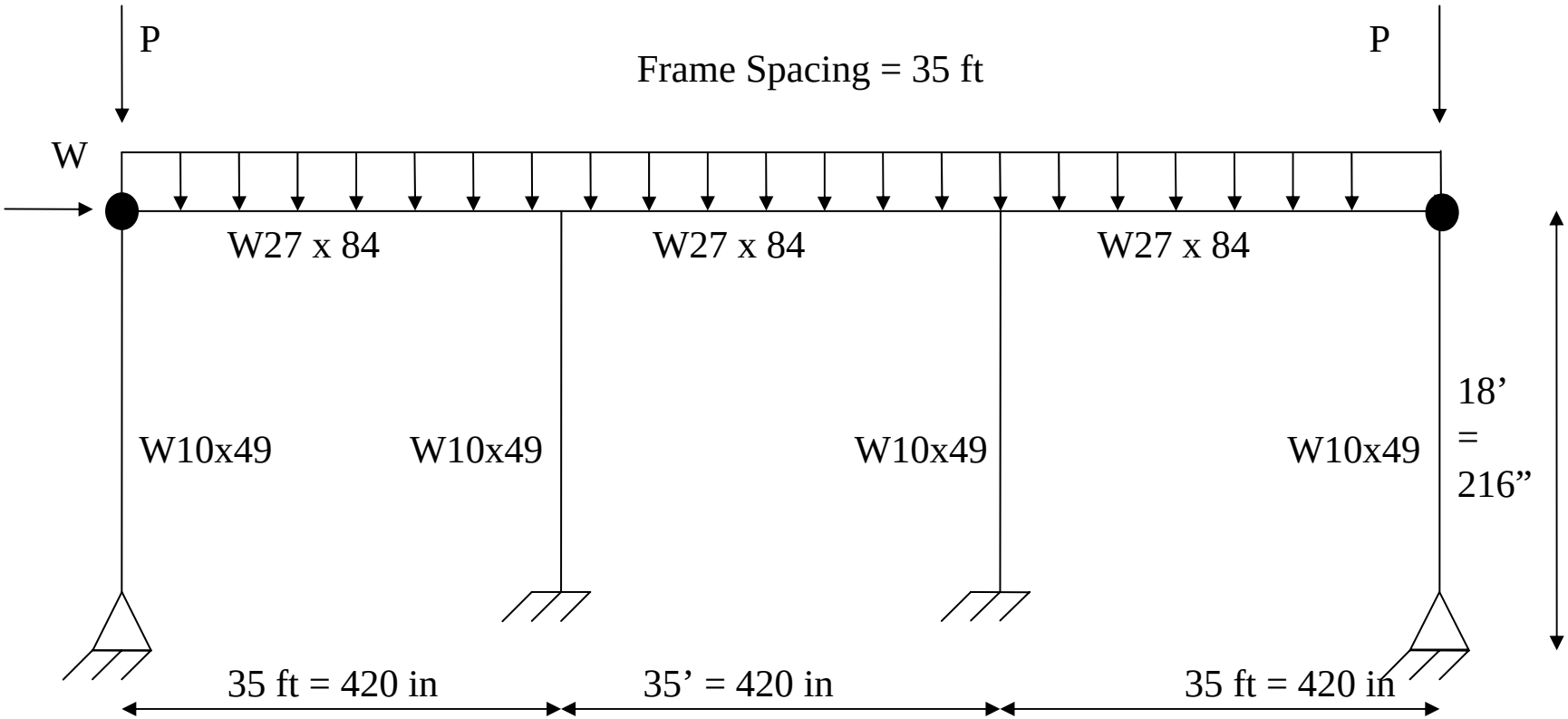
Tutorial to SAP2000 Student Edition

by Keith Kowalkowski

for CE470, Design of Metal Structures

September 15, 2005

FRAME AND LOADING CONDITIONS



Exterior Loads: P = 168.4 kips

Wind: W = 8.19 kips

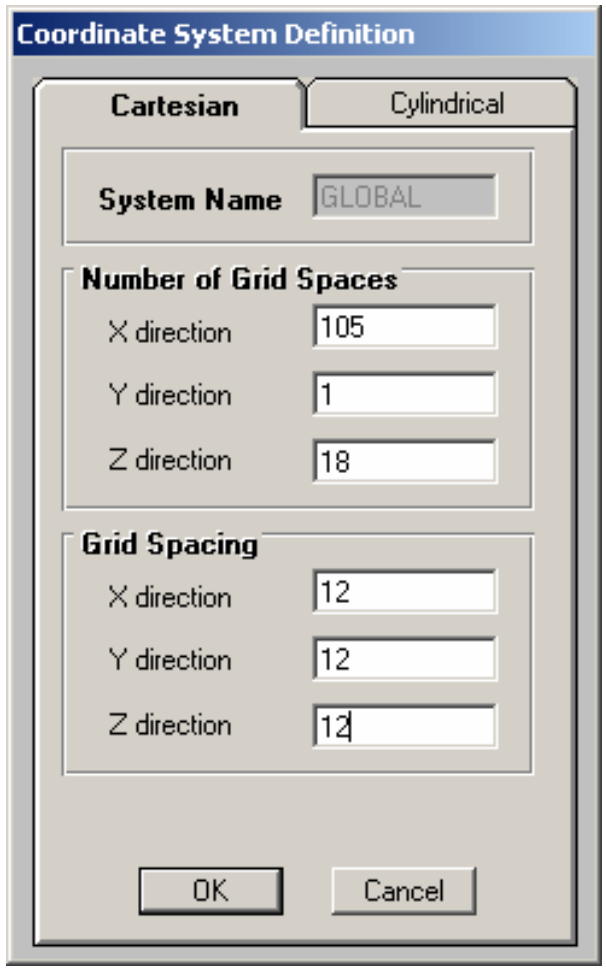
Distributed Load: w_u = (1.2*0.233333 + 0.5*0.1166667) = 0.33833 kips/in

Step 1. Open Program

- Different Versions of Sap2000 vary differently.
- On, ECN computers on the third floor.
- Click “Start-Programs-CE Software-SAP2000 Nonlinear-SAP2000 Nonlinear”.

