APPLICATION OF SAP2000: ANALYSIS OF A WARREN TRUSS BRIDGE

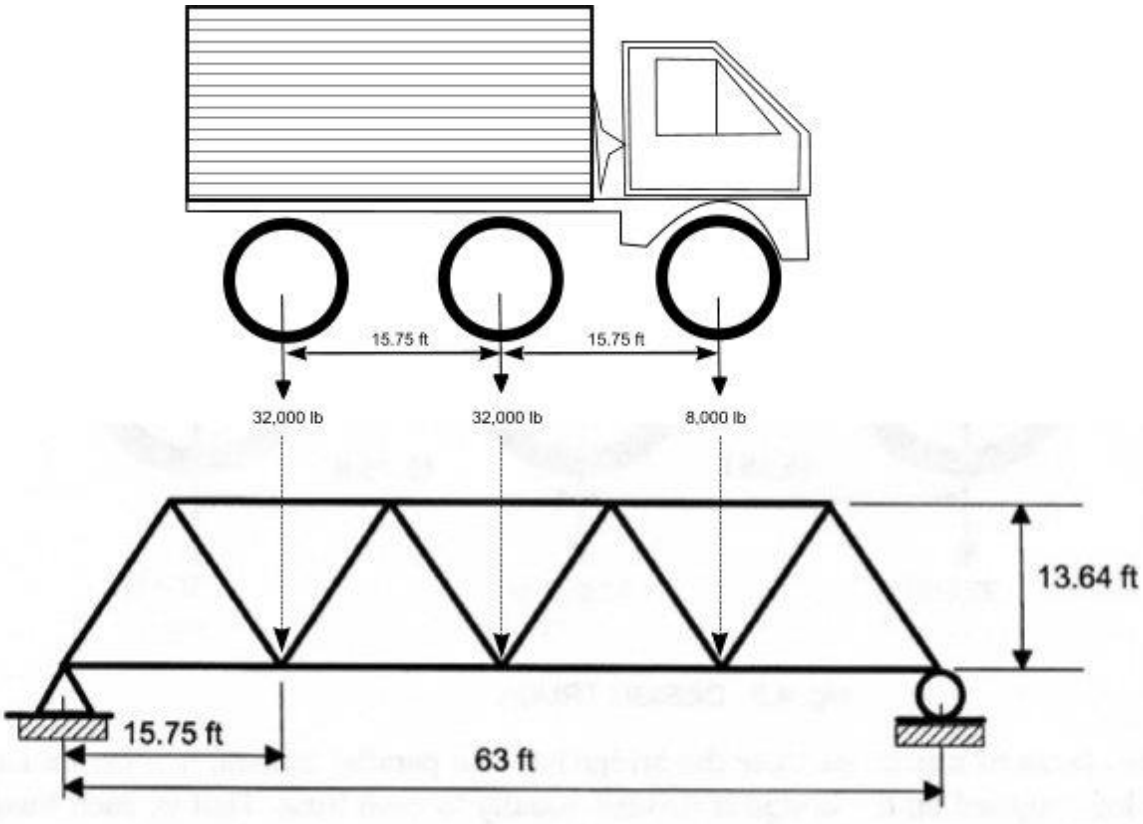
By Ait Elmouden Abdellah | LaGuardia Community College 31-10 Thomson Ave. Long Island City, NY 11101

Problem statement

Perform the structural analysis of a steel truss bridge that spans a total distance of **63 ft**. The truss bridge consists of two parallel trusses and the truss configuration is given in the figure below. The lower chords of the trusses support the roadway which in turn carries the car and truck traffic. The object of this analysis is to **calculate the internal forces in each member of the bridge** under the effect of the weight of the bridge and roadway (the dead load) and the weight of the vehicles (the live loads) that are likely to cross the bridge. It is herein assumed that the bridge carries two lanes of vehicle traffic. Make a first estimate of the internal forces in each truss member under regular truck traffic. The analysis must be performed using the SAP2000 package. Use the simplifying assumptions given below to analyze the four-bay Warren truss bridge shown in Fig 1.

Assumptions

The bridge is supposed to carry its ***dead load*** (that is, the weight of the steel members and the weight of the roadway), as well as the ***live load*** (that is, the weight of the cars and trucks) that will cross over it. In order to simplify the analysis process, it is assumed that the live load can be represented by the truck configuration shown below in Fig 1. The effect of this truck is assumed to model the effect of the heaviest trucks that are expected to cross the bridge in its lifetime.



By Ait Elmouden Abdellah | LaGuardia Community College 31-10 Thomson Ave. Long Island City, NY 11101

Fig 1: Four-Bay Warren Truss Bridge & Design Truck

Assume that the bridge is **26 ft wide** with a two-foot curb on each side and that the roadway carries two lanes of traffic. Because the bridge has two parallel trusses, it is herein assumed that the total load applied on the bridge is divided equally to each truss. That is, each truss will carry the weight of its members, half the weight of the roadway and one lane of traffic (that is, one truck like the one shown in Fig. 1).

When calculating the weight of the roadway, assume that it is formed by one concrete deck with an **8-inch thickness**. Concrete weighs **150 lb/ft**^{3} and the weight of concrete must be counted as part of the applied load. Since the roadway is carried by the lower chords of the truss, the weight of the roadway will have to be divided to each joint of the lower chord.
At this stage assume that each of the steel truss members has cross sectional area $A = 2Q$ in^2 and the modulus of elasticity of the steel is E = 29,000,000 lb/in^2. Steel weighs **490 lb/ft**^{3}. The weight of each member will be assumed to be equally divided between the two joints at each end of the member

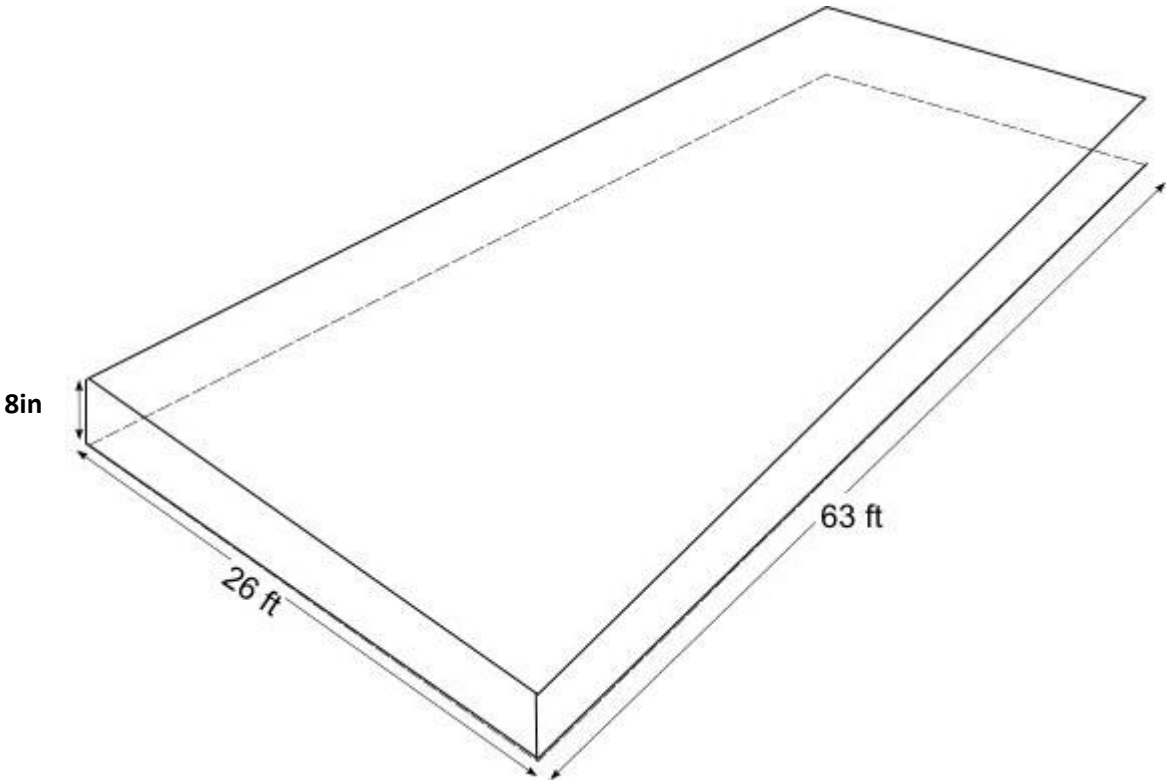
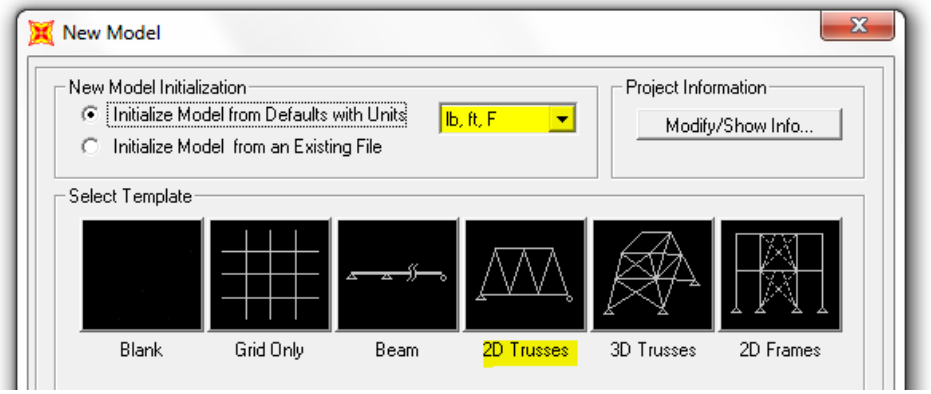


Fig 2 Concrete Deck

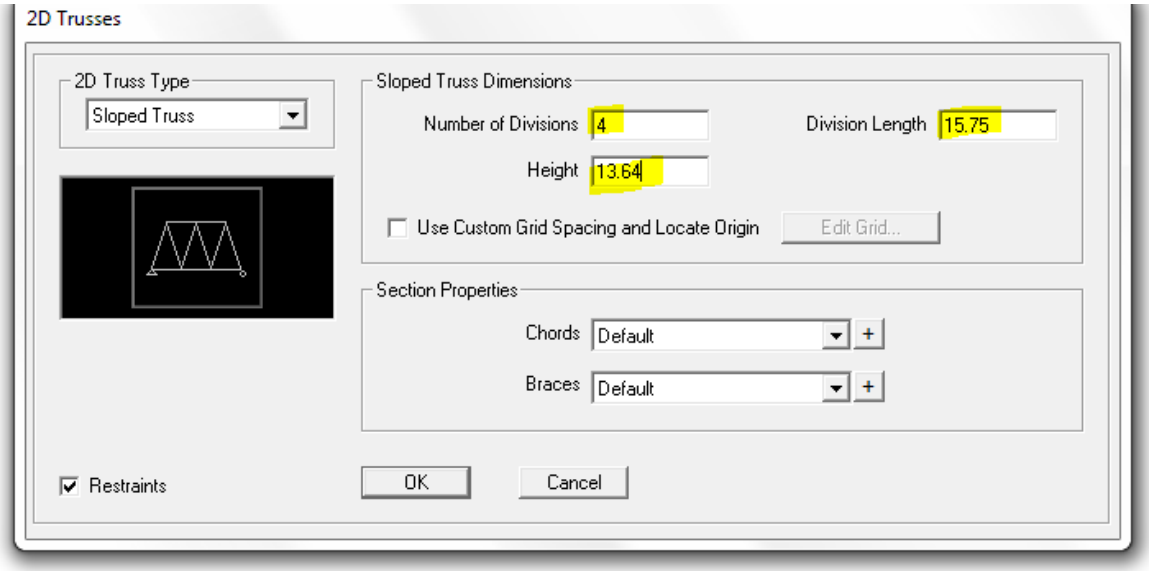
Step 1: Define Model Geometry

- The first step when you start SAP2000 is to start A New Model under the File menu.

