. This is either 🔨		
1	Design Code	ACI 318-05/IBC2003	"A", "B", "C", "D", "E" or "F".	Define Load Combinations	
	Time History Design	Envelopes			
3	Number of Interaction Curves	12		Load Combinations	Click to:
4	Number of Interaction Points	5			Add New Combo
5	Consider Minimum Eccentricity	No			
	Seismic Design Category	B			Add Copy of Combo
	Phi (Tension Controlled)	0.9			
	Phi (Compression Controlled Tied)	0.65			Modify/Show Combo
	Phi (Compression Controlled Spiral)	0.7			
	Phi (Shear and/or Torsion) Phi (Shear Seismic)	0.75		Add Code-Generated User Load Combinations	Delete Combo
	Phi (Joint Shear)	0.85			
	Pattern Live Load Factor	0.85		Select Design Type for Load Combinations	
	Utilization Factor Limit	0.95			Add Default Design Combos
	CAREGOTT GOOD ENTRY			C Steel Frame Design	Convencombos to Nonlinear Cases
				Concrete Frame Design	
					ок
				C Aluminum Frame Design	
				C Cold Formed Frame Design	Cancel
			~	C. Pridao Desian	
				O Bridge Design	-
			Explanation of Color Coding for Values		
			Blue: Default Value	Set Load Combination Data	
- Set T	o Default Values	Reset To Previous Values	Black: Not a Default Value	OK Cancel	Y. 4 X
			Red: Value that has changed during	OK Cancel	K 47
	All Items Selected Items	All Items Selected Items	the current session		
		OK Cancel			

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



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



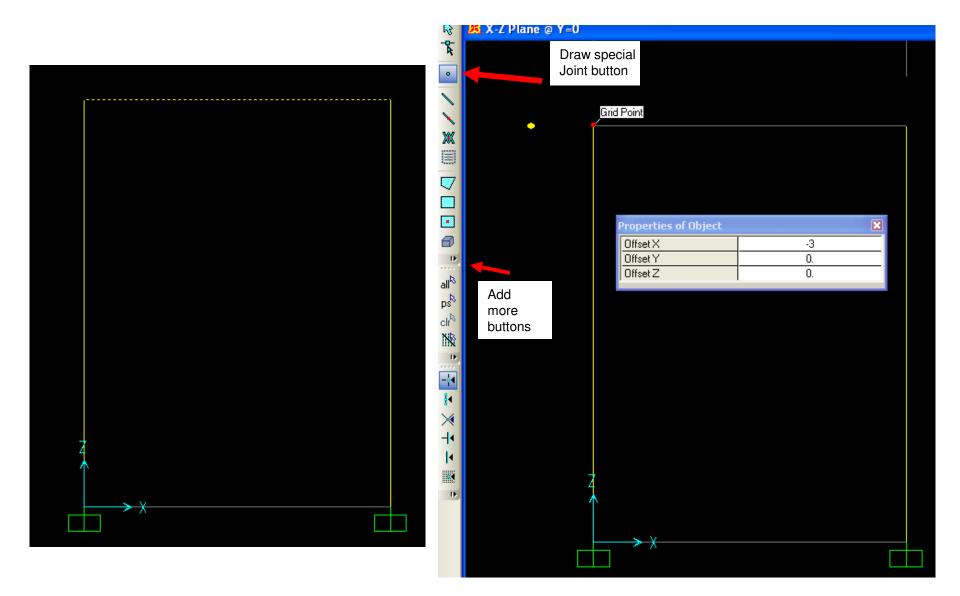
Frame Properties	
Properties Find this property: 1.5X1CapBM 15"CircularCOL 3x1CapBM 3x3	to: Import New Property Add New Property Add Copy of Property Modify/Show Property Delete Property
	Cancel

Add Frame Section Pro			_
Frame Section Property	Туре	Other	-
Click to Add a Section—			
General	Nonprismatic	Section Designer	
	C	ancel	