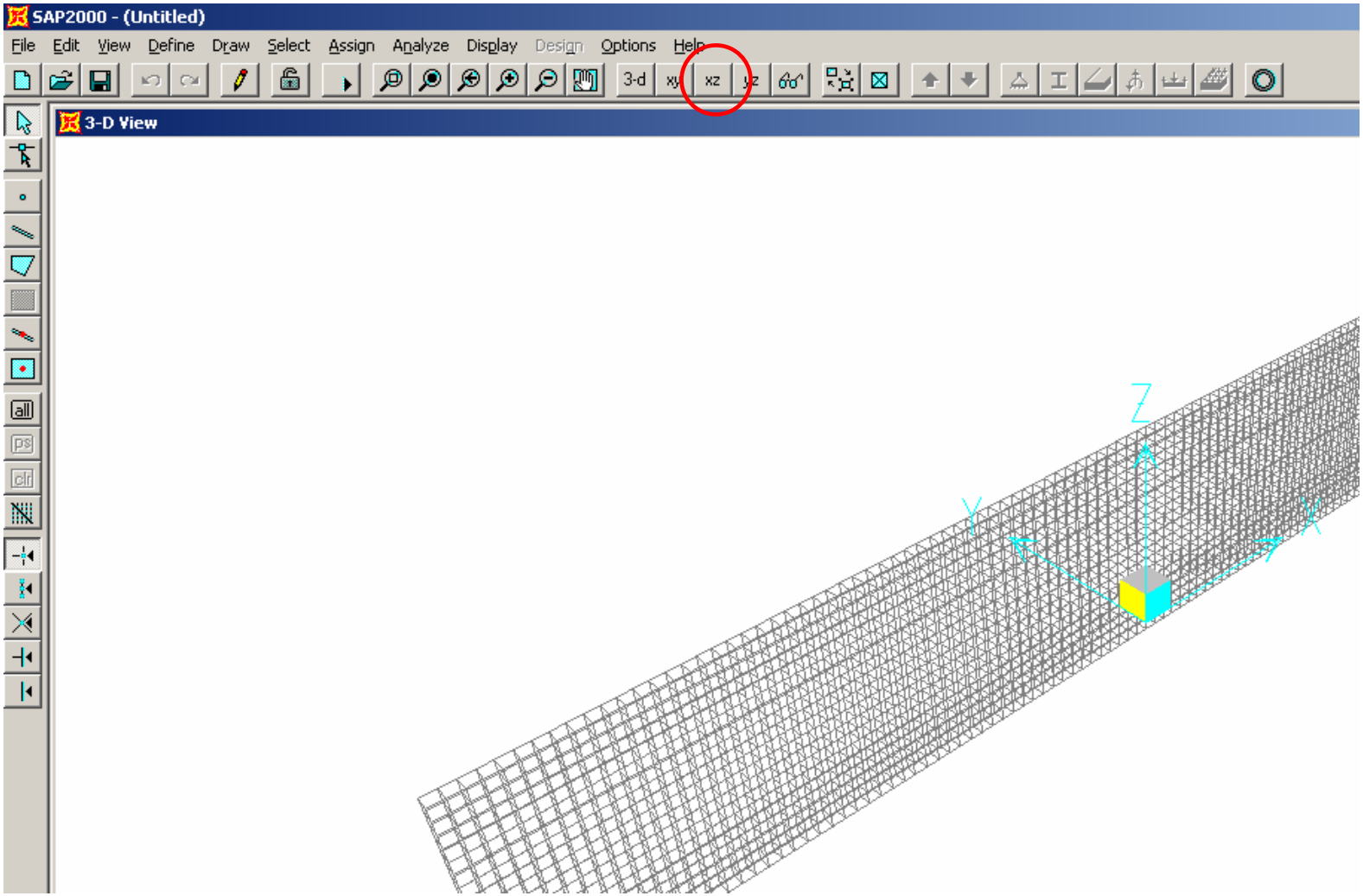
Step 2 Getting Started (Student Edition)

- Click: “File-New”
- 3-35’ spans = 105 ft
- I choose to have grid marks in feet and making each 12 units to have units of inches



