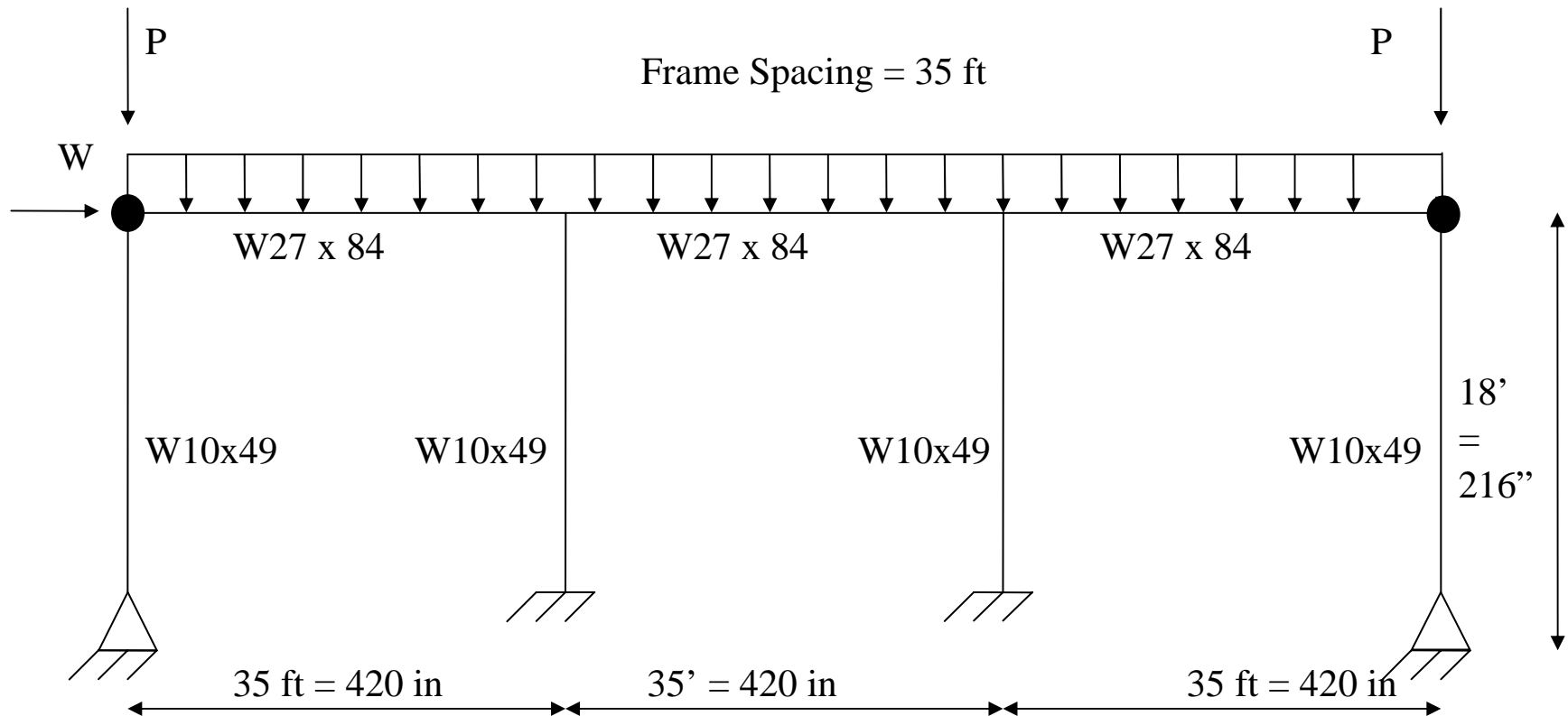


Tutorial to SAP2000 Student Edition

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

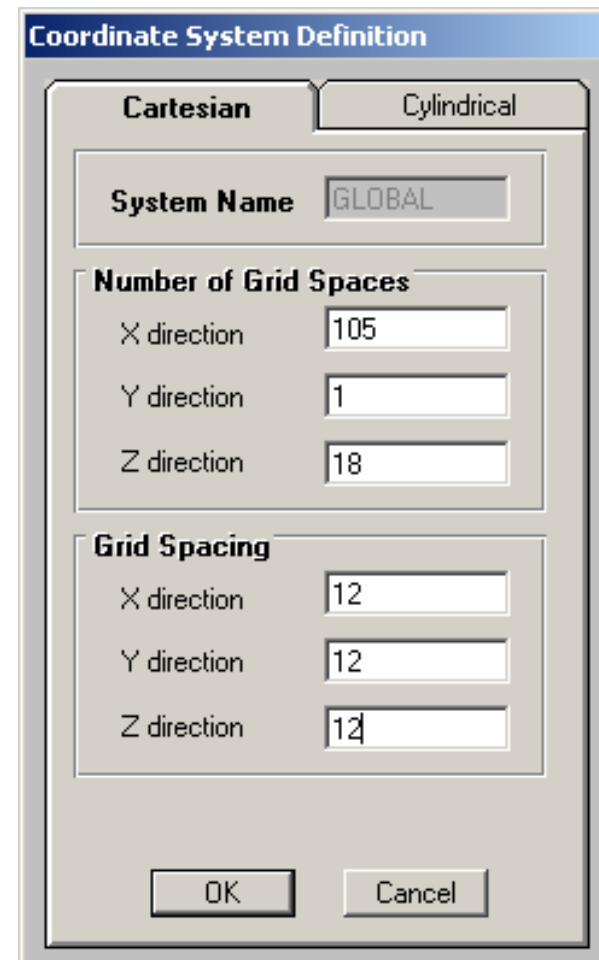
Distributed Load: $w_u = (1.2 \cdot 0.233333 + 0.5 \cdot 0.1166667) = 0.33833$ kips/in