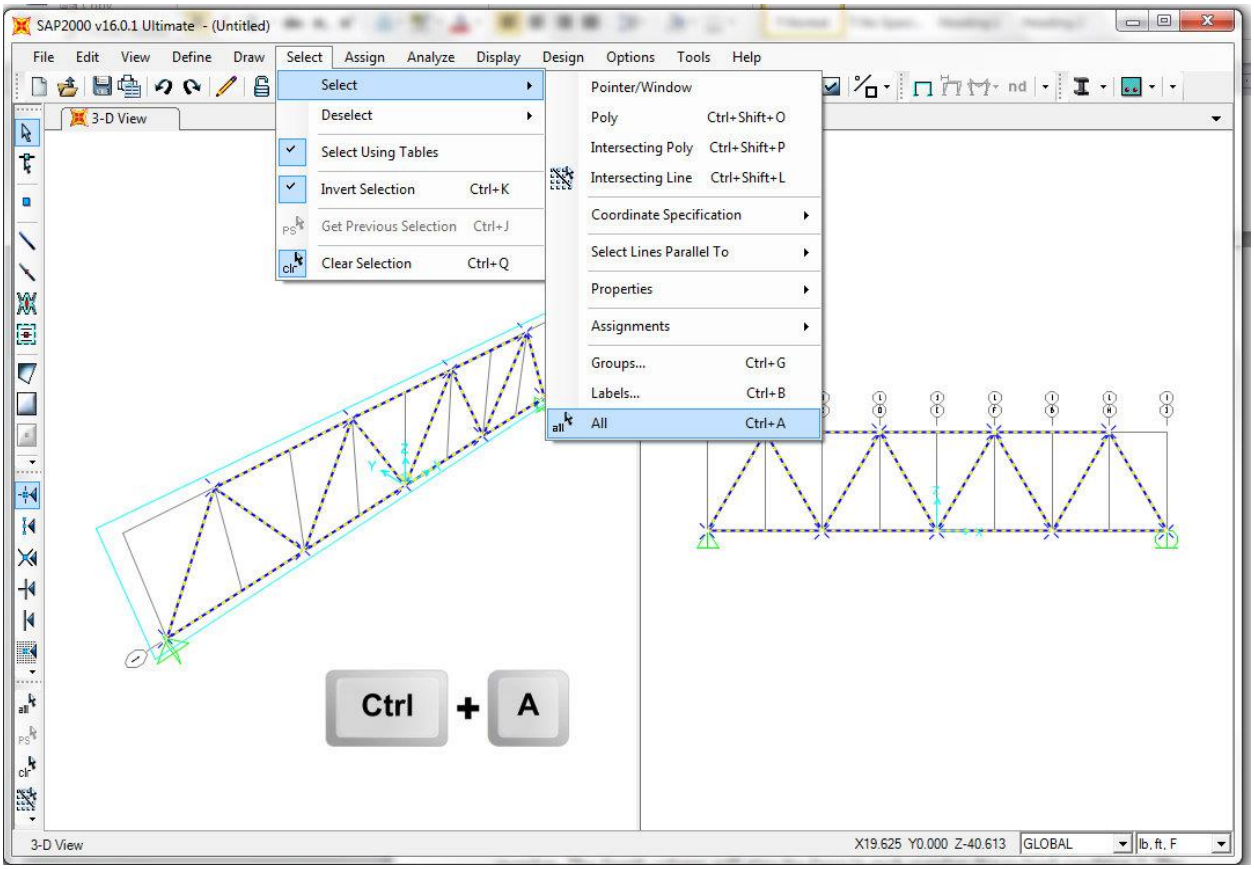
- **Basic Units**: If you are starting a new model, choose the Initialize Model from defaults with Units you want to use (Fig3). The units used to start a model become the basic units for that model. Set the units to lb-ft. Then, select from the template menu the 2D Trusses template to start.



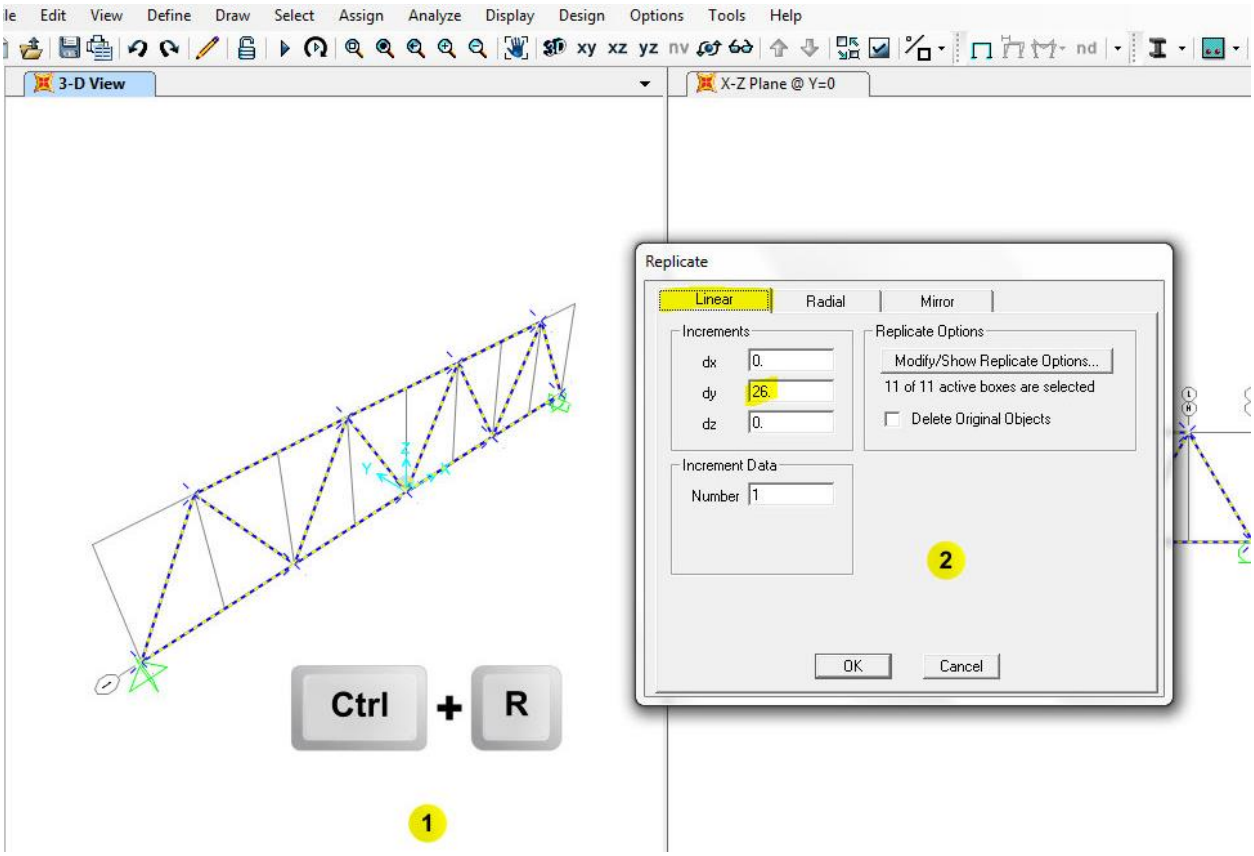
- From the 2D Trusses template, set the number of divisions to **4**. Enter a height of **13.64** ft, a division length of **15.75**. Leave the section properties as default, and make sure that the restraints box on the lower left corner is checked.



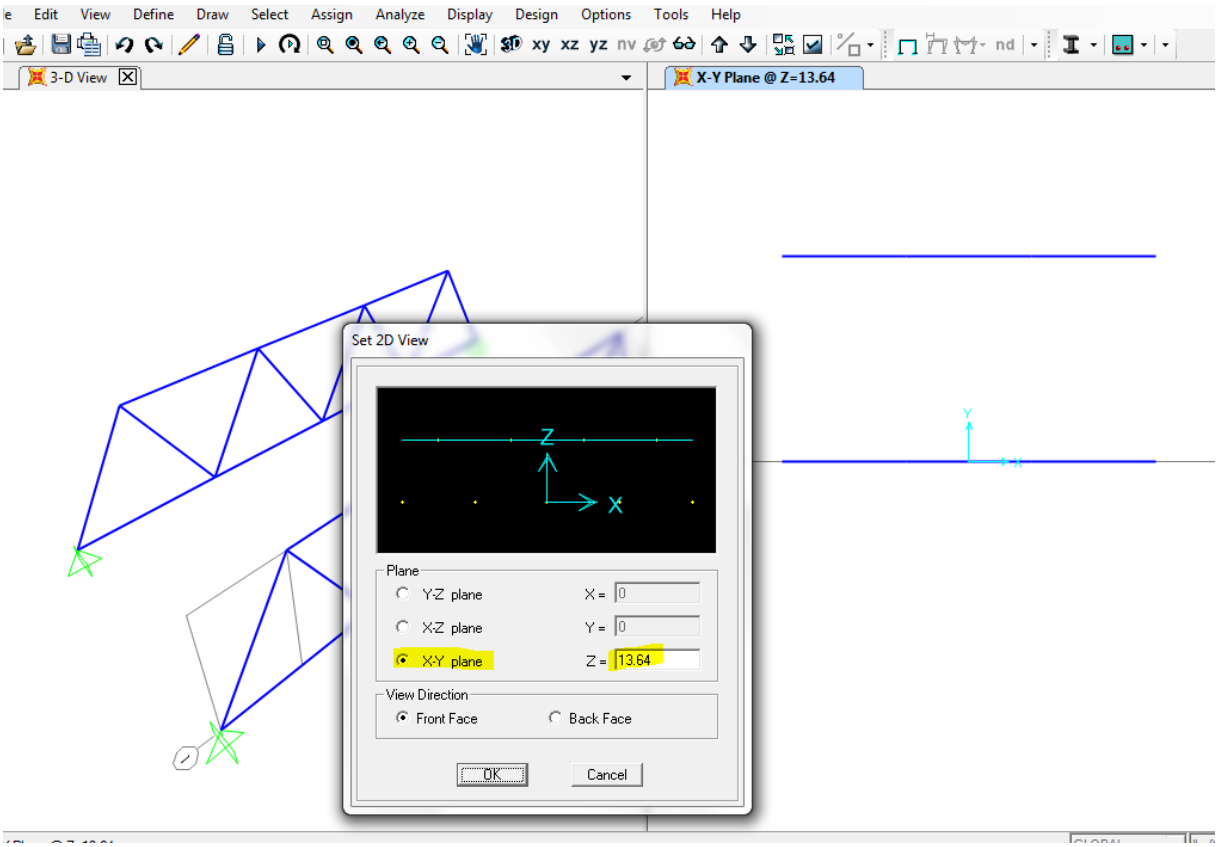
truss: A 3-D and an X-Z plane view. Click inside the 3-D window. From the top menu click on Select -> Select -> All, or use the command Ctrl +A, that will select the entire truss as well.



- Now the truss elements will appear in dotted lines. From the Edit menu, select the option Replicate, or use the command Ctrl +R to get the Replicate dialog box. When the box opens, select the linear tab, and enter a value of 26 feet on the **dy** box. Leave 1 for the box Number and click OK.

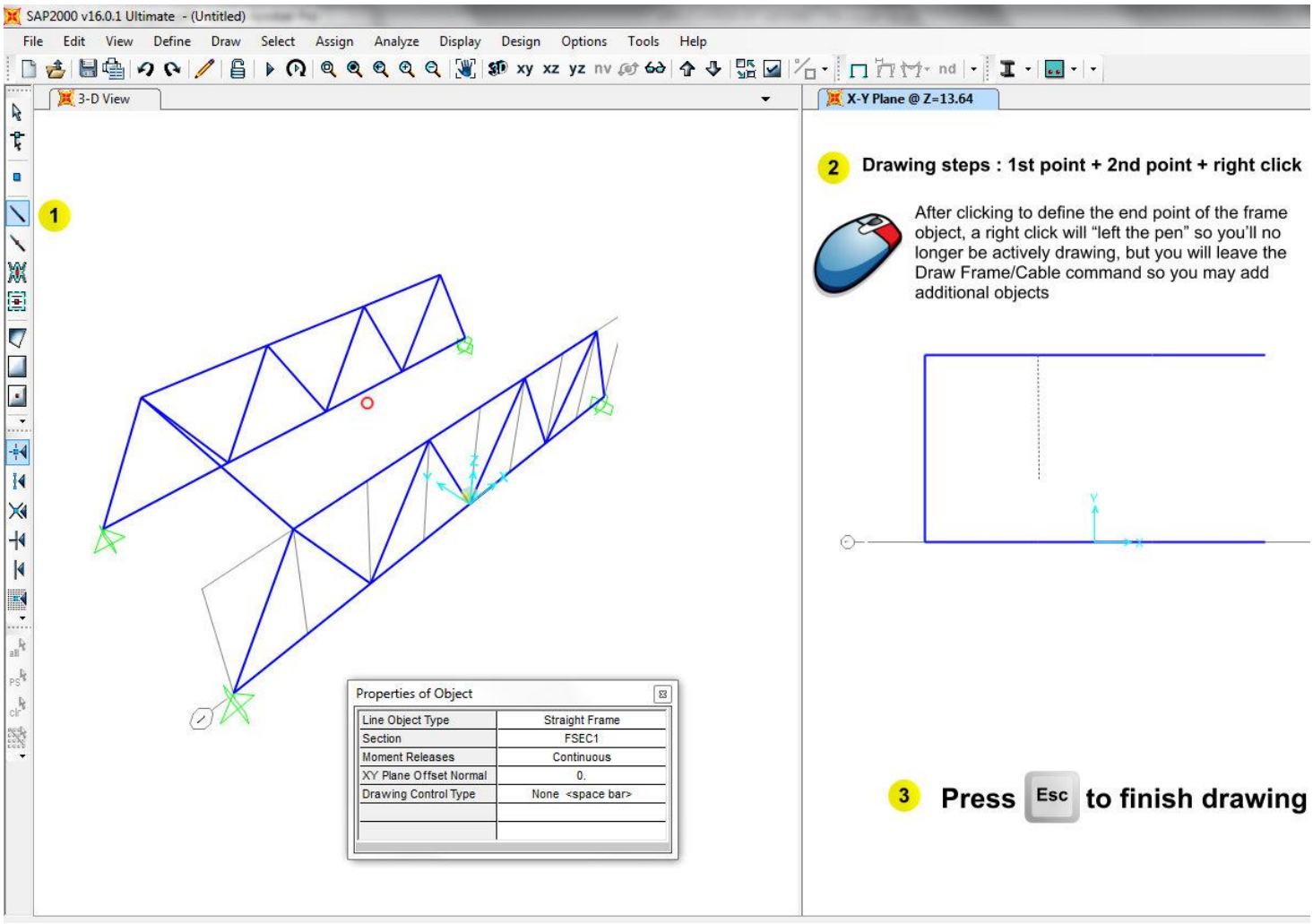


The truss will replicate in the 3-D window. Now select the opposite window by clicking on it, and then go to the View menu and select the 2-D view. The 2-D View box will show you the different planes available. Select the X-Y plane, and make sure that the Z value is set to **13.64 ft**, which is the top of the trusses.