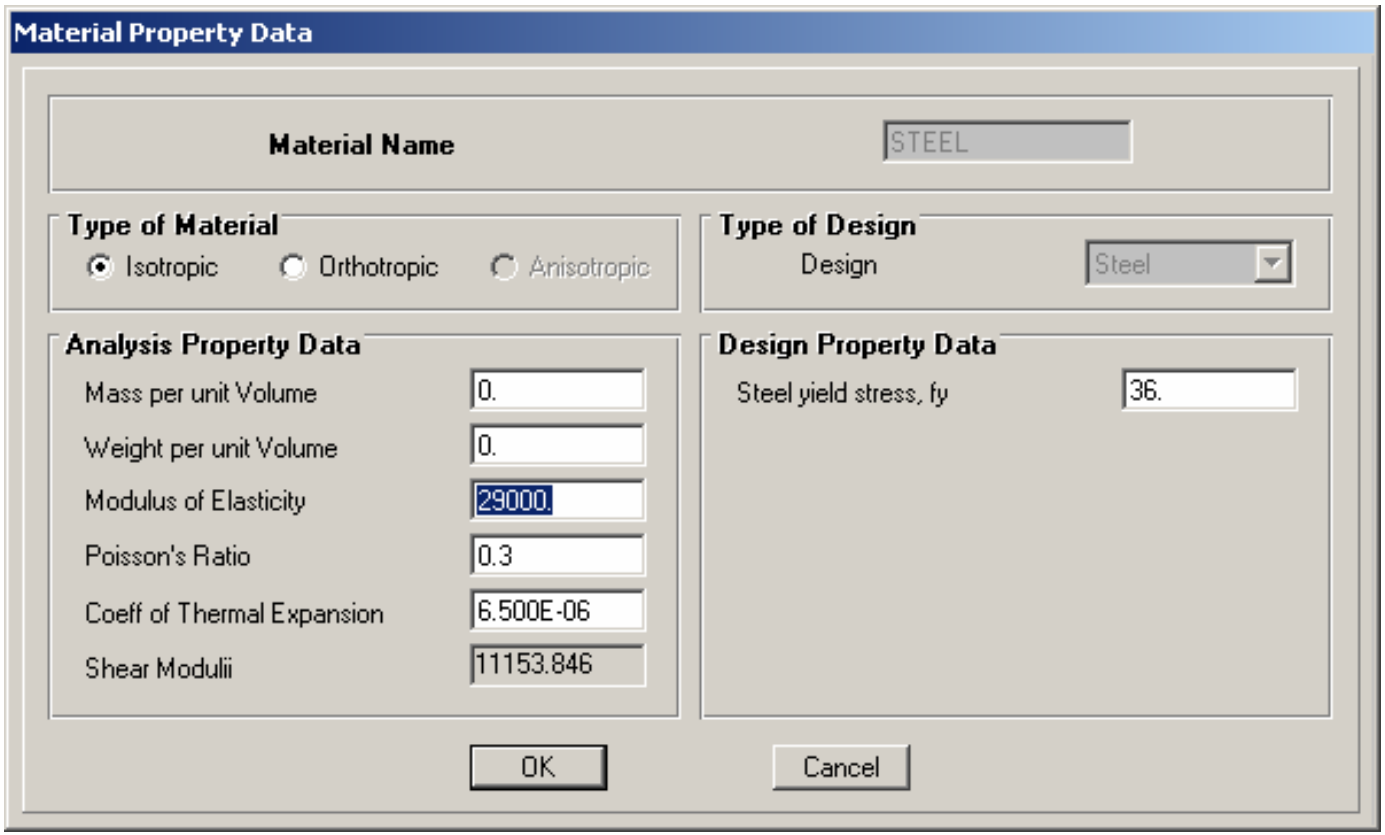
Step 2: Getting Started

- Click on the xz tab on the top



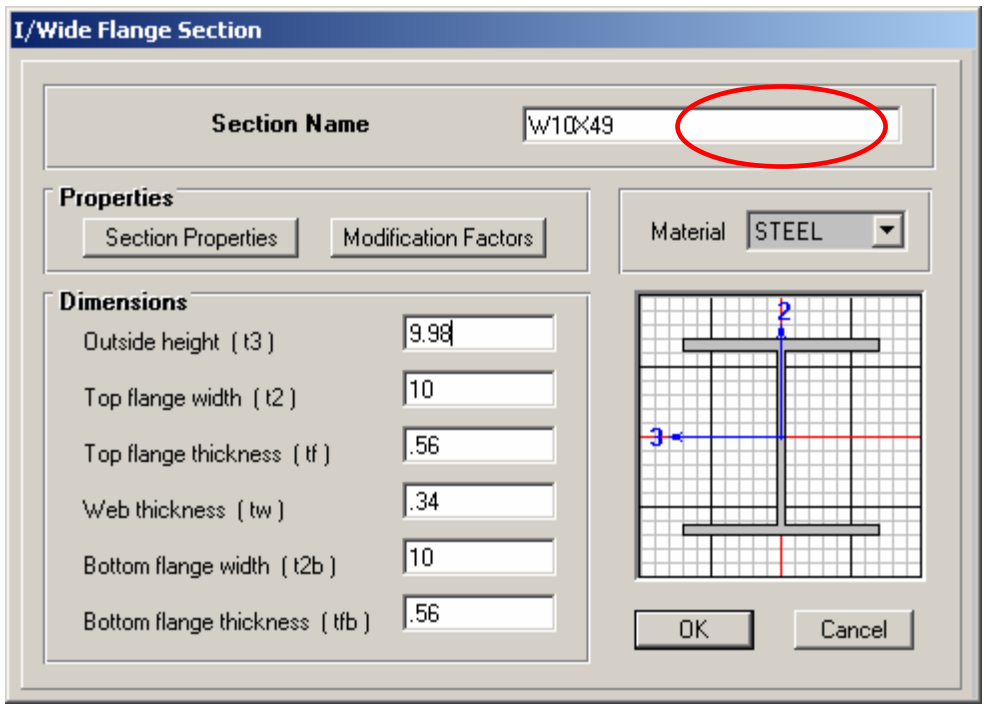
Step 3. Define Materials and Sections

- Click “Define-Materials” at the top
- Click on the material “steel”. It should be listed and then click on “Modify/Show” Material
- Type in the following properties for steel, we don’t want to consider dead load separately



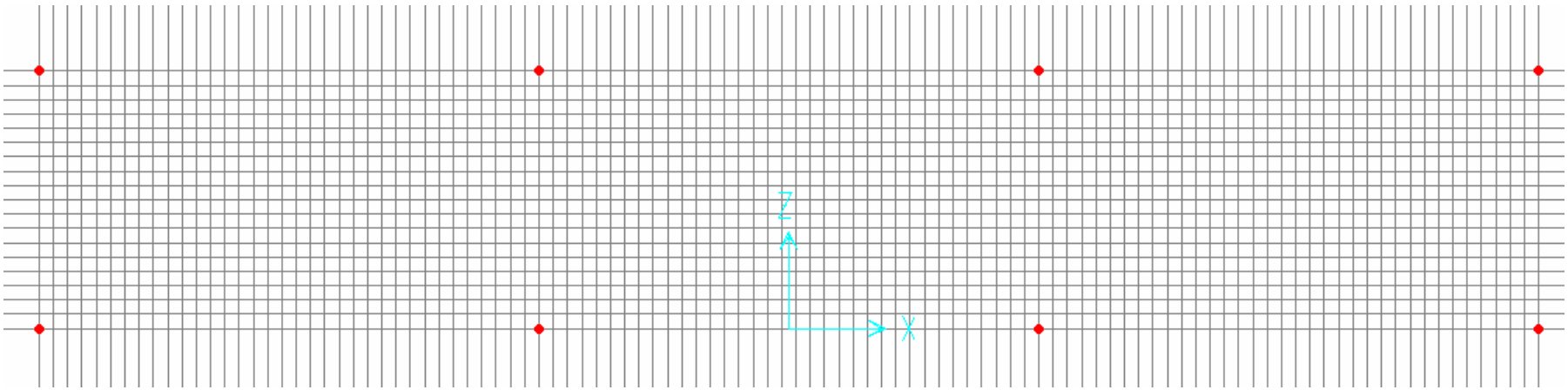
Step 3. Define Materials and Sections

- Click “Define-Sections” at the top.
- Click to: “Add I/Wide Flange”.
- Give the section a name and input the properties from the AISC manual.
- Make sure the proper material is listed (STEEL).
- Define all sections for the model



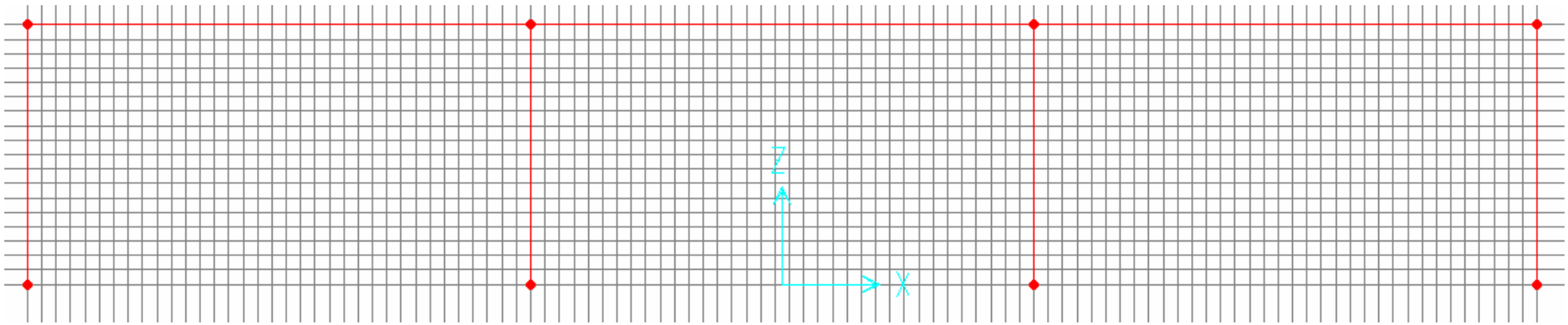
Step 4. Define Nodes and Elements

- Click “Draw-Special Joint”. This is a *node*.
- Add the nodes to the appropriate locations.
- See below.