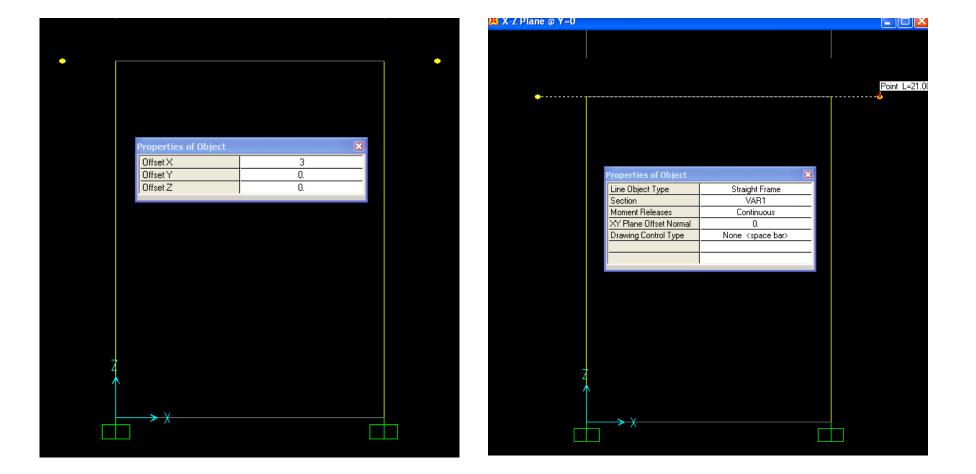
Accept the default VAR1 section name and type the values shown below for sections, lengths and length type. SAP2000 lets you define the length type by variable percentage or absolute length using current units (ft in this example). Press OK twice to return to the model

	VAR1		Dis	play Color 📃
Section Notes		Modify/Show Notes		
End Section	Length	Length Tupe	FI33 Variation	EI22 Variation
		Absolute 💌	Parabolic 💌	
3x1CapBM 3x1CapBM 1.5X1CapBM	3 15 3	Absolute Absolute Absolute	Parabolic Parabolic Parabolic	Linear Linear Linear
Бру	Insert	Modify	Delete	
	3x1CapBM 3x1CapBM 1.5X1CapBM	End Section Length 1.5×1CapBM ▼ 3 3x1CapBM 3 1.5×1CapBM 3 1.5×1CapBM 3 1.5×1CapBM 3	End Section       Length       Length Type         1.5X1CapBM       3       Absolute          3x1CapBM       3       Absolute          3x1CapBM       3       Absolute          1.5X1CapBM       3       Absolute          1.5X1CapBM       3       Absolute          1.5X1CapBM       3       Absolute	End Section       Length       Length Type       EI33 Variation         1.5X1CapBM       3       Absolute       Parabolic       Image: Comparabolic and the second

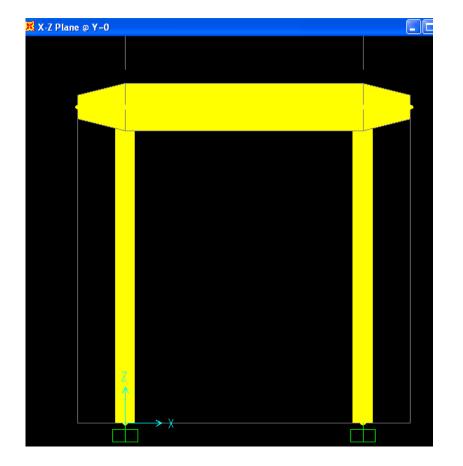
Select the existing beam and press the Del (delete) key on your keyboard. Next, click the 'Draw special joint' button, type -3(feet) in the X direction and click the top of the left side column to draw a joint 3 feet away as shown. You may or may not have to add the Draw special joint button to your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

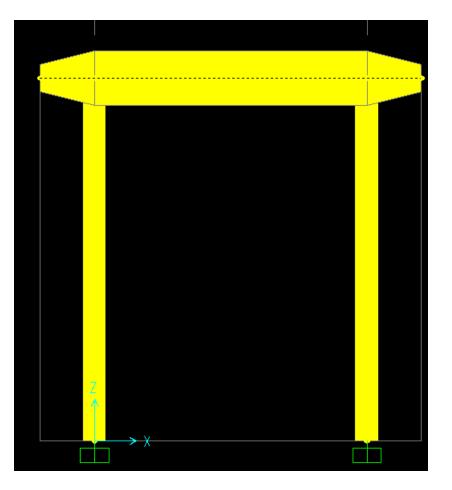


Next, change offset dimension to (+) 3 feet in the X direction and click the top of the right column to add another joint. Next, click 'Draw frame/cable element' button , change section to VAR1 with continuous connection and connect the dots by clicking once on the special joint on the left and then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