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

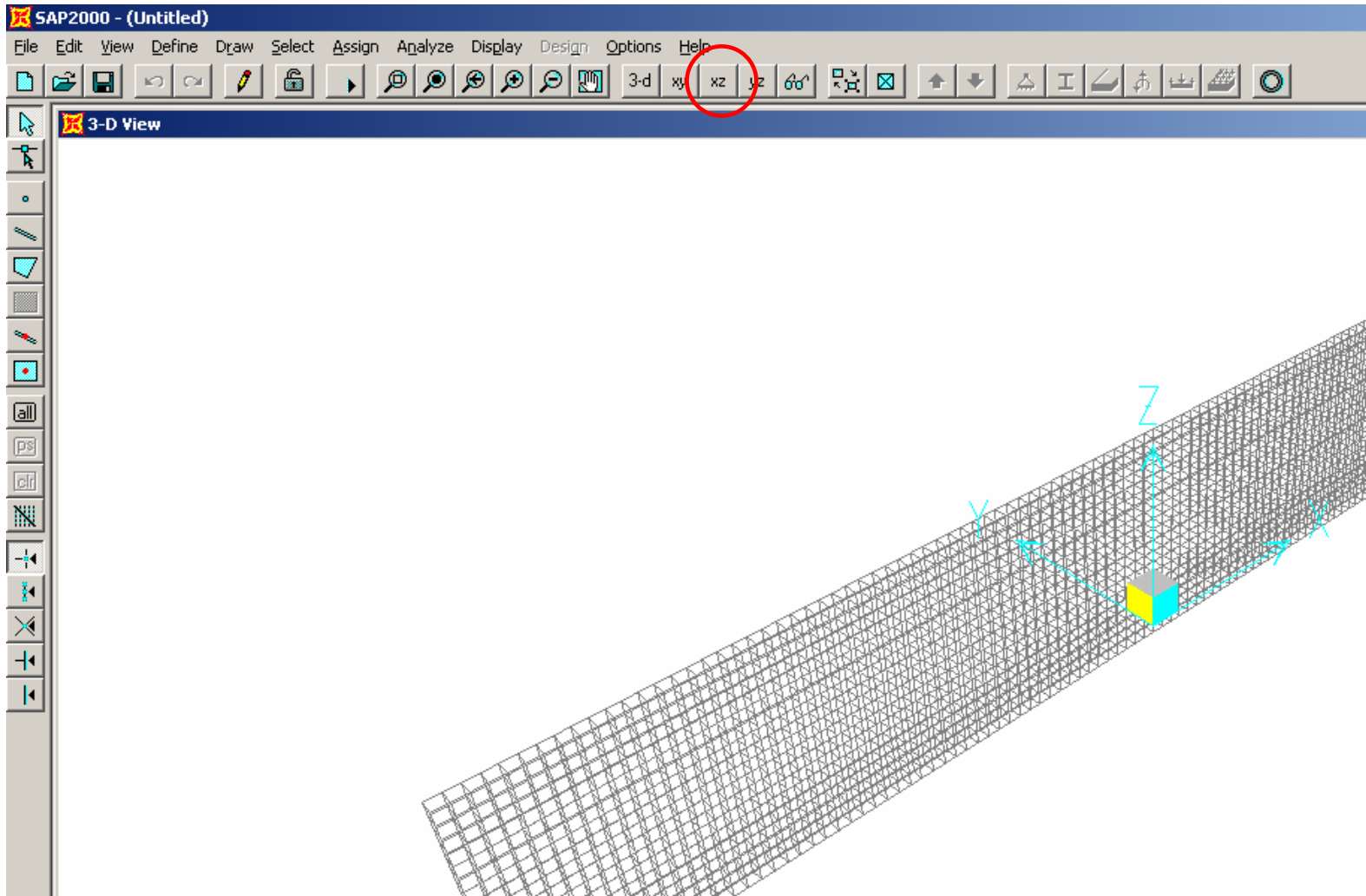


The image shows a software dialog box titled "Coordinate System Definition". It has two tabs: "Cartesian" (selected) and "Cylindrical". Under the "Cartesian" tab, there is a "System Name" field with the text "GLOBAL". Below this is a section titled "Number of Grid Spaces" containing three input fields: "X direction" with the value "105", "Y direction" with the value "1", and "Z direction" with the value "18". Another section titled "Grid Spacing" contains three input fields: "X direction" with the value "12", "Y direction" with the value "12", and "Z direction" with the value "12". At the bottom of the dialog are "OK" and "Cancel" buttons.