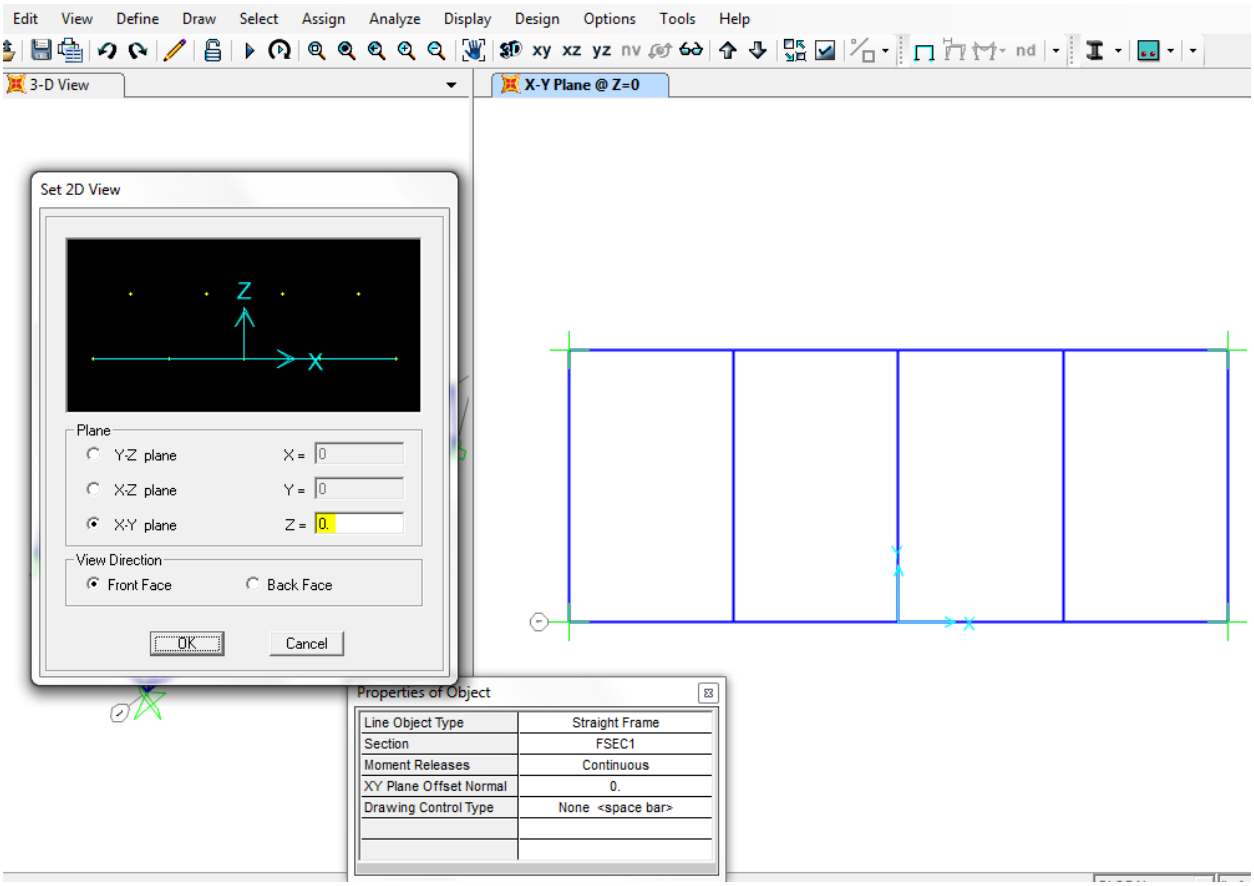


From the Draw menu, select **Draw** Frame/Cable/Tendon (or selected from the left tool bar).

The pointer of the mouse will convert into a pointing down arrow. Click on the top nodes of the truss to draw the top chord members of the truss as it is show in the pictures below, joining the top nodes of the truss.



Draw the bottom cables of the bridge: Again from the View menu, select the 2-D view. This time, set the X-Y plane distance to 0 to reach the bottom nodes in the truss. Repeat the process performed for the top nodes. Joint the nodes drawing frame elements from the Draw menu. The end result should look similar to the picture that follows.

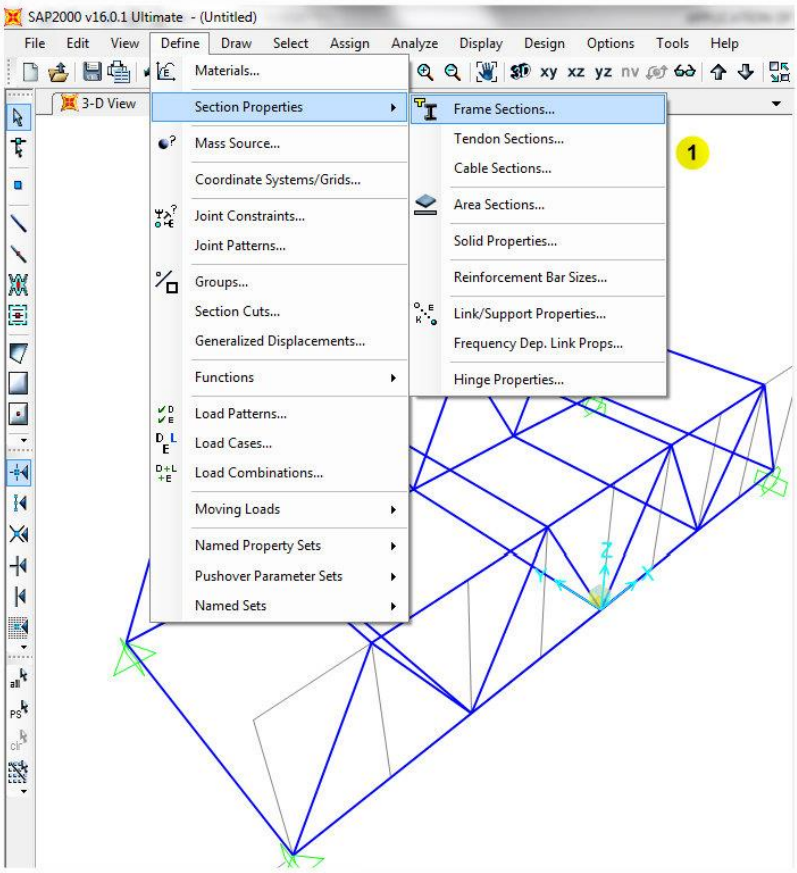


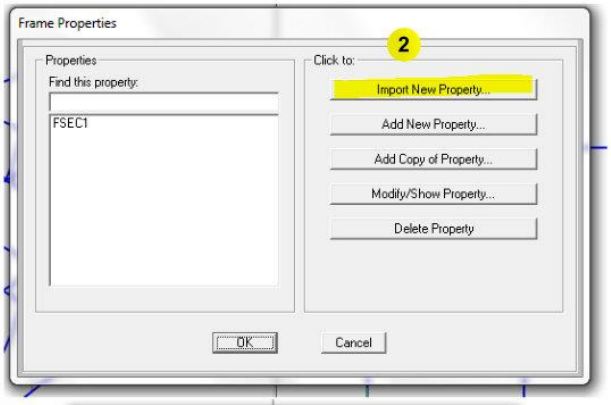
Step 2: Defining Structural Sections

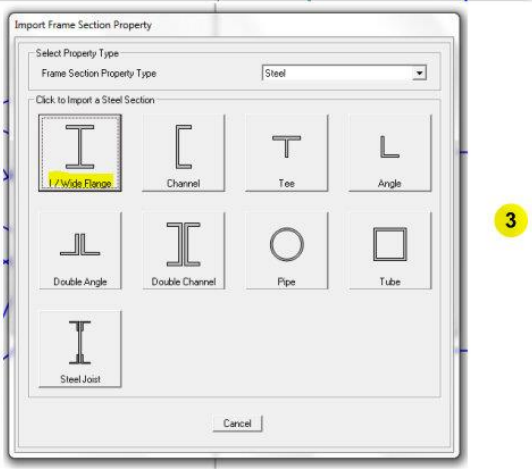
At this stage assume that cross sectional area, A = 20 in^2, is the same for each steel member.

From the **define** menu, choose **Section Properties -> Frame Sections**. This will open the Define Frame Properties window:

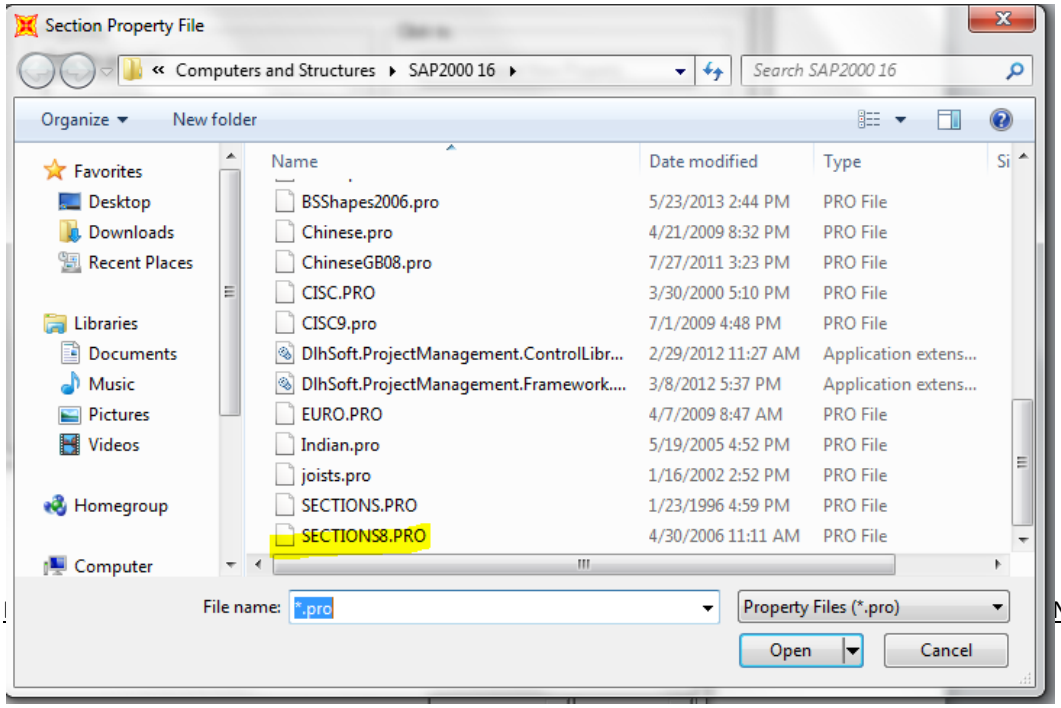
- Click on the **Import New Property**.
- From Frame section property type list select **Steel**
- From steel section Click **I/Wide flange**
-