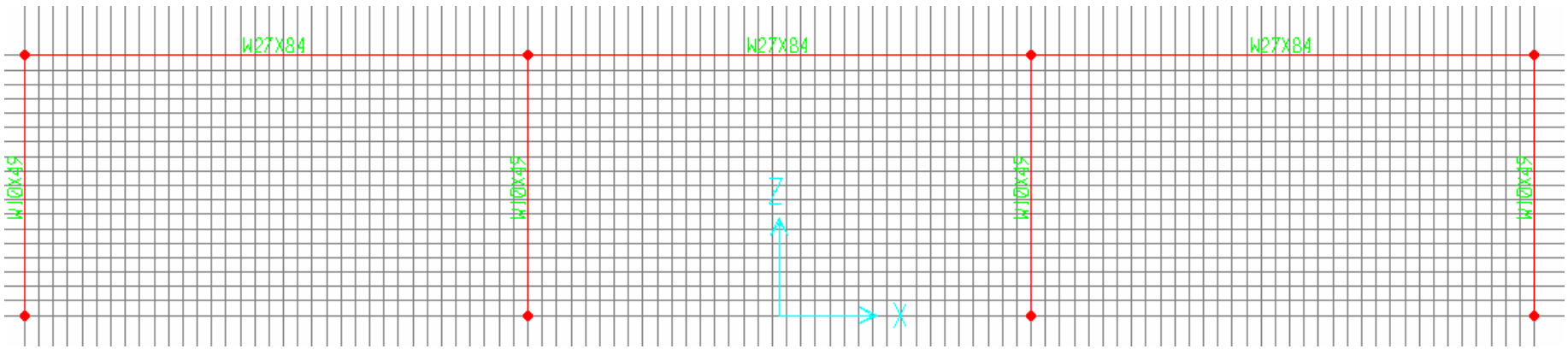
Step 4. Define Nodes and Elements

- Click “Draw-Draw Frame Element”.
- Insert the elements by connecting the nodes.



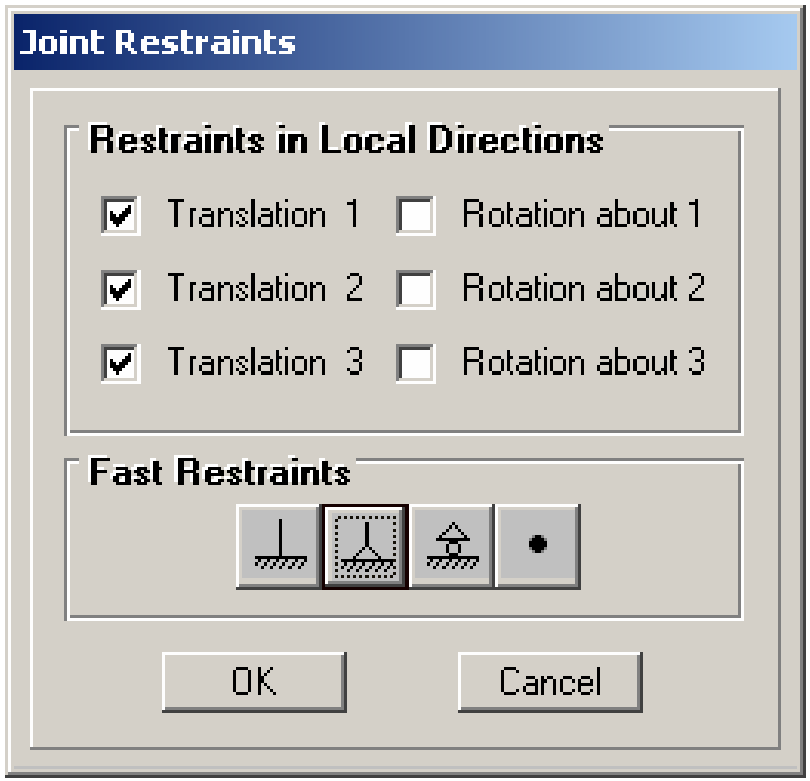
Step 5. Assigning Sections (For this model)

- Click on (highlight) the three beams
- Click “Assign-Frame-Sections”
- Click on “W27x84” and “ok”
- Click on (highlight) the four columns
- Click “Assign-Frame-Sections”
- Click on “W10x49” and “ok”
- You should see as below



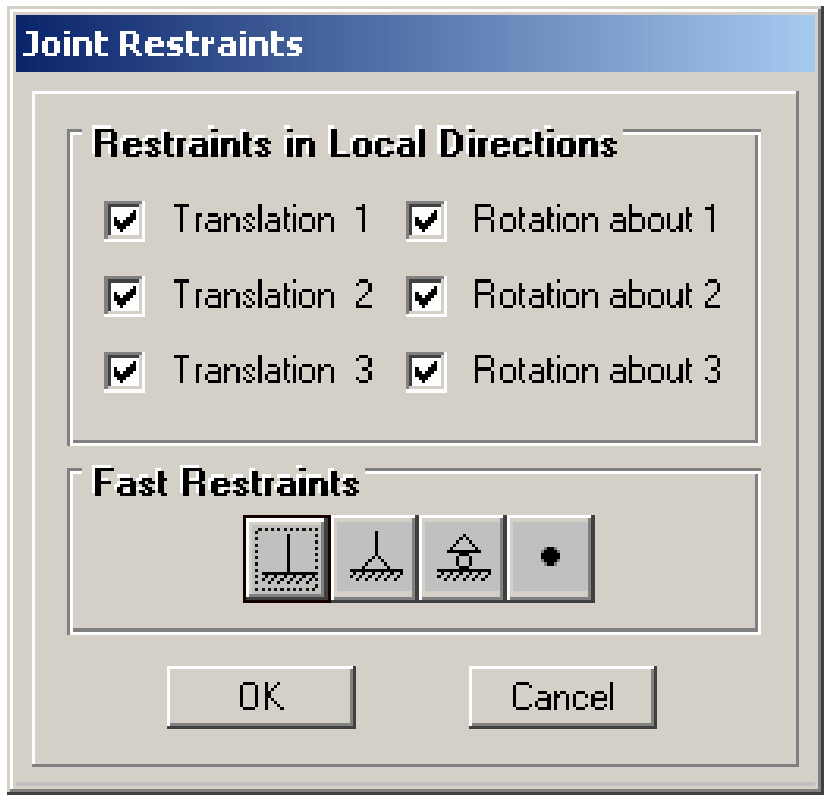
Step 6. Define Restraints

- Click on the two end nodes on the bottom (pinned nodes). Make sure they are highlighted.
- Click on “Assign-Joint-Restraints”
- Click on the second tab so the three translations are highlighted.
- Click “OK”