Field	Value
System Name	GLOBAL
Number of Grid Spaces	
X direction	105
Y direction	1
Z direction	18
Grid Spacing	
X direction	12
Y direction	12
Z direction	12

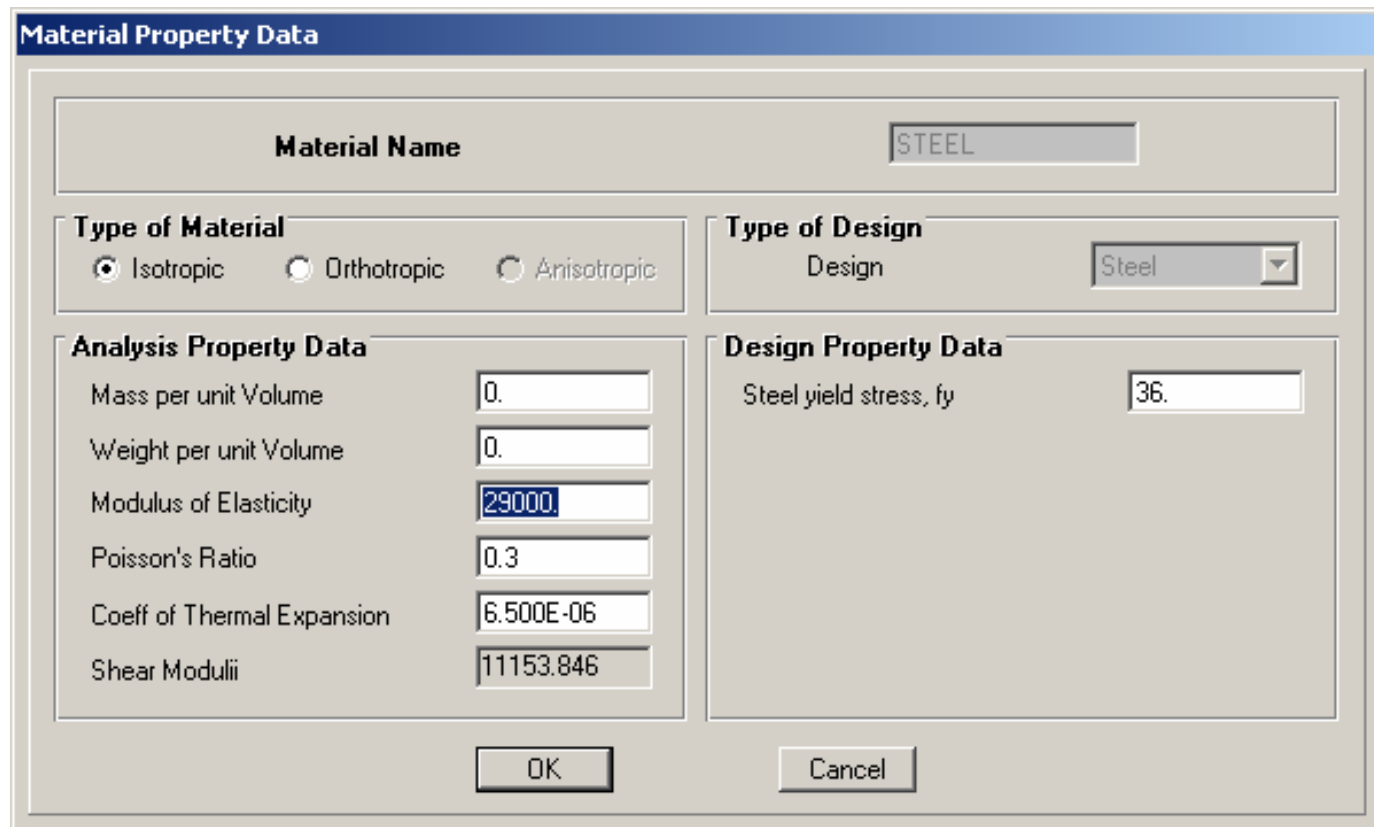
Step 2: Getting Started

- 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



The image shows a software dialog box titled "Material Property Data". It is used to define the material properties for a selected material, in this case, "STEEL".

Material Name: STEEL

Type of Material: ☒ Isotropic ☐ Orthotropic ☐ Anisotropic

Type of Design: Design: Steel (dropdown menu)

Analysis Property Data:

Property	Value
Mass per unit Volume	0.
Weight per unit Volume	0.
Modulus of Elasticity	29000.
Poisson's Ratio	0.3
Coeff of Thermal Expansion	6.500E-06
Shear Moduli	11153.846

Design Property Data:

Property	Value
Steel yield stress, fy	36.