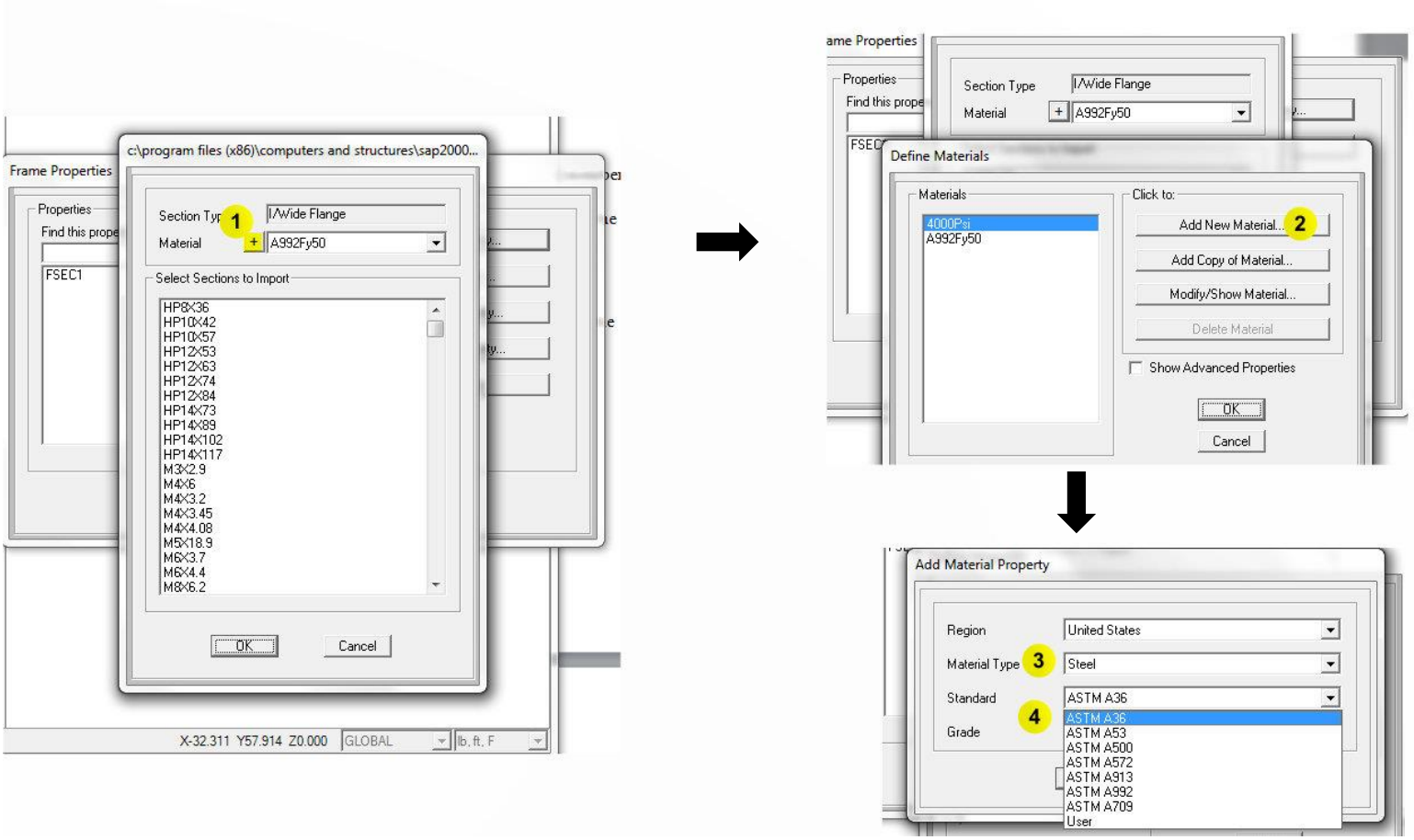


- From the Section Property File windows select **SECTIONS8.PRO** (last file in the list)

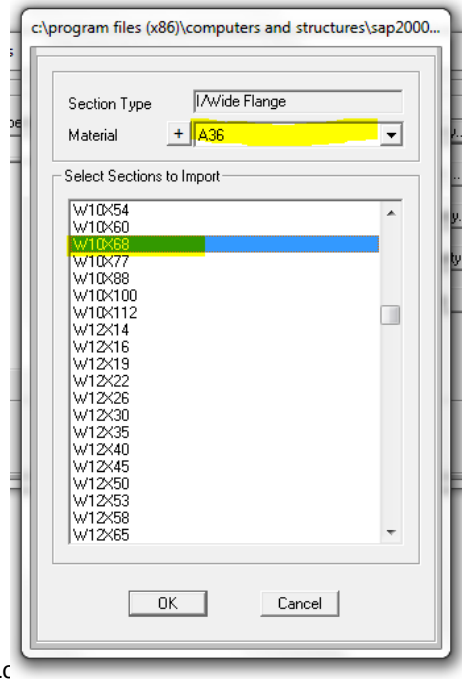


NY 11101

- Click Material (1), then new Material (2)
- Select Material as steel from Material type (3)
- Select ASTM 36 from Standard (A36 steel is a standard steel and a common structural steel in the United States)



- Click OK, then OK again
- Select A36 from Material and W10x68 From Select Section to Import
- Click OK, then OK again

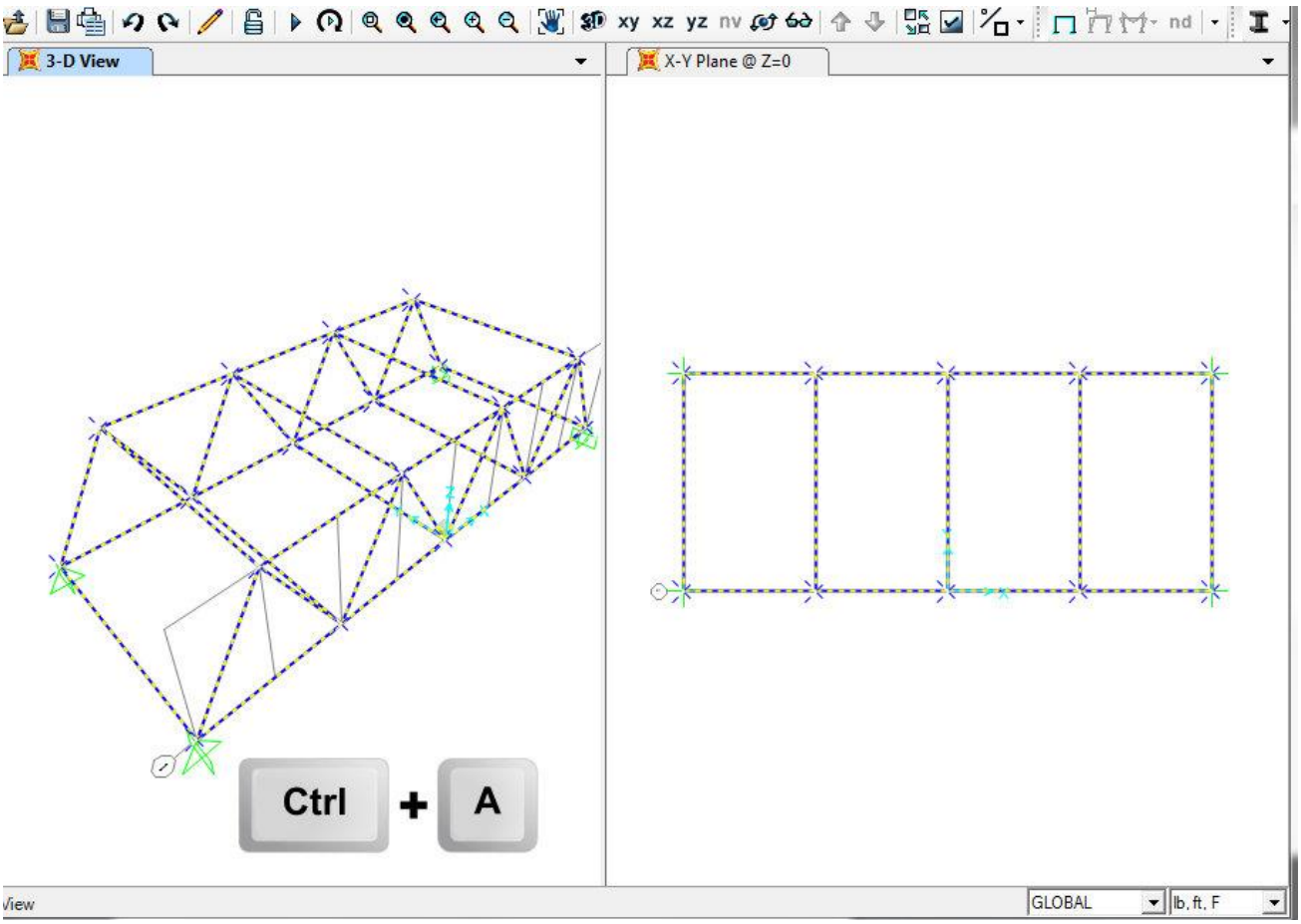


Note: *W10x68 is the Designation of Wide flange 20" cross section area. For More type of* ***S*** *or* ***W*** *I-beams visit :*
http://www.structural-drafting-net-expert.com/steel-beam.html

Step 3: Assigning a Structural Sections

Now we will assign the W10x68 I-beam (20” cross section) to our bridge:

- Press Ctrl +A to select the whole bridge
- Go to Assign/Frame/Frame Section
- From the Frame property window select W10x68 then press OK



Now we finished designing our bridge frame and assigning a 20in cross section area to steel member. Next step will be adding the concrete deck.

Step 4: Defining and assigning the roadway (Concrete Deck)

Roadway property:

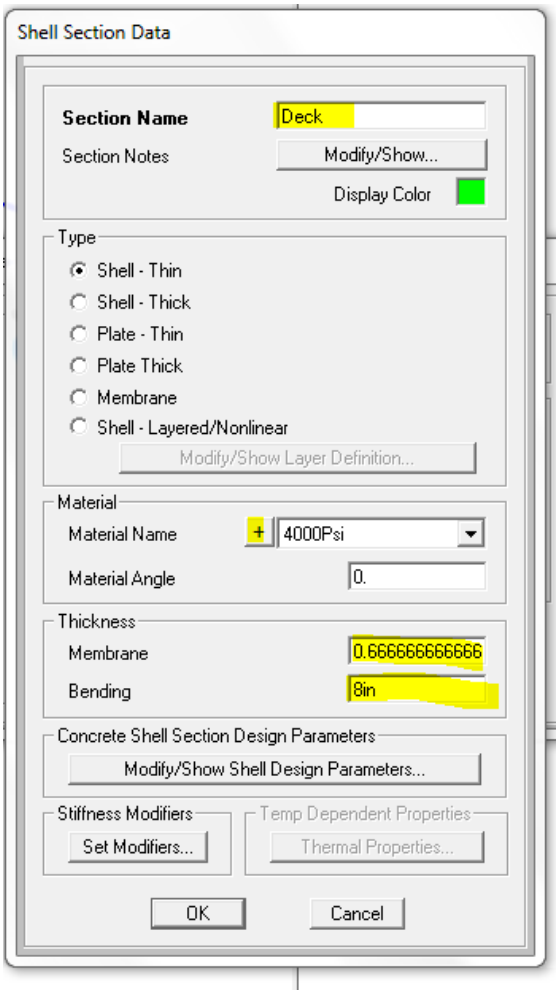
- Concrete deck thickness = 8in
- Concrete weights = 150 lb/ft^3
- The roadway will have to be divided to each joint of the lower chord

In SAP200:

- From the View menu, select the 2-D view. This time, set the X-Y plane distance to 0 to reach the bottom nodes in the truss.

a- <u>Define the concrete area :</u>

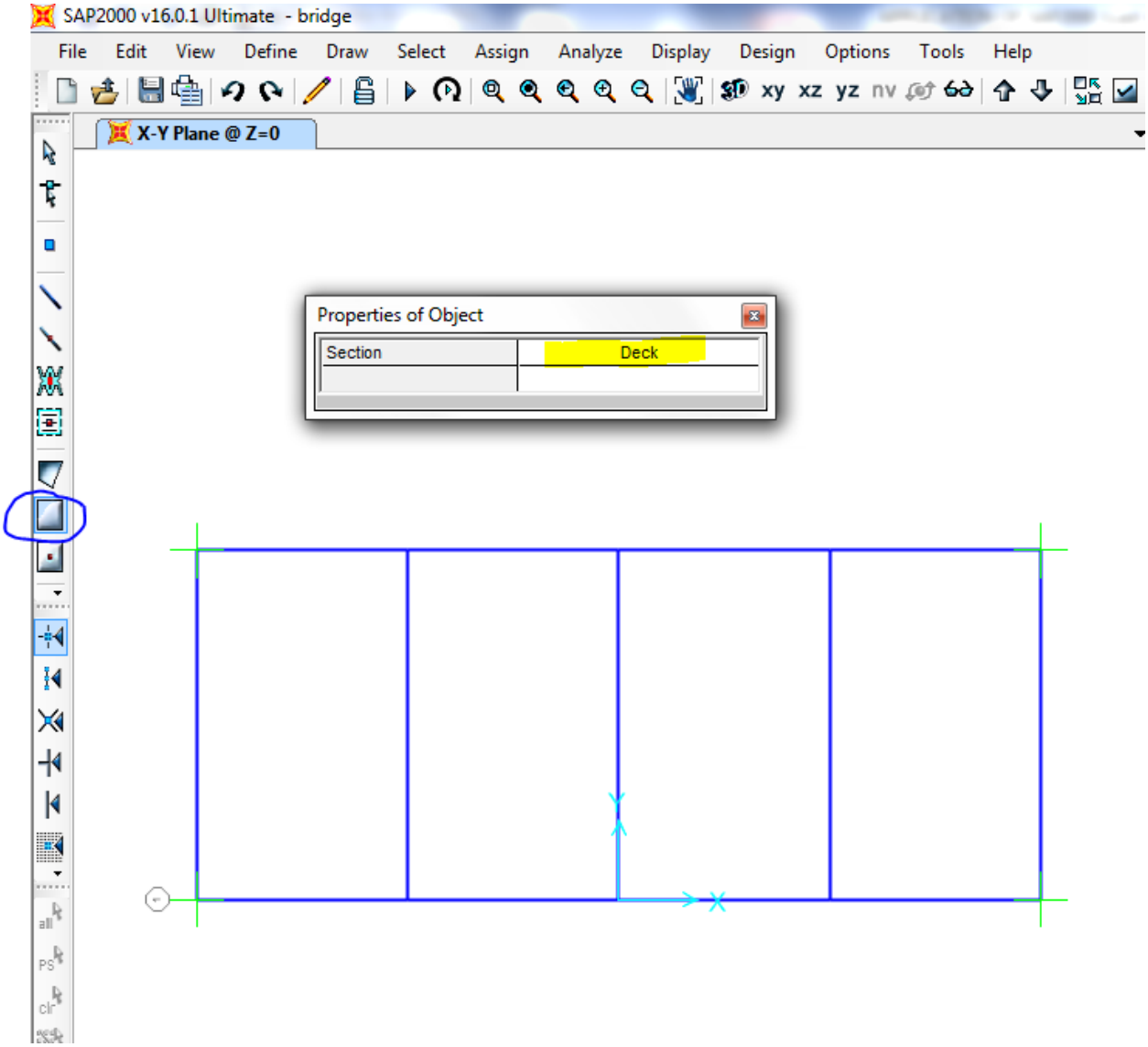
- Go to Define/Section property/Area section
- From the Select Section Type to Add Select Shell
- Click Add new Section
- Type DECK into the section name edit box
- Verify that the Material Name is set to 4000Psi in the Material area. Clicking + button will display the define materials from where material Properties may be altered or added.
- Set the Thickness (both Membrane and Bending) to 8in (make sure you add "in" after 8).
- Click the OK button then click the OK to complete The Deck definition