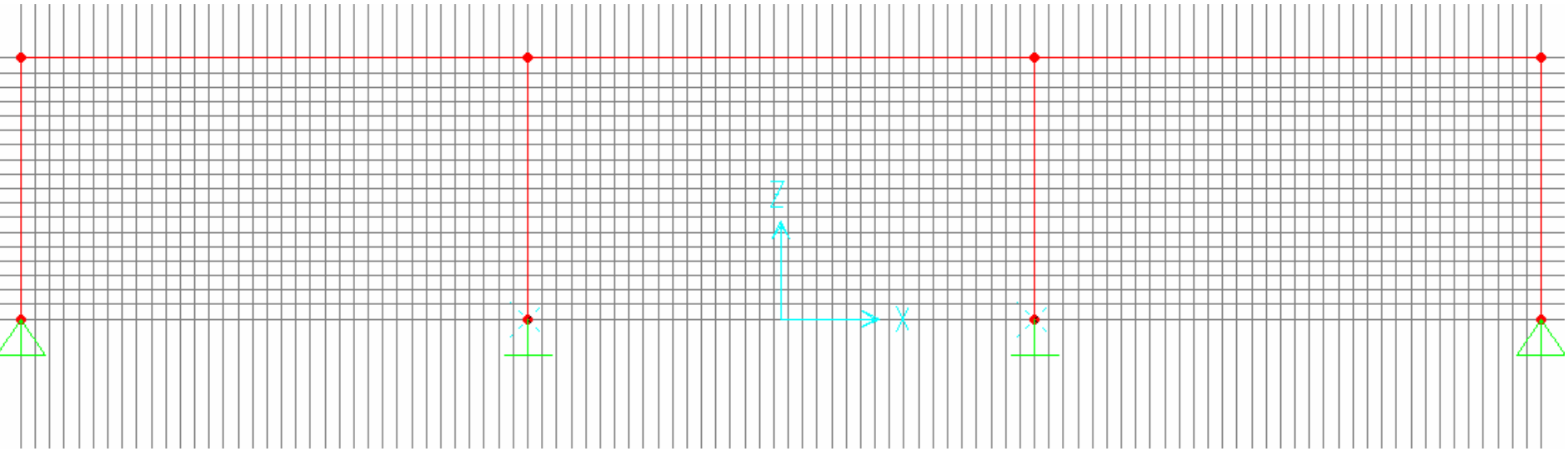
Step 6. Define Restraints

- Click on the two center nodes on the bottom. Make sure they are highlighted
- Click on “Assign-Joint-Restraints”
- Click on the first tab so all translations and rotations are highlighted.
- Click “OK”