Buttons: OK, Cancel

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

I/Wide Flange Section

Section Name W10x49

Properties
Section Properties Modification Factors

Material STEEL

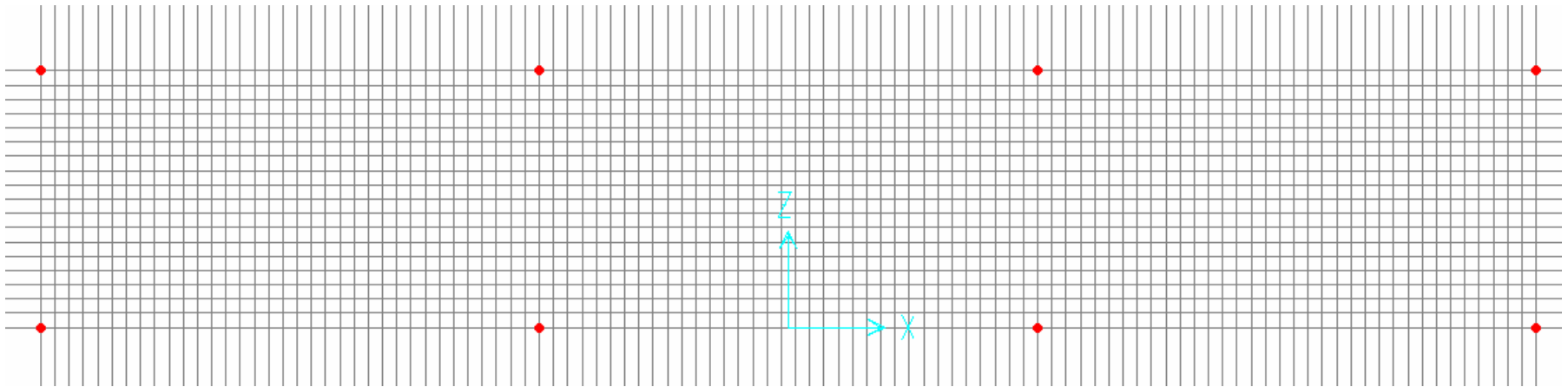
Dimensions

Outside height (t3)	9.98
Top flange width (t2)	10
Top flange thickness (tf)	.56
Web thickness (tw)	.34
Bottom flange width (t2b)	10
Bottom flange thickness (tfb)	.56