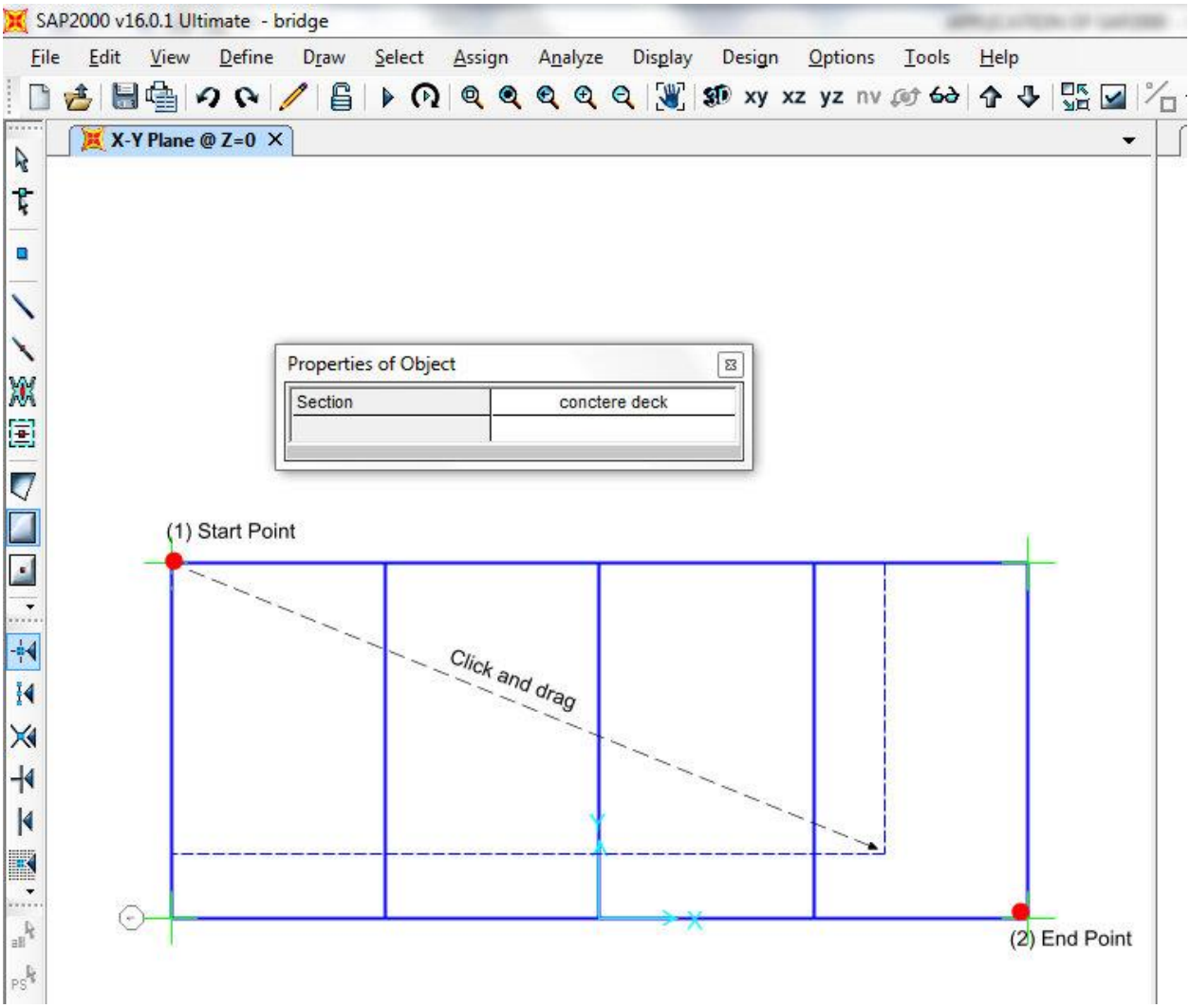
Note: you can modify you sections property any time you want

b- <u>Assign the concrete area :</u>

- From the View menu, select the 2-D view. This time, set the X-Y plane distance to 0 to reach the bottom nodes in the truss.
- Select Draw Rectangular Area from the left tool bar, then select the D**eck** section from Properties of Object window



- Draw the rectangle area following the figure below (*The drawn concrete rectangle will be highlight in red*)



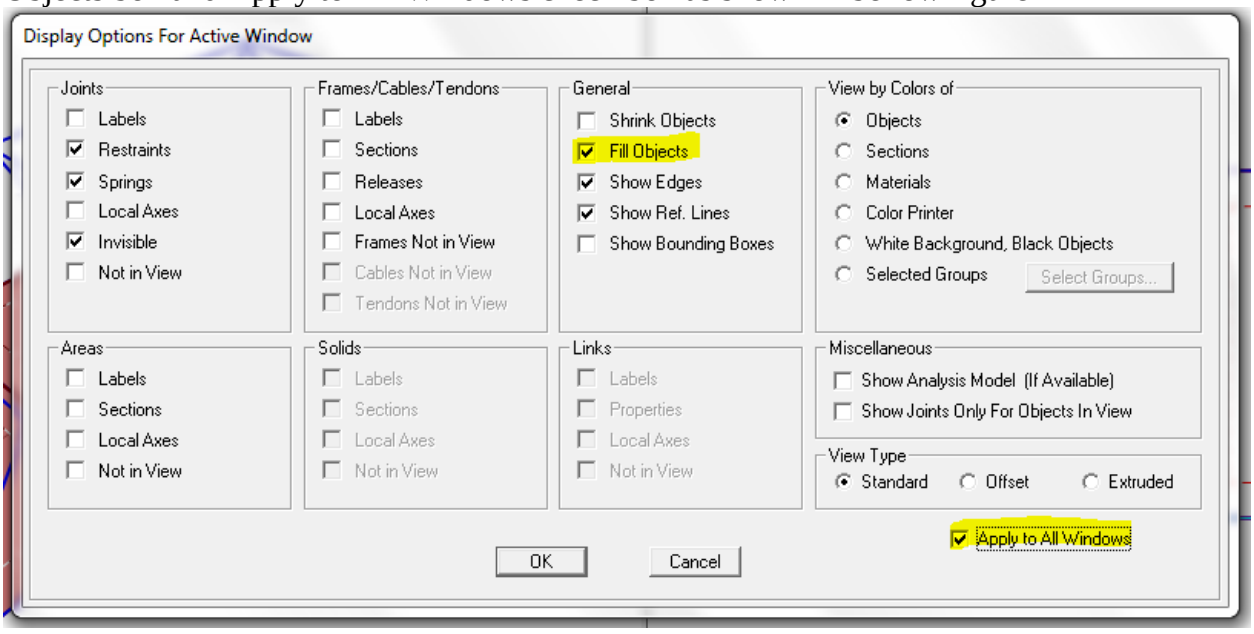
- Press Esc to exit the Tool and go back to selection mode 

Mesh the Area Object

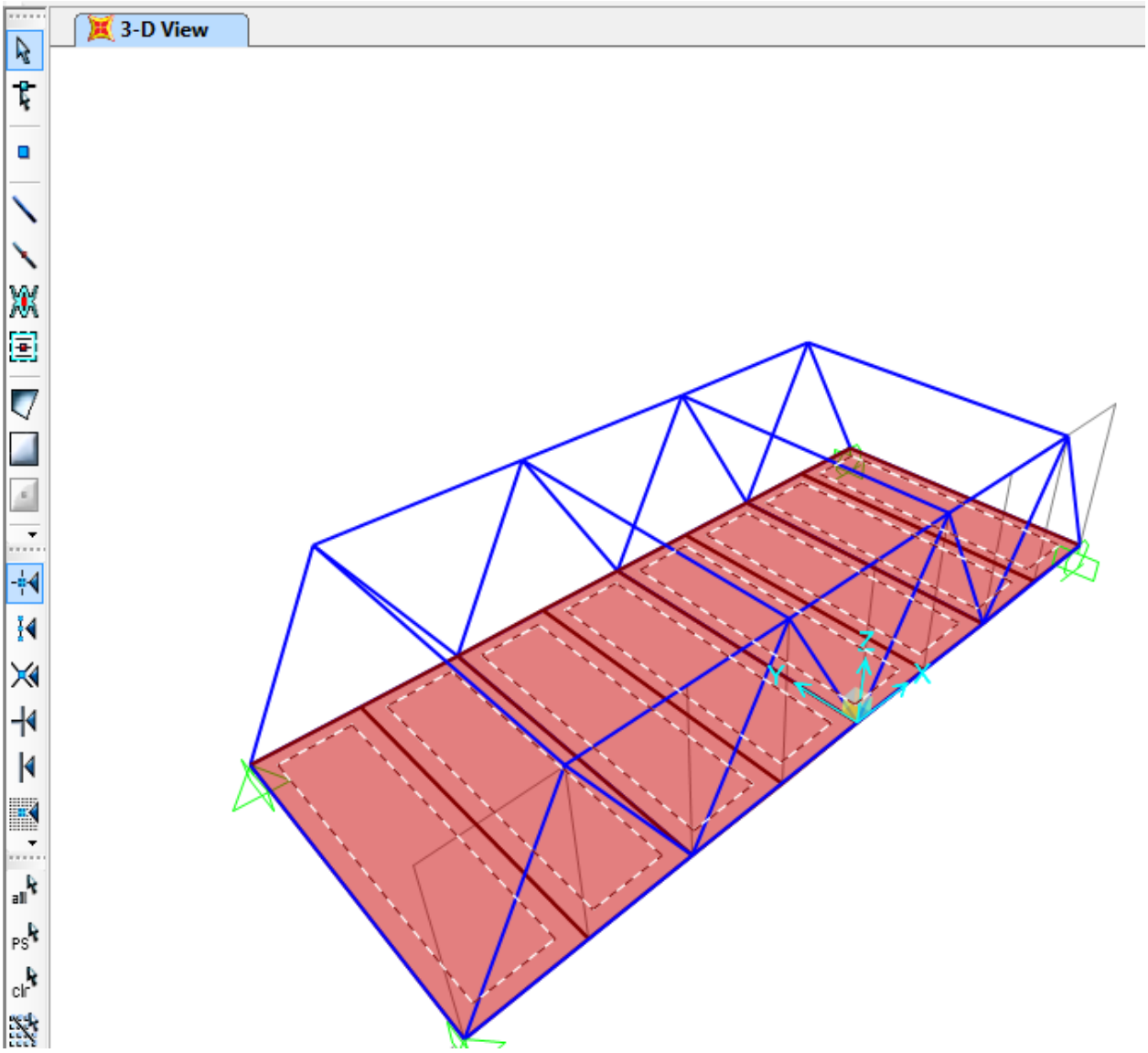
- Click anywhere on the area object to display the Object Model
- Go to Edit/Edit Area/Divide Area
- Select the Mesh Area Based On Points On Area Edges option.
-

The roadway (Area Object) is now meshed. (Click inside each box to check)

To better view the deck addition, click the set Display Options button , or go to the view menu > Set Display Option command (Or press Clrl+W). When the form appears, check Fill Objects box and Apply to All Windows check box as shown in bellow figure



- Click Ok



Step 5: Define Load Cases (Live Load)

The load used in this tutorial consists of dead and live static load acting in the gravity direction.