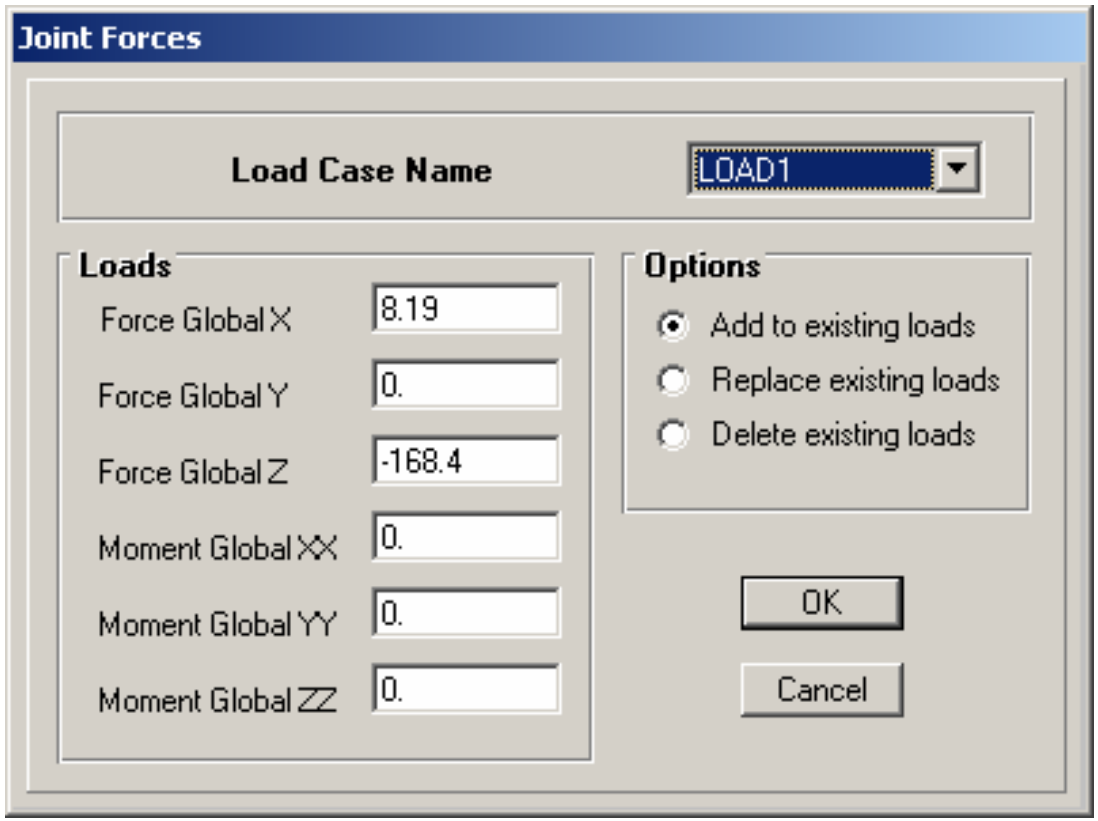
Step 6. Define Restraints

- You should see restraints as below



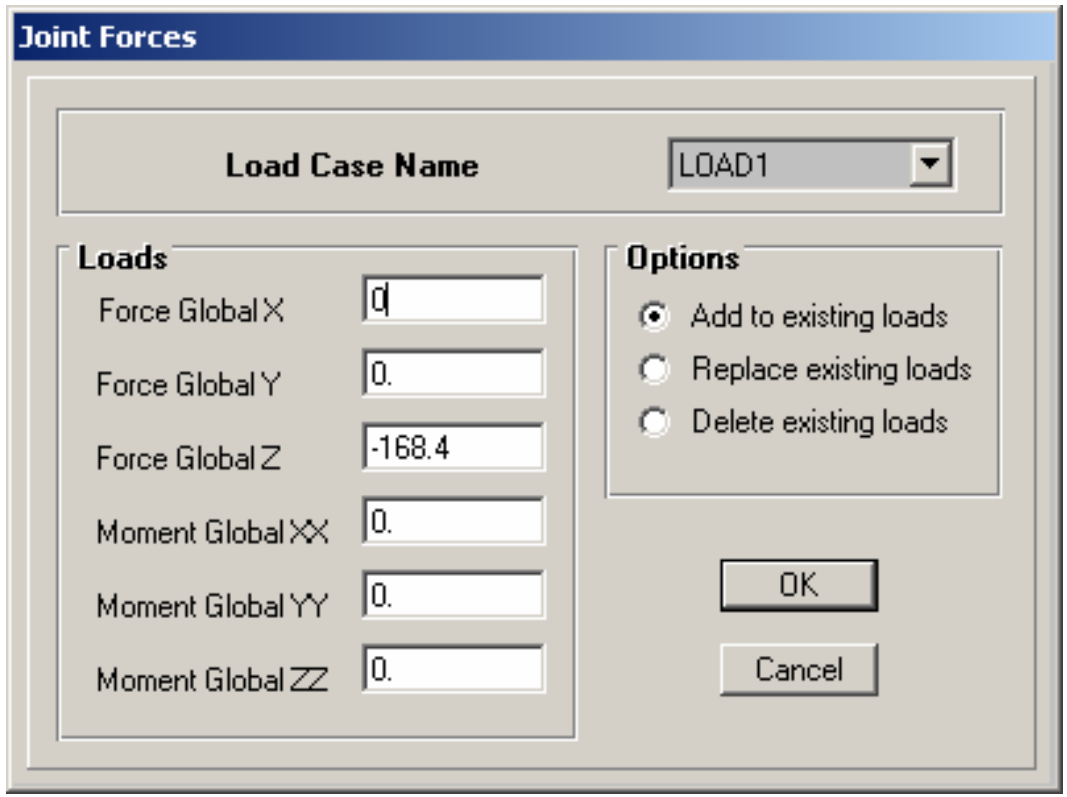
Step 7. Assign Loads

- Click on the top left node.
- Click on “Assign-Joint Static Loads-Forces”
- Enter the proper loads as shown below.
- Click “OK”



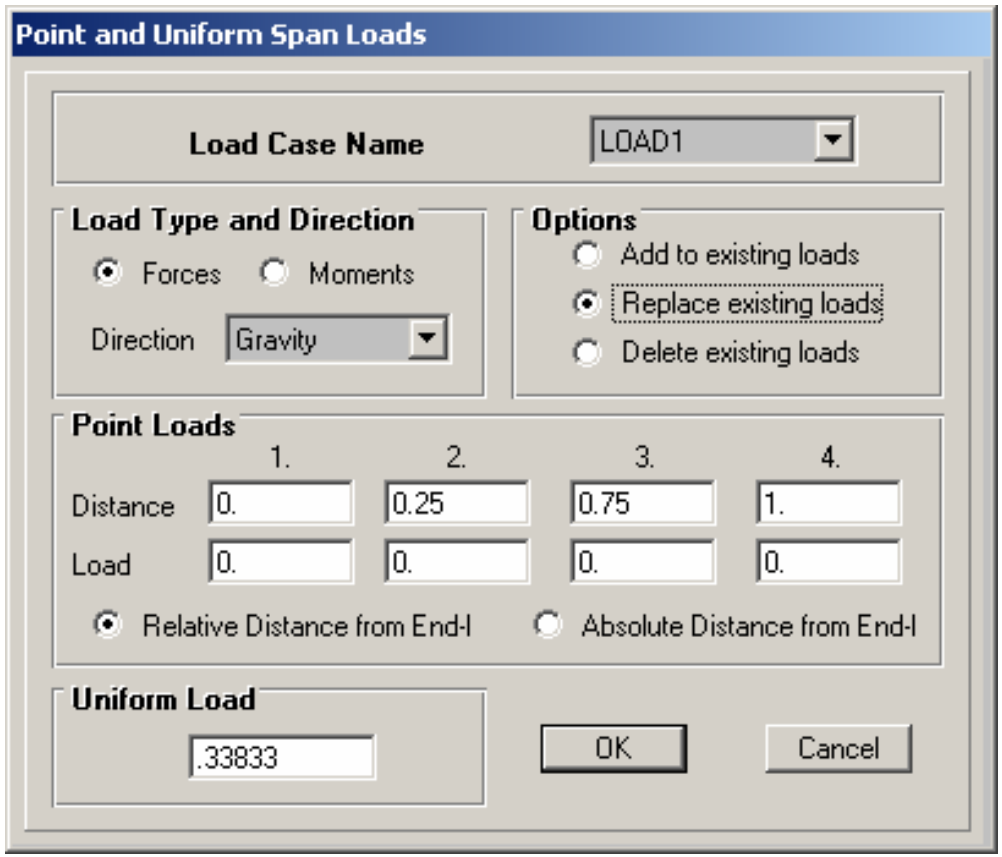
Step 7. Assign Loads

- Click on the top right node (make sure it is the only one clicked).
- Click on “Assign-Joint Static Loads-Forces”
- Enter the proper load as shown below.
- Click “OK”



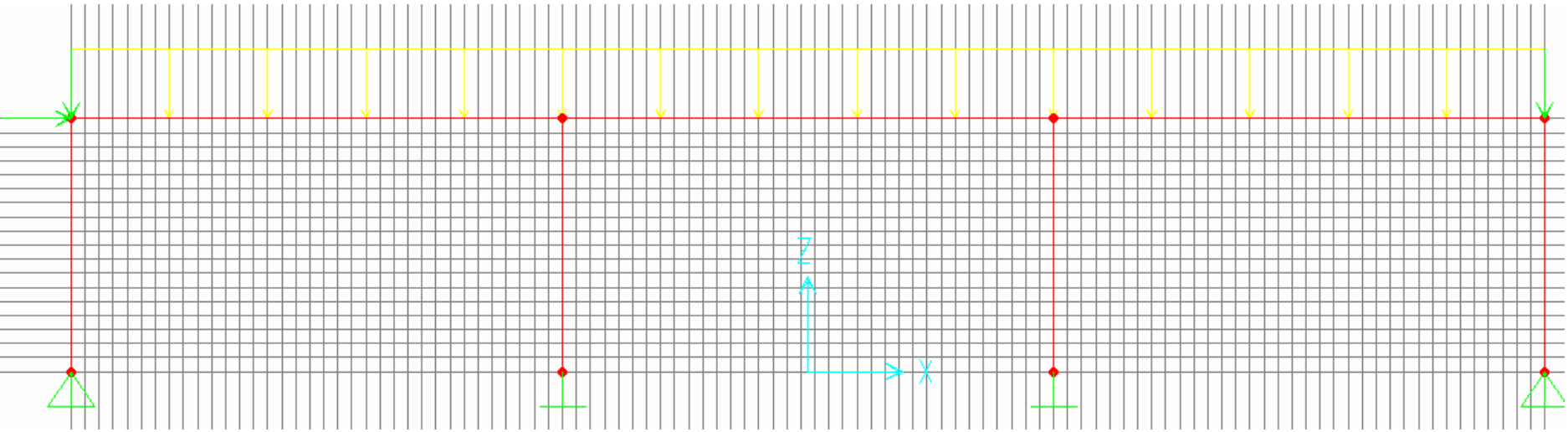
Step 7. Assign Loads

- Click on all four top beam elements.
- Click on “Assign-Frame Static Loads-Point and Uniform”
- Enter the proper load as shown below.
- Click “OK”