OK Cancel

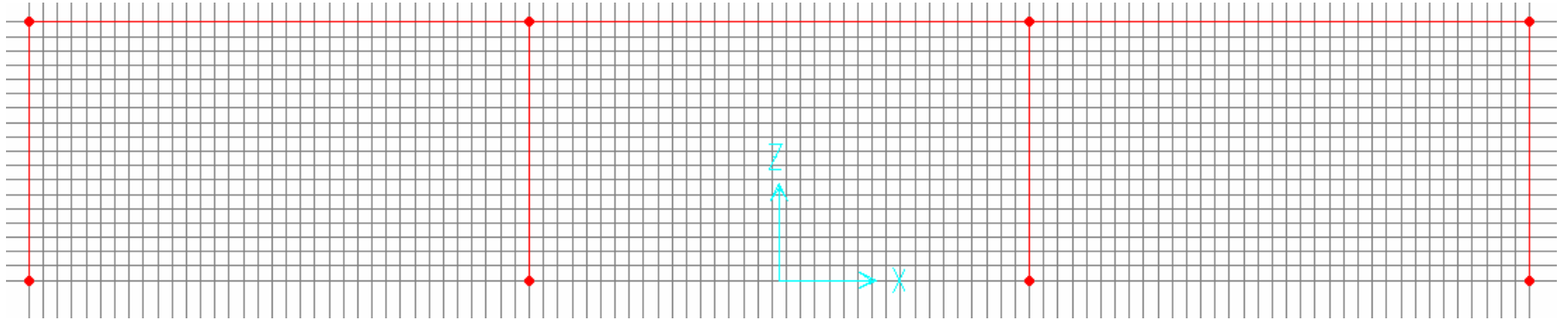
Step 4. Define Nodes and Elements

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



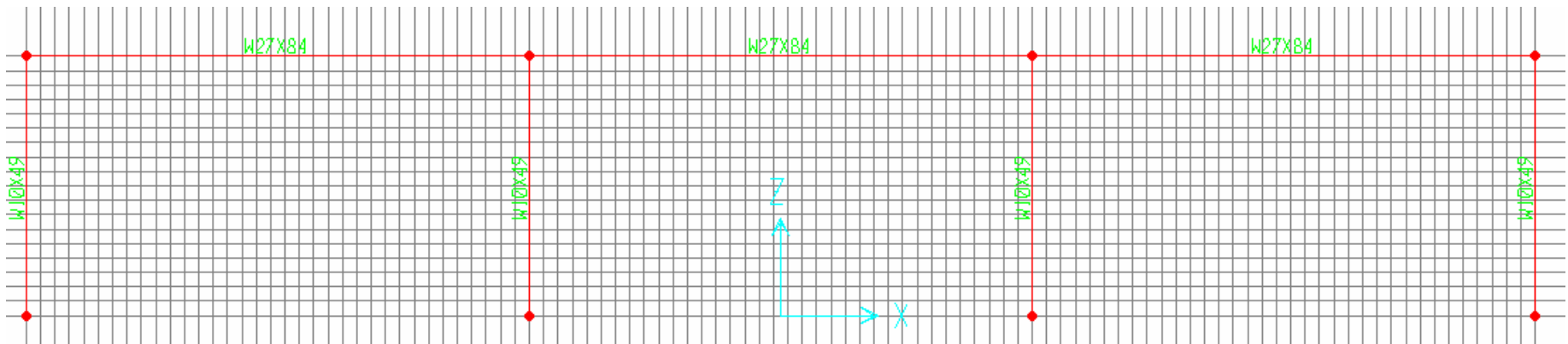
Step 4. Define Nodes and Elements

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



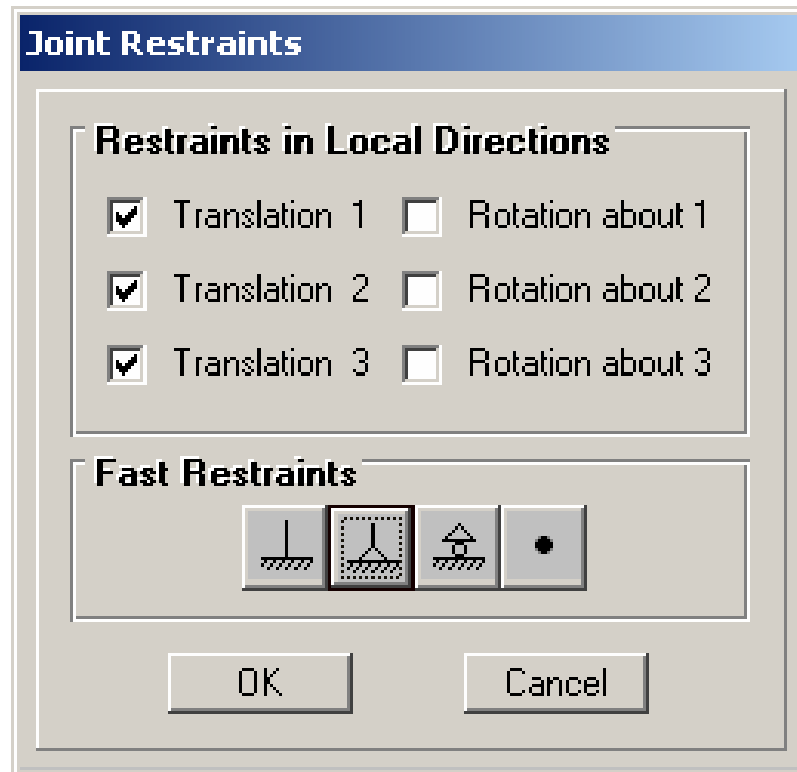
Step 5. Assigning Sections (For this model)

- 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



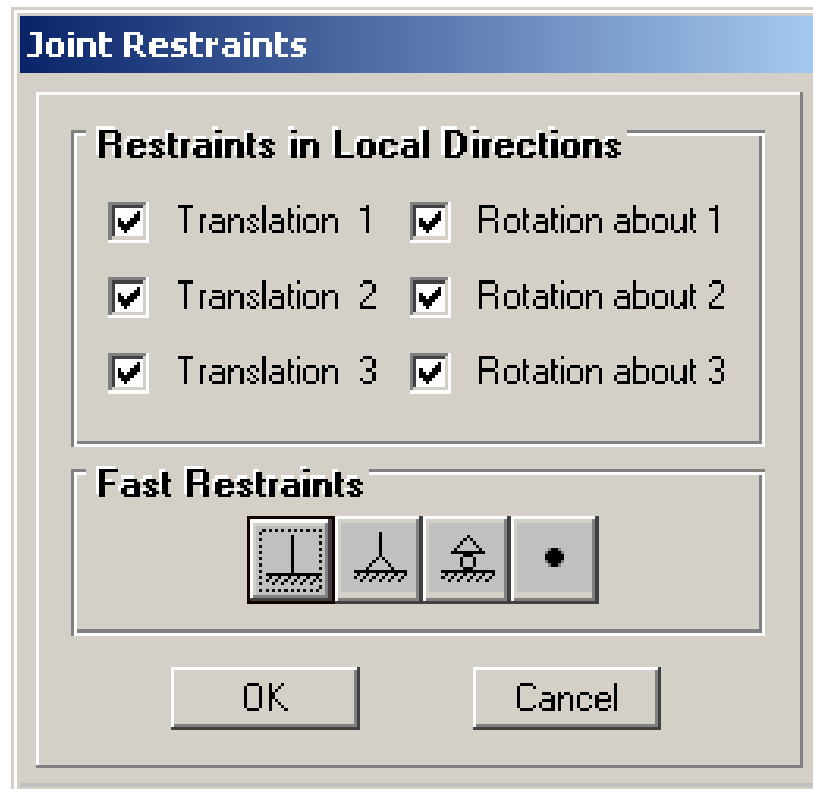
Step 6. Define Restraints

- 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”