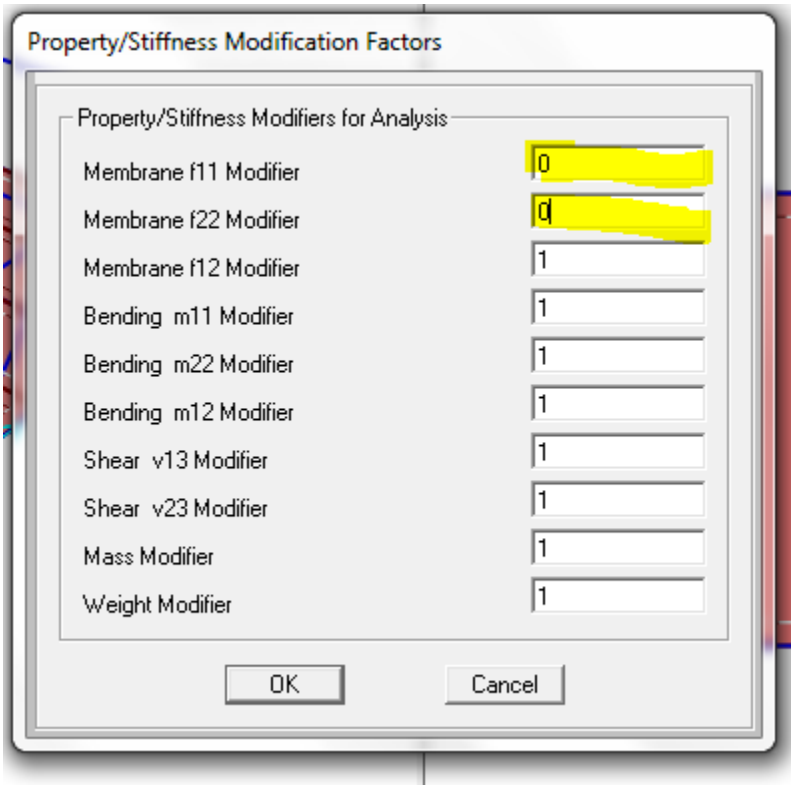
For this project, assume that dead consists of the self-weight of the bridge plus an additional live load **(**the weight trucks).

Dead load = the weight of the steel members + the weight of the roadway

Assign Area Stiffness Modifiers

In this step, the membrane properties of the Area object is modified to prohibit the deck from acting as flange for the bottom chords of the trusses. Make sure that X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

A- Click anywhere on the area object to select the deck.
B- Click the Assign menu > Area > Area Stifness Modifiers command to access the property/Stiffness Modification Factors from shown in Figue.
C- Type 0 in the Membrane f11 Modifier edit box
D- Type 0 in the Membrane f22 Modifier edit box



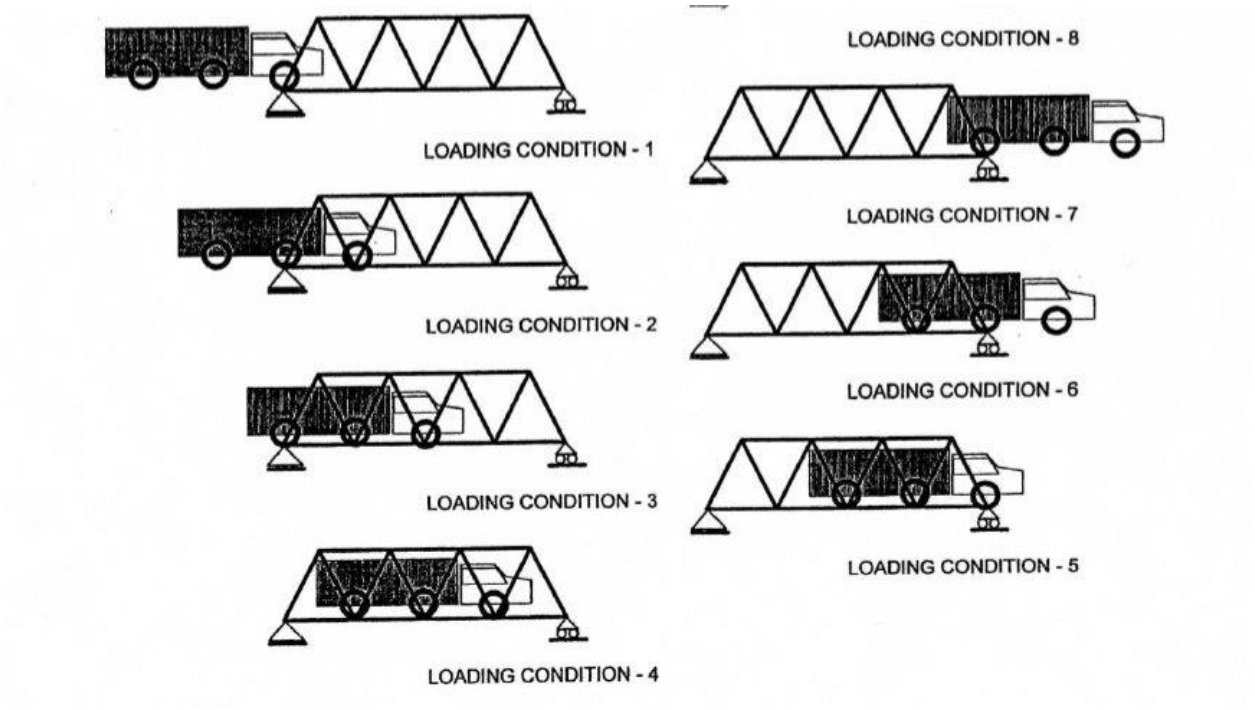
Assign Joints Load (Live load)

When the cars and trucks move across the bridge, they normally produce dynamic oscillations of the bridge structure. This dynamic oscillation will in turn produce forces in each member that are higher than the forces obtained if the loads are statically applied. These additional dynamic forces can be modeled using a dynamic amplification factor or impact factor. It is herein assumed that the **impact factor** is equal to 0.27 times the truck load. This dynamic amplification load should be added to the total truck load.

The impact factor is computed as a percentage increase in vehicle live load stress.

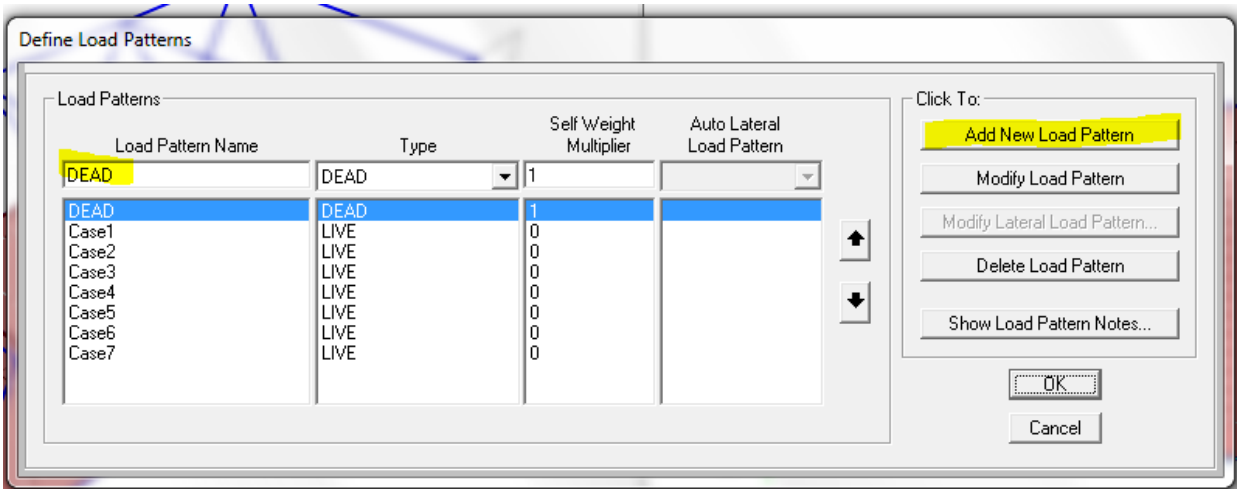
	Back Wheel	Middle	Front Wheel
Live load without addition of the Impact factor	32000 lb	32000 lb	8000 lb
Live load after addition of the Impact factor	?	?	?

Calculate the new loads after adding the impact factor

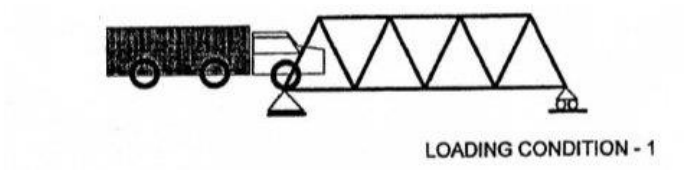


- In SAP2000 go to Define menu > load pattern
- Add the seven cases and make sure you choose Live for load type and leave Self Weight Multiplier set as 0 for the live loads.

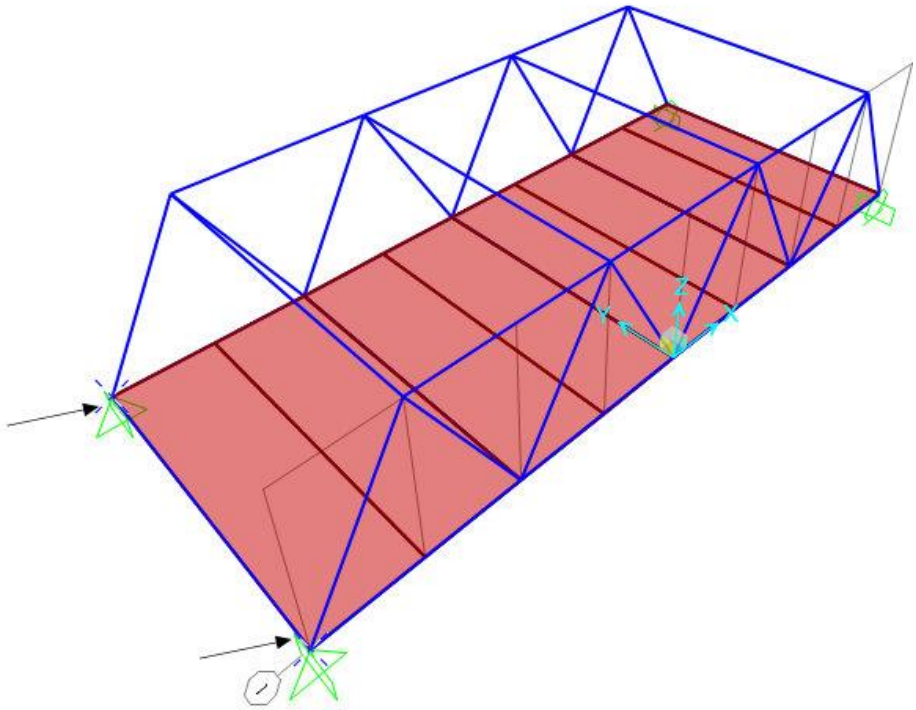
Note that Self Weight Multiplier is set to 1 for default case (DEAD). This indicates that this load case will automatically include 1.0 times the self-weight of all members.



Case 1:



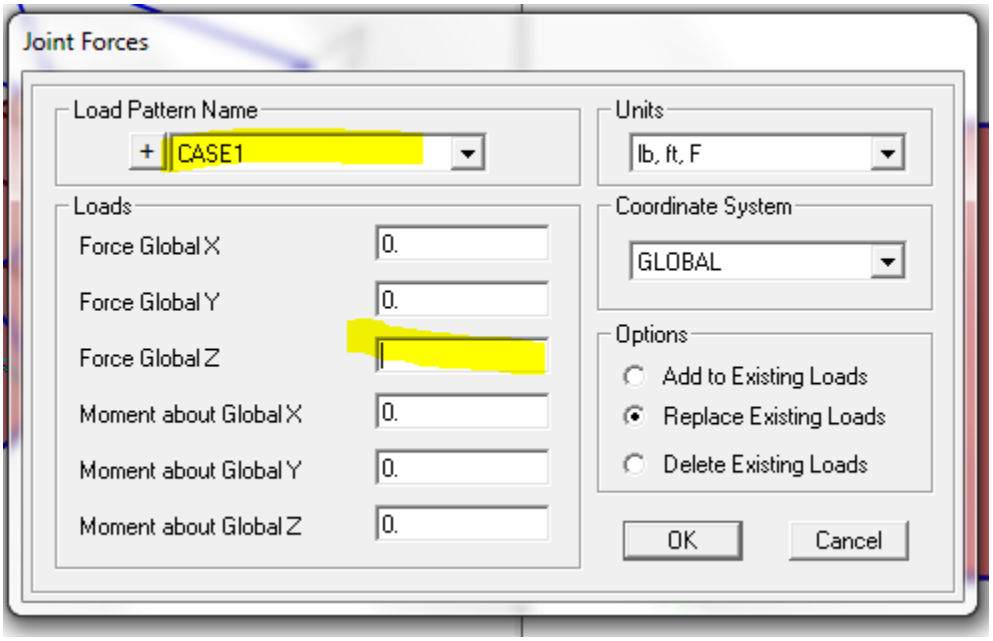
- Click the pointe tool icon . Select the first two lower cord joints



Note: It is herein assumed that the live load applied on the bridge is divided equally to each truss.