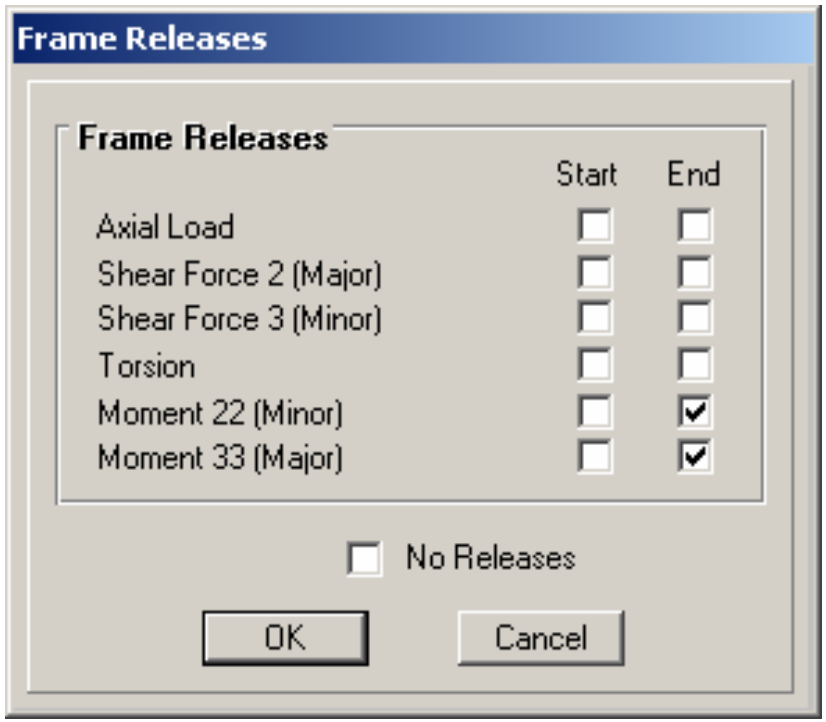
Step 7. Assign Loads

- You should see as below.
- Note they may be difficult to see on your printout.



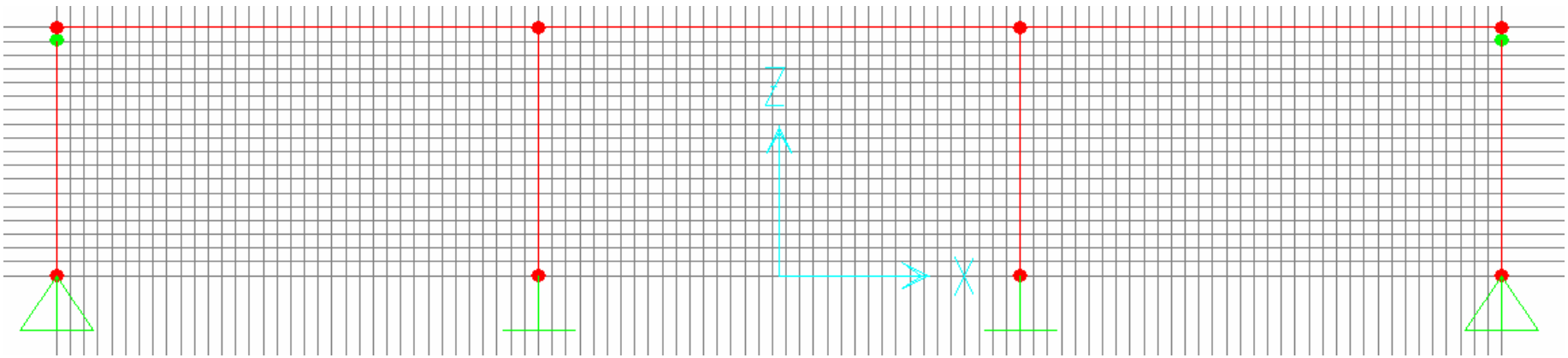
Step 8. Making Pin Connections

- Click on the elements you want to release (The two end columns or end beams will work, you want to make the connection *pinned*). You will need to do this for truss members in roofs.
- Click on Assign-Frame-Releases. Release the “Start nodes or End nodes as required”. It depends on which node you click first for the element.



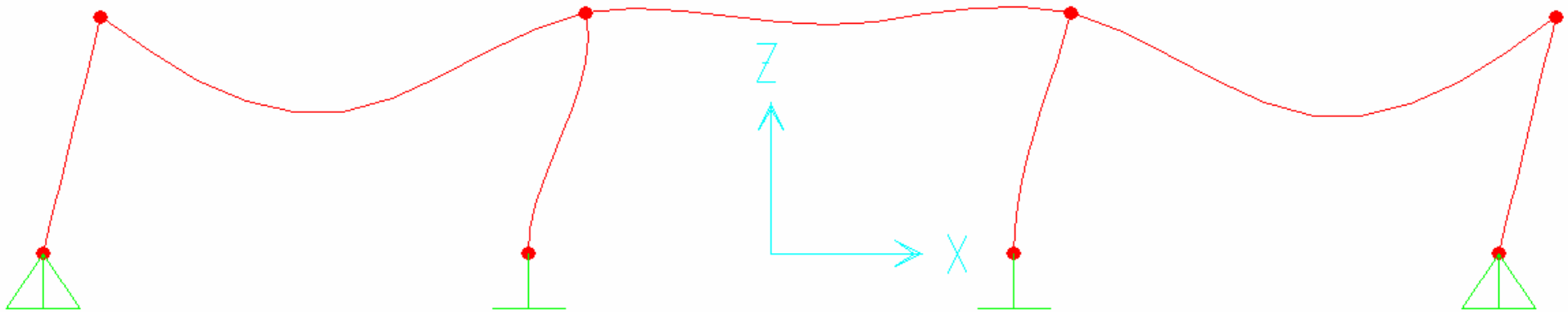
Step 8. Making Pin Connections

- You should see as below. However, if you make a mistake you can always Click on “Edit-Undo”.