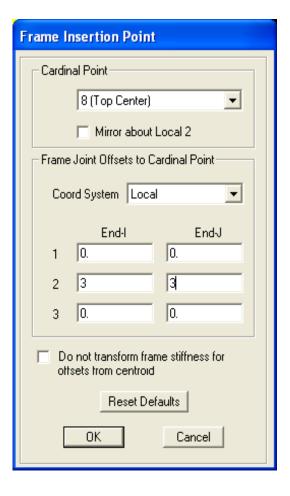


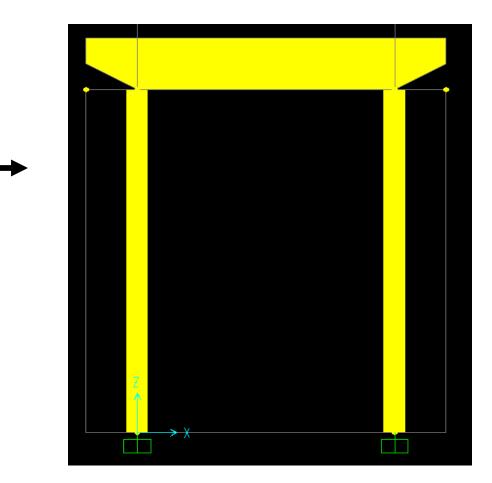
Press Esc key or click select arrow buttor. to exit draw mode. Click the 'Set display options' menu and checkbox extrude view for the planar view. Hold down your left mouse key and drag right to left while intersecting the beam in order to select





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





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

Joint Restraints
Restraints in Joint Local Directions
🔲 Translation 1 🔲 Rotation about 1
🔲 Translation 2 🔲 Rotation about 2
🔲 Translation 3 🔲 Rotation about 3
Fast Restraints
OK Cancel

Link/Support Directional Properties Edit	
Identification Property Name Direction Type NonLinear Yes	Hysteresis Type And Parameters Hysteresis Type Kinematic No Parameters Are R Takeda Pivot
Properties Used For Linear Analysis Cases Effective Stiffness 0. Effective Damping 0. Multi-Linear Force-Deformation Definition Used Force 1 0. 1 0. 0. 2 0. 1 0. 0. 4 1. 1. 5 10. 1. V Order Rows Delete Row 2 Add Row 6	Hysteresis Definition Sketch Multilinear Plastic - Kinematic
	OK Cancel

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

Link/Support Directional P	roperties					
_ Identification						
Property Name	LIN1					
Direction	JU1					
Туре	T/C Friction Isolator					
NonLinear	Yes					
Properties Used For Linear Analysis Cases						
Effective Stiffness	0.					
Effective Damping	0.					
Properties Used For Nonlinear	Analysis Cases					
Stiffness for Compression	0.					
Stiffness for Tension	0.					
Gap Opening for Compression	on 0.					
Gap Opening for Tension	0.					
Damping Coefficient	0.					
Camping Coemciant	,					
<u> </u>	Cancel					

	۱ <u> </u>					
Property Na	me	FF	REQD1			
Section Notes				Modify/Show		
000.0000000			-			
				Display	Color	
Degrees of Freedom with Nonzero Properties Shown Checked						
💌 U1	🔲 U1U2	🗖 U1U3	🔲 U1R1	🔲 U1R2	🔲 U1R3	
	🗖 U2	🗖 U2U3	🔲 U2R1	🔲 U2R2	🔲 U2R3	
		🗖 U3	🔲 U3R1	🔲 U3R2	🔲 UBRB	
			🗖 B1	🗖 R1R2	E B1B3	
	Sum	metrical		🗖 R2	E R2R3	
	Jym	notrical			E R3	
Show Prope	rties for this D	egree of Free	dom			
U1			dom			
Jui		<u> </u>				
Properties fo	r Degree of F	reedom U1 —				
r toperdes to	quency 9	Stiffness (Re)	Damping (	lm) S,D		
		Kip/in	Kip/in			
Free	Hz					
Free 1	25.	0.15	0.19			
Free	25. 40.	0.15	0.19 0.12		F	
Free	25.		0.19			
Free	25. 40. 50.	0.15	0.19 0.12 0.15		Clear	
Free	25. 40.	0.15	0.19 0.12		Clear	

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

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

Frame Local Axis	
Angle from Default Direction	
Angle in Degrees 0	
Advanced Axes Advanced	
Cancel	

Assign Frame Releases						
Frame Releases						
	<u>Release</u> Start End		<u> </u>			
Axial Load						
Shear Force 2 (Major)	Γ	Γ				
Shear Force 3 (Minor)	Γ	Γ				
Torsion	◄	Γ	0			
Moment 22 (Minor)	◄	$\checkmark$	0	0		
Moment 33 (Major)		) [	0			
No Releases Units Kip, ft, F						
OK Cancel						

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

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

## **Darrell Foster**

CSI Houston, TX office Tel: 713-533-4420

## **Computers & Structures, Inc.**

TECHNOLOGY FOR A BETTER WORLD

darrell@csiberkeley.com

www.csiberkeley.com