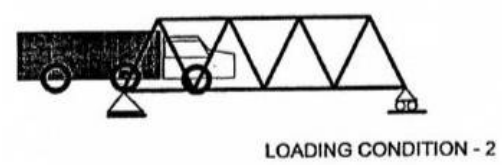
- Go to Assign > Joint Loads > Forces
- Select CASE1 for Load Pattern Name
- Add the value of the front wheel load in the Load edit box

Remember that the Gravity Direction is in the negative Global Z direction



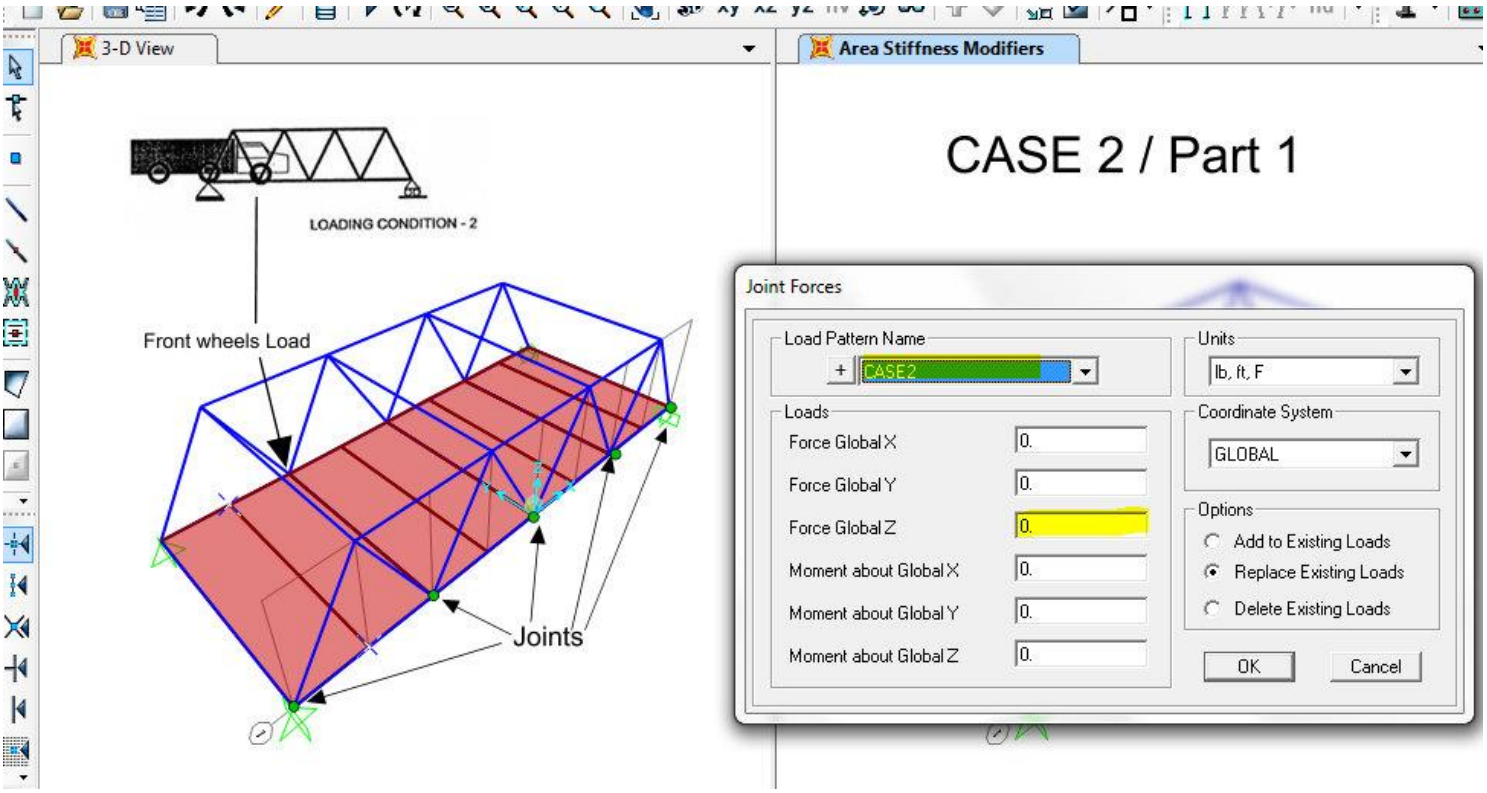
- Click OK

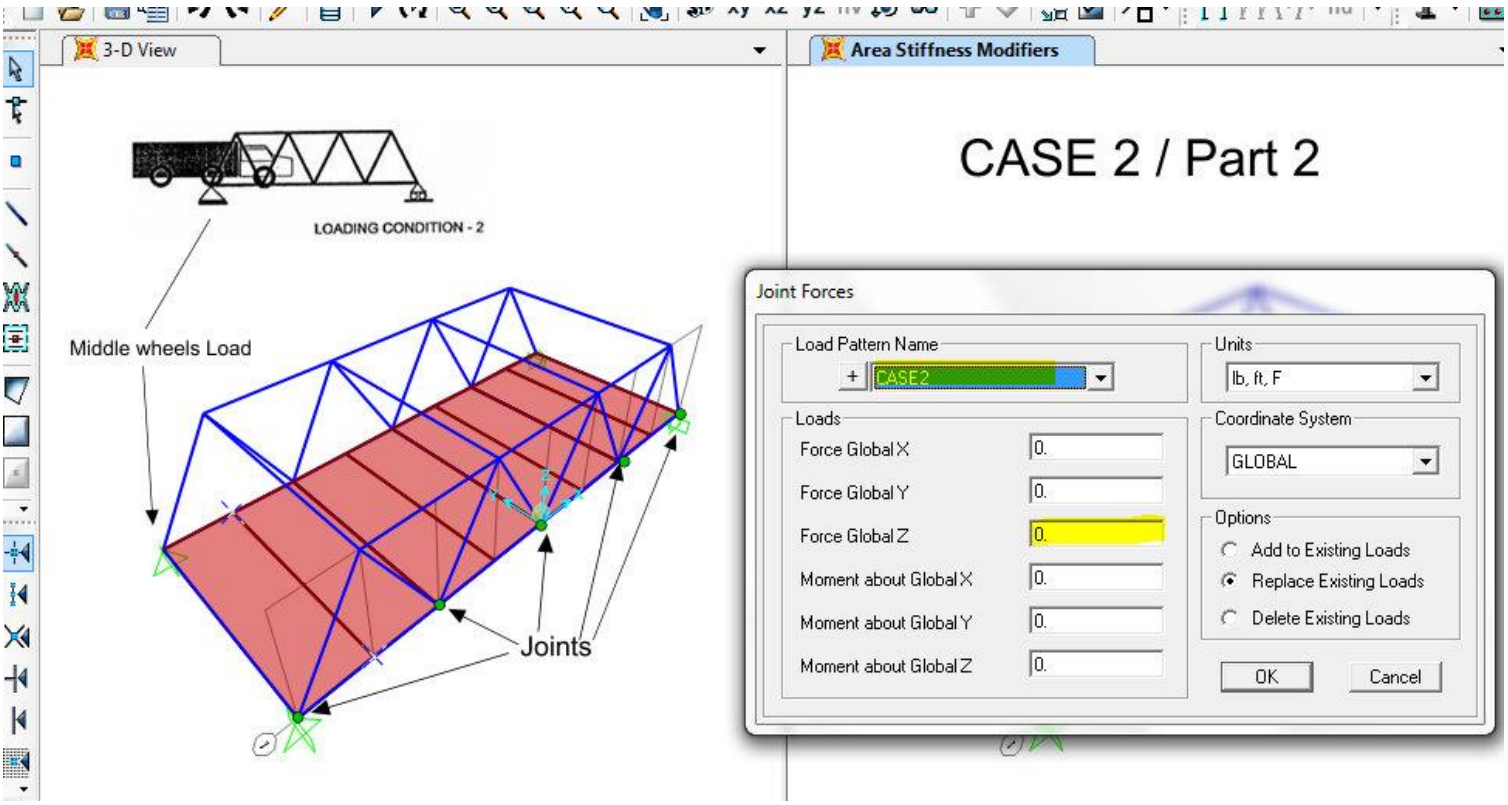
Case 2:



Now we'll apply load in four joints (front wheels load and middle wheels load)

- Select the second two joints only, then go to Assign > Joint Loads > Forces
- This time select CASE2 for Load Pattern Name
- Add the value of front wheel load, then press OK
- Next select the first two joints only, then go to Assign > Joint Loads > Forces
- Make sure you are selecting the same Load Pattern Name CASE2
- Add the value of the middle wheel load in the Load edit box, Then press OK

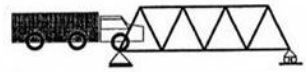




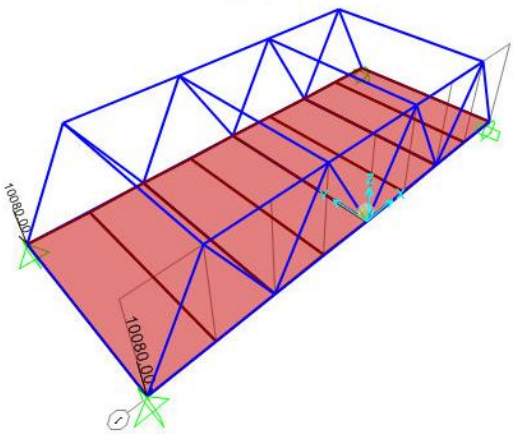
Continue with the other cases by following the same steps. Make sure you choose the right CASE from Load Pattern Name before assigning the loads

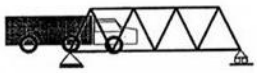
You can display the Load Pattern by going to Display > Show Load Assigns > Joint