Step 9. Run analysis

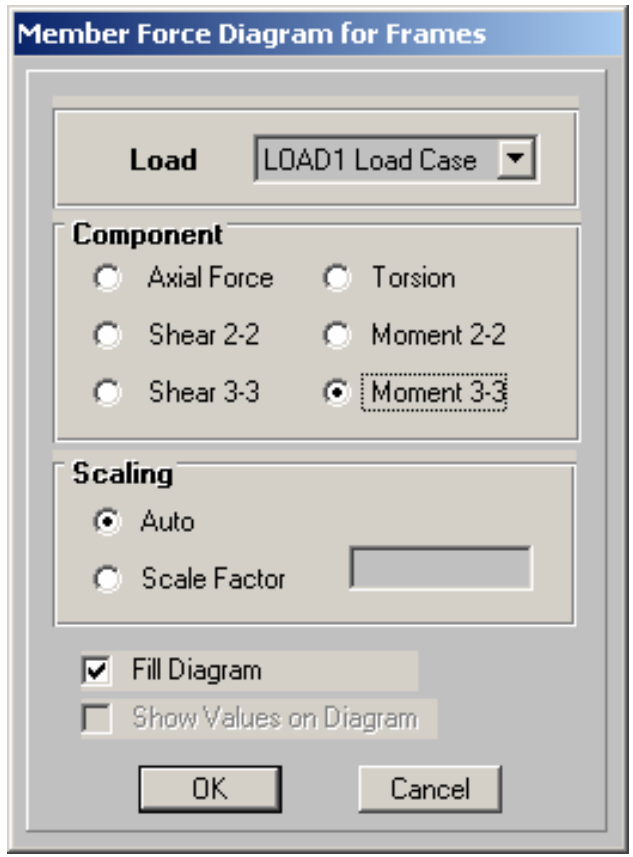
- Click “Analyze-Run”
- You should get a deformed shape that looks as below.
- You will be asked to save at this point



Step 10. Analysis Results

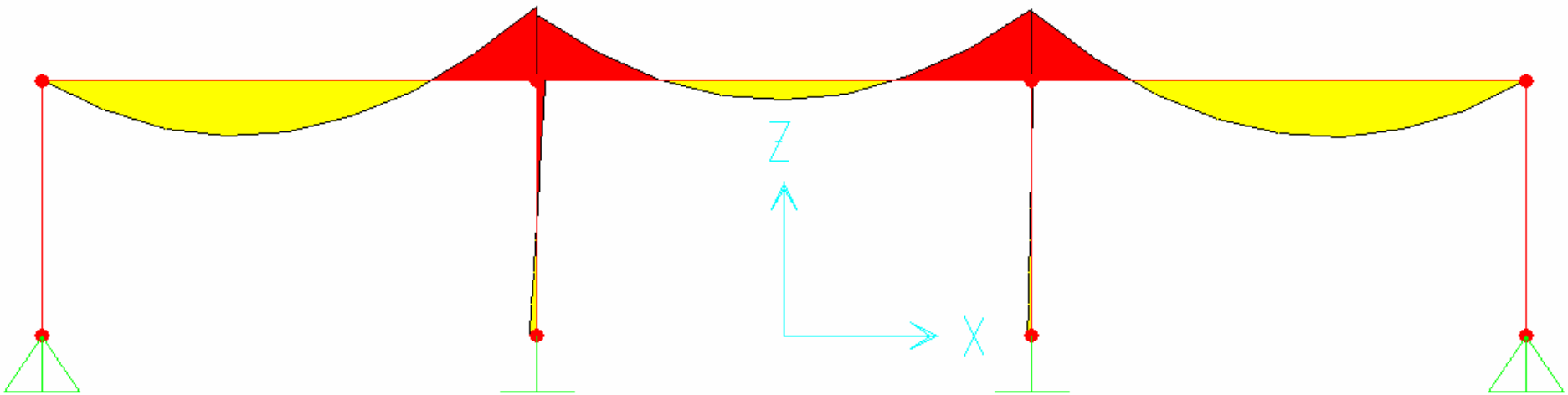
- You have four tabs on the top as shown below.

- The J stands for Joint Reactions.
- The F stands for member forces. This is where you will get the design axial forces, shears, and moments. For instance, click on Moment 3-3.
- Click “OK”



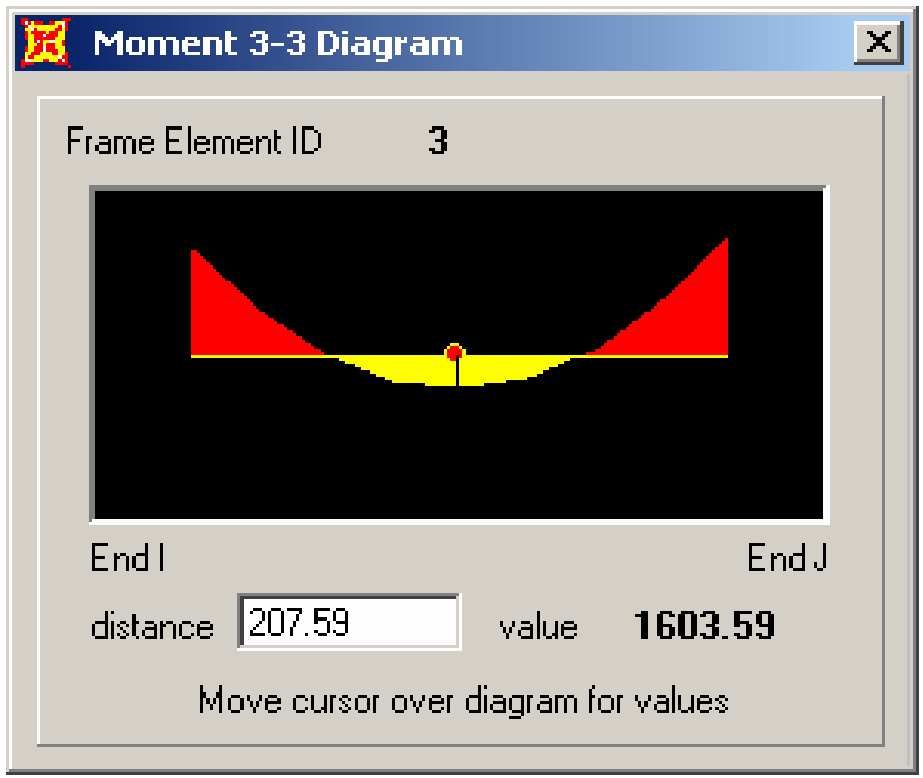
Step 10. Analysis Results

- You will see something as below:



Step 10. Analysis Results

- If you right click on the element, it will compute the forces on the beam
- Units are in kip-in
- This is what you will do to design your members



Step 10. Analysis Results

- If you want displacements at the nodes and maximum element forces, click on “File-Print Output Tables”
- Click the appropriate boxes and click “OK”.
- The file will be found as indicated in the textbox.