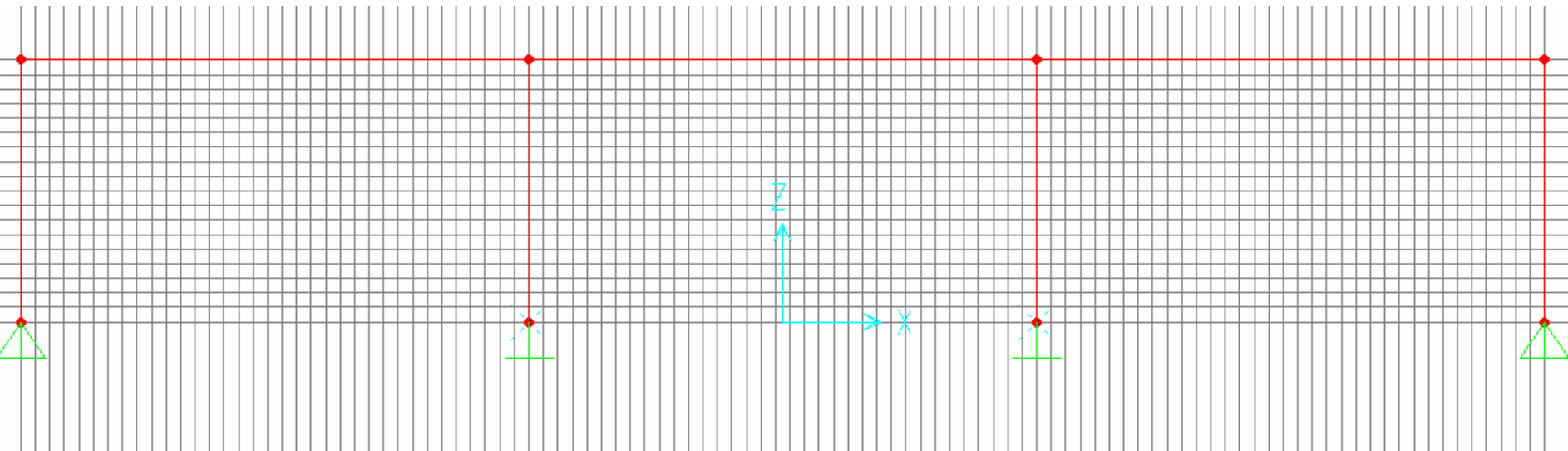
Step 6. Define Restraints

- 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”

Joint Forces

Load Case Name: **LOAD1**

Loads

Force Global X	8.19
Force Global Y	0.
Force Global Z	-168.4
Moment Global XX	0.
Moment Global YY	0.
Moment Global ZZ	0.

Options

- ☒ Add to existing loads
- ☐ Replace existing loads
- ☐ Delete existing loads

OK Cancel

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”

Joint Forces

Load Case Name LOAD1

Loads

Force Global X	0
Force Global Y	0.
Force Global Z	-168.4
Moment Global XX	0.
Moment Global YY	0.
Moment Global ZZ	0.

Options

- ☒ Add to existing loads
- ☐ Replace existing loads
- ☐ Delete existing loads

OK

Cancel

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”

Point and Uniform Span Loads

Load Case Name LOAD1

Load Type and Direction
☒ Forces ☐ Moments
Direction Gravity

Options
☐ Add to existing loads
☒ Replace existing loads
☐ Delete existing loads

Point Loads

	1.	2.	3.	4.
Distance	0.	0.25	0.75	1.
Load	0.	0.	0.	0.

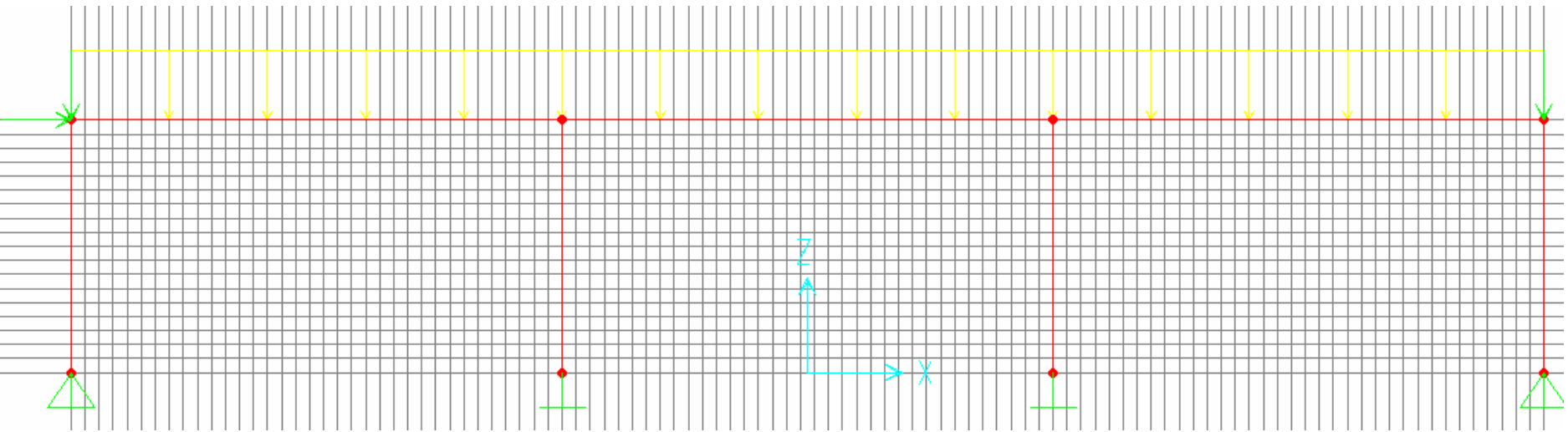
☒ Relative Distance from End-I ☐ Absolute Distance from End-I

Uniform Load
.33833

OK Cancel

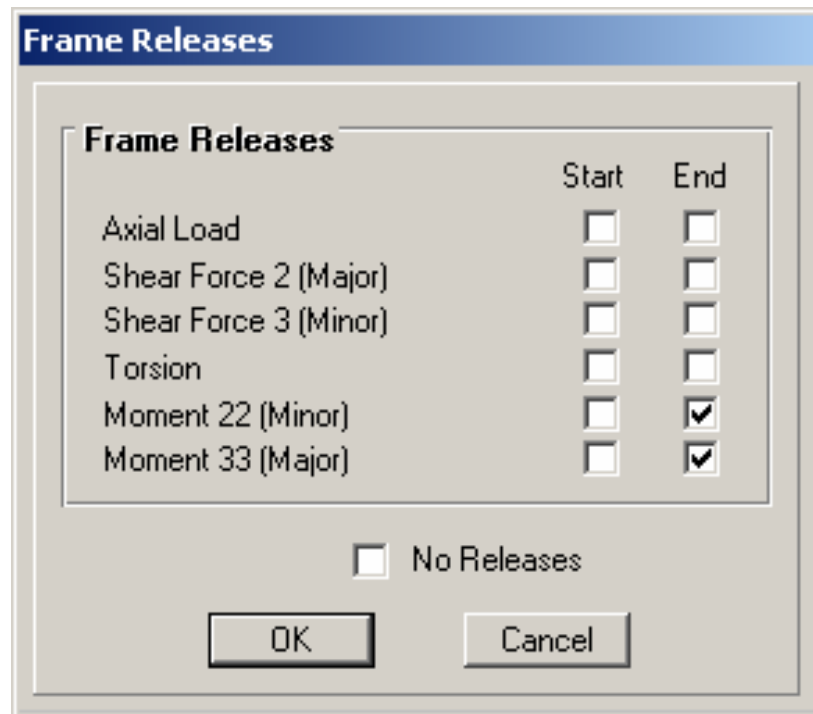
Step 7. Assign Loads

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



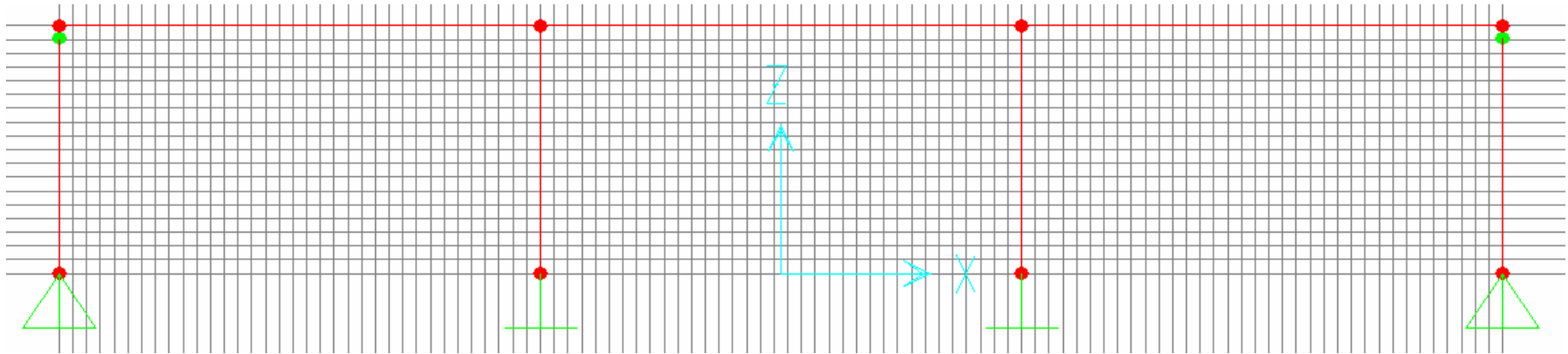
Step 8. Making Pin Connections

- 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.



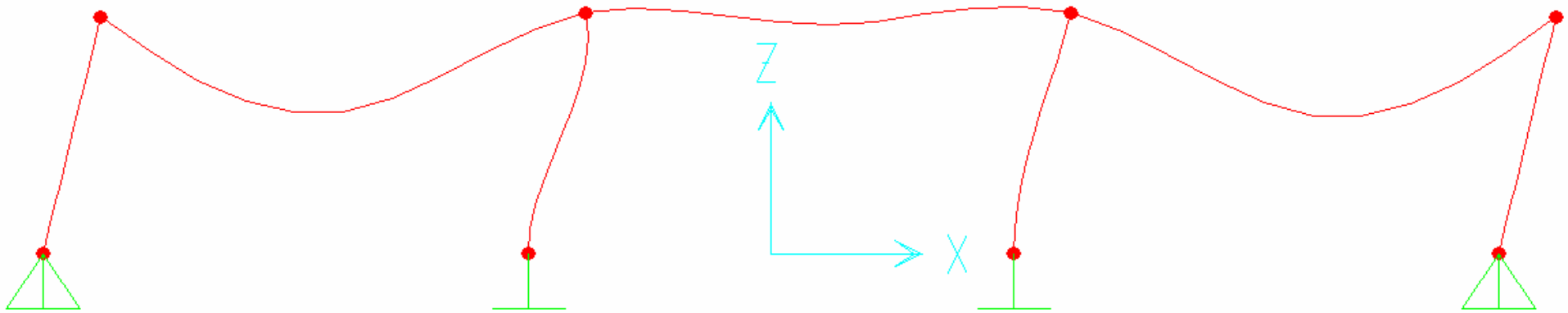
Step 8. Making Pin Connections

- 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

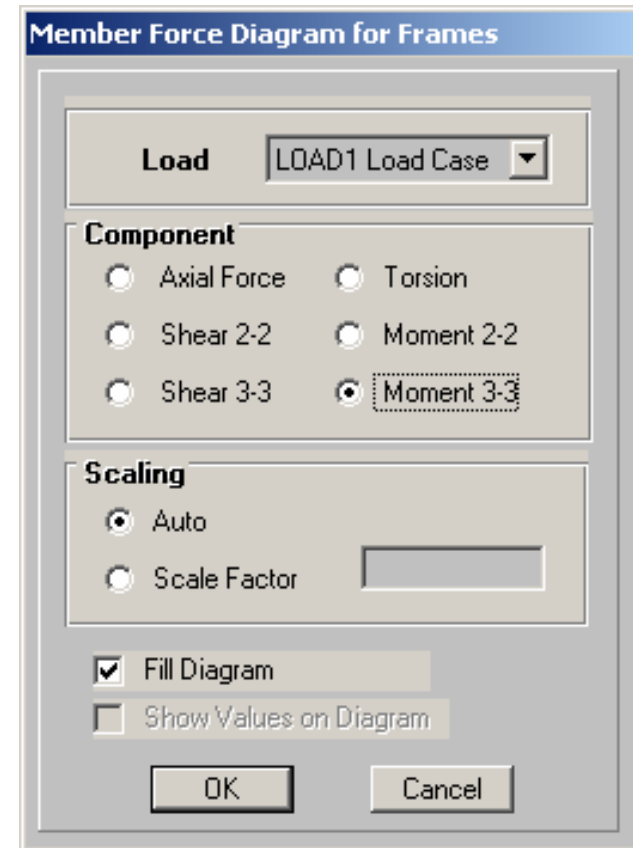


Step 10. Analysis Results

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

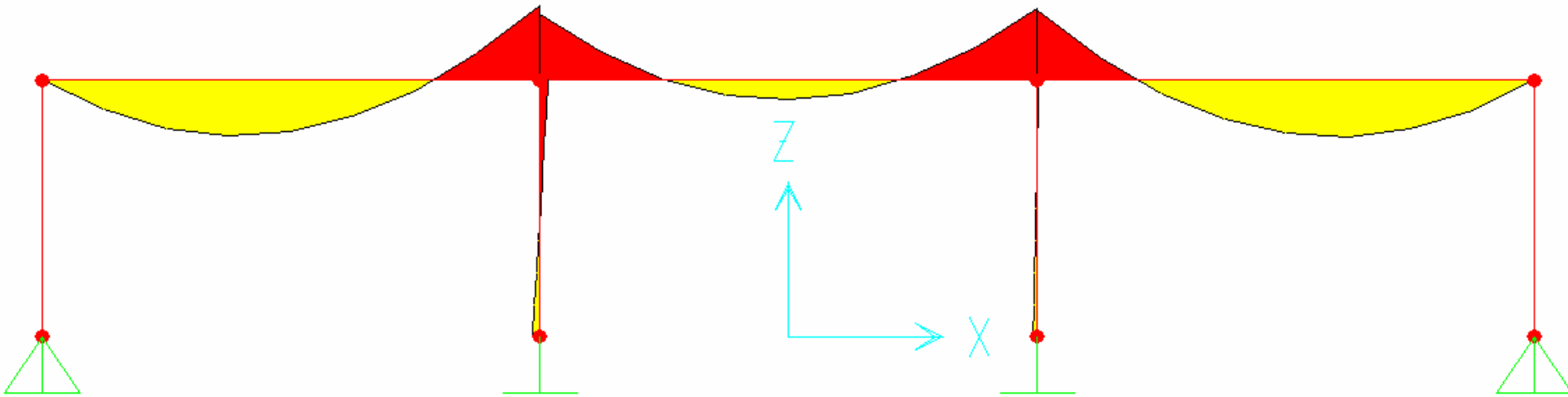


- The J stands for Joint Reactions.
- The F stands for member forces. This is where you will get the design axial forces, shears, and moments. For instance, click on Moment 3-3.
- Click “OK”