The

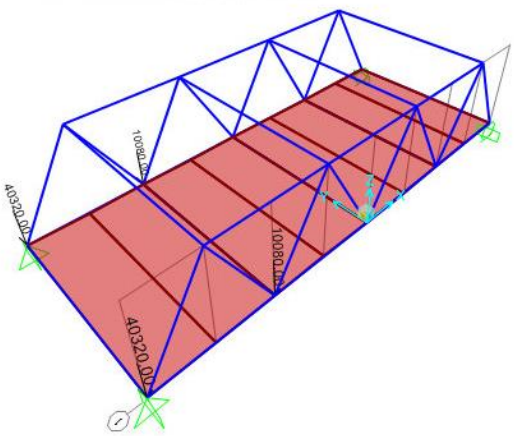


LOADING CONDITION - 1

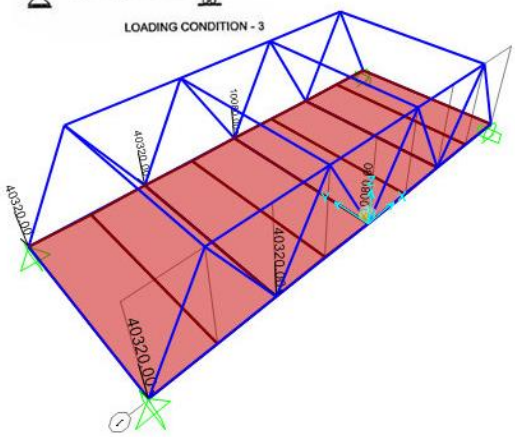




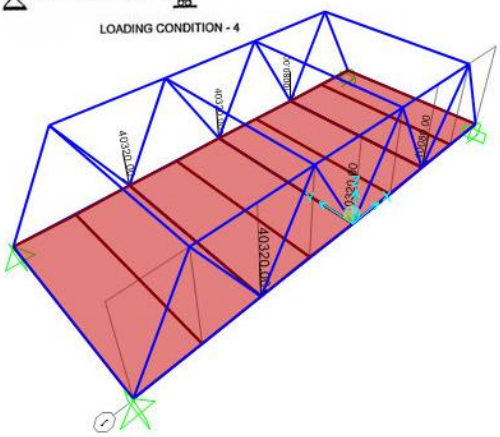
LOADING CONDITION - 2



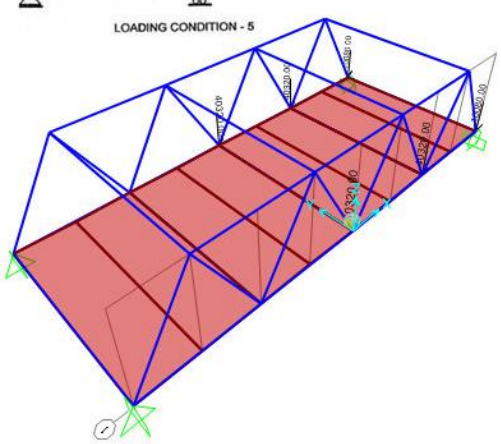
LOADING CONDITION - 3



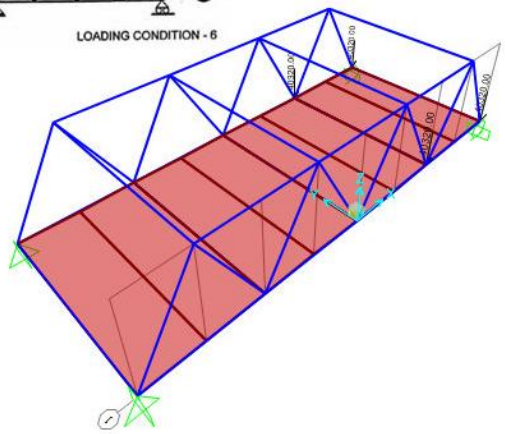
LOADING CONDITION - 4

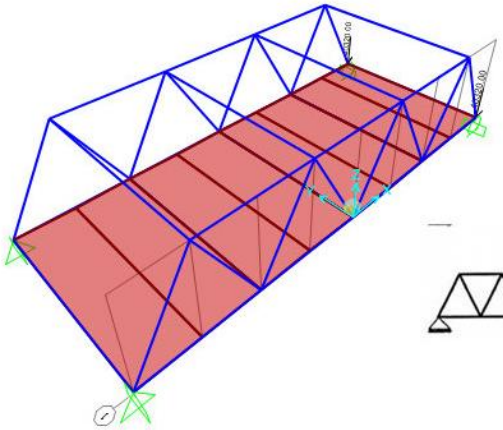


LOADING CONDITION - 5



LOADING CONDITION - 6



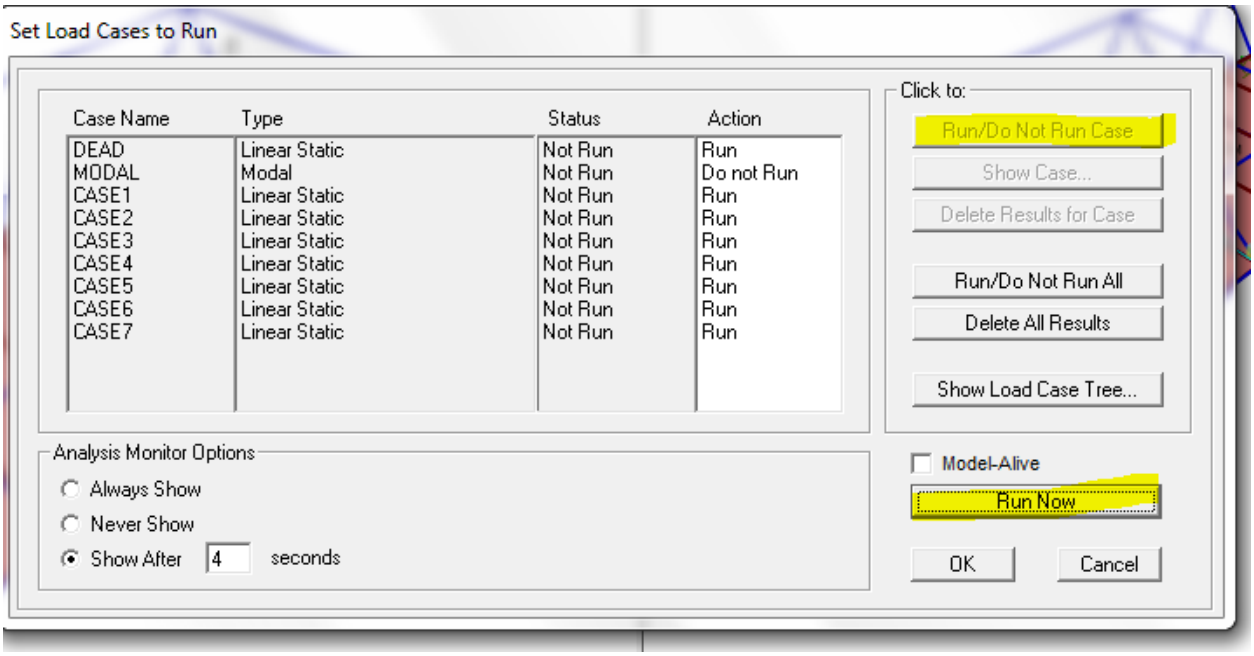


LOADING CONDITION - 8

Step 6: Run the Analysis

In this Step, the analysis model will be viewed and the analysis will be run

Click the Analyze menu > Run Analysis command or the Run Analysis  to access the Set Analysis Cases to Run form as shown bellow



- Select MODAL from the Case Name list.
- Click the Run/do Not Run Case button to set the action for *MODAL* to *Do Not Run*, as we intend to run only a static analysis
- Click run Now button

The program will create the analysis model from the object-based SAP2000 model, an will soon display an analysis window. Data will scroll in this window as the program run the analysis. This information may be accessed at a later time by going to the File menu > Show Input/Output Text Files command and selection the file with .Log extension

Step 10: Graphically Review the Analysis Results

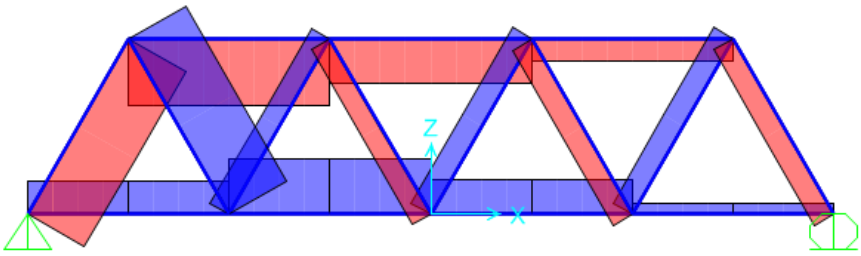
In this Step, the analysis results will be reviewed using graphical presentation of the results.

- Select the XZ plan 

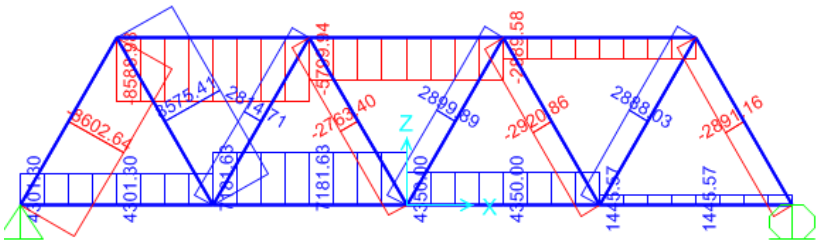
- Click the show Forces/Stresses > Frames/Cables button  , or the Display menu > show Forces/Stresses > Frames Cables command to bring up the Member Force Diagram for Frames (You can also press F8 for a quick access)
- Select CASE2 / Combo Name Drop-down list
- Select the Axial Force option.
- Select the Auto option under the Scaling area
- Select the Fill Diagram option
- Click OK button to generate the axial force diagram shown in figure

Fill Diagram Option:

- ✓ Blue: Tension
- ✓ Red: Compression



Show Values on Diagram option

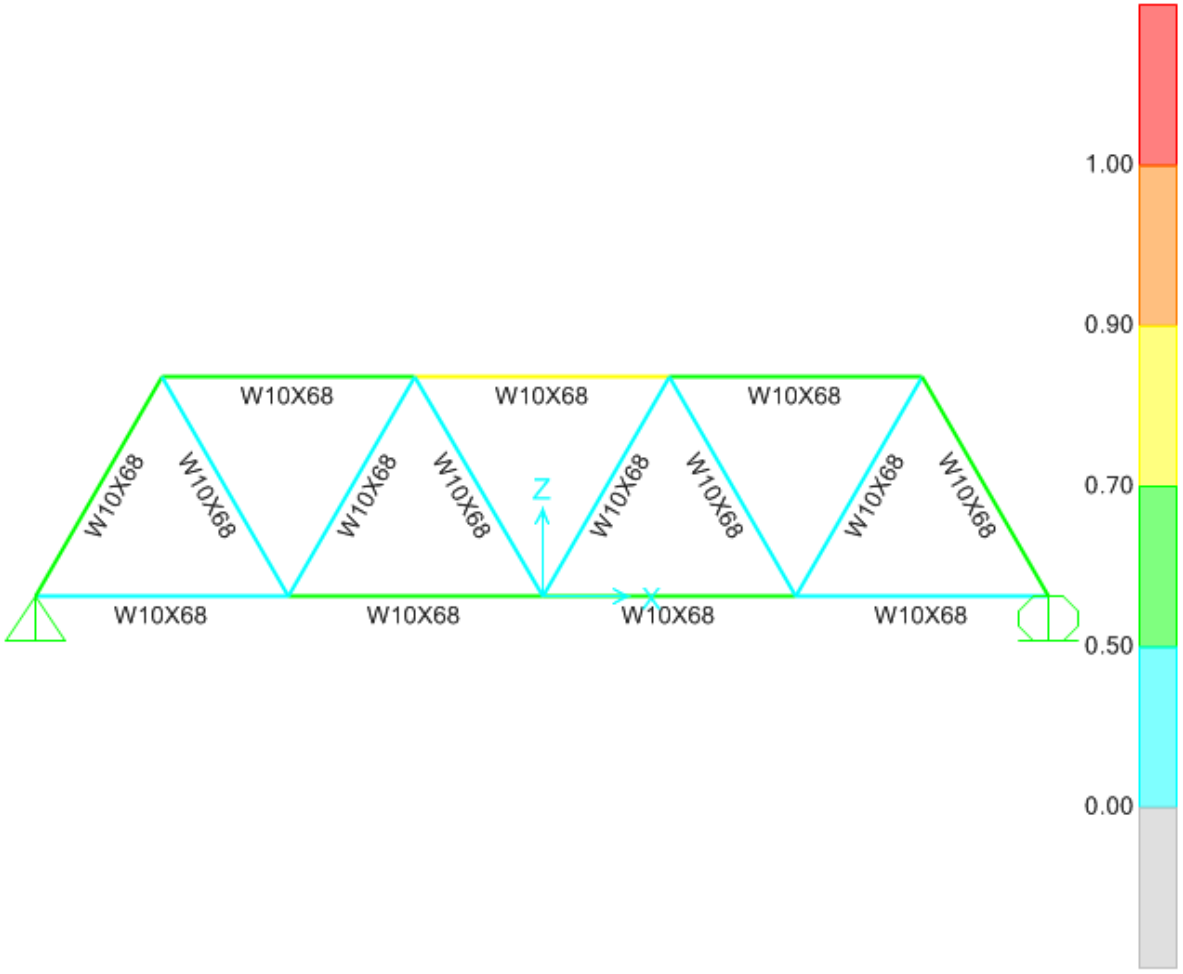


By Ait Elmouden Abdellah | LaGuardia Community College 31-10 Thomson Ave. Long Island City, NY 11101

Designing the structural members and checking the safety of a design.

In This Step, the steel frame members of the trusses will be designed.
Not that the analysis should be run before completing the following action items

- Click the Design menu > Steel Frame Design > Start Design/Check of Structure command or the Start Steel Design/Check of Structure **button, to start the steel frame process.**



When the design is complete, the selected sizes and stress ratios are displayed on the model. The goal is to repeat the analysis and design process until the analysis and design sections for the truss are all the same.

Note that when the bridge is reanalyzed, SAP2000 will use the current design sections, as new analysis sections for the next analysis run. You will need to optimize you design by changing the cross section area of the frame steel, by following the same procedure in **Step 2**

Notes for the design and analysis of the Truss Bridge

Part A –Procedure:

Create an EXCEL table for listing all the members. EXAMPLE

Member#	COMBO. FORCE (DSLTL4)	$A\_min = (\frac{FORCE}{\sigma_{ALL}})$	Optimum $A_{OP} =$ A_min + i
1			
2			
3			
…			

Note:DSTL4 $= \left(\text{All cases} \quad\right) + \text{Dead Load. To Select DSTL4}:$

- Click the show Forces/Stresses > Frames/Cables button , or the Display menu > show Forces/Stresses > Frames Cables command to bring up the Member Force Diagram for Frames (You can also press F8 for a quick access)

- Consider the type of force (tension or Compression to select your i (Cross section area increment).
- Go to the Steel Sections website (see http://www.structural-drafting-net-expert.com/steel-beam.html) and you the A_{op} value to find the cross section Designation (e.g. W10x68 for $20 in^2 Cross\ Section$).
- Select the member that you want to change the cross section then follow step 3 in the tutorial to assign it.
- Check the safety of a design again and make sure your member is optimum in terms of cost and not overstressed (green Color):

  Click the Design menu > Steel Frame Design > Start Design/Check of Structure command or the Start Steel Design/Check of Structure **button, to start the steel frame process.**

Part B – Cost analysis

Steel cost $560 per ton

Concrete cost $50 per cubic yard

Calculate only the material cost for the steel members and roadway (neglect connections and

Labor costs)

Part C – Group Report - (format similar to lab reports). Include all concepts and

Cost analysis: Use the excel file provided to calculate the cost

Part D – Prototype building and testing (bridge model). Material will be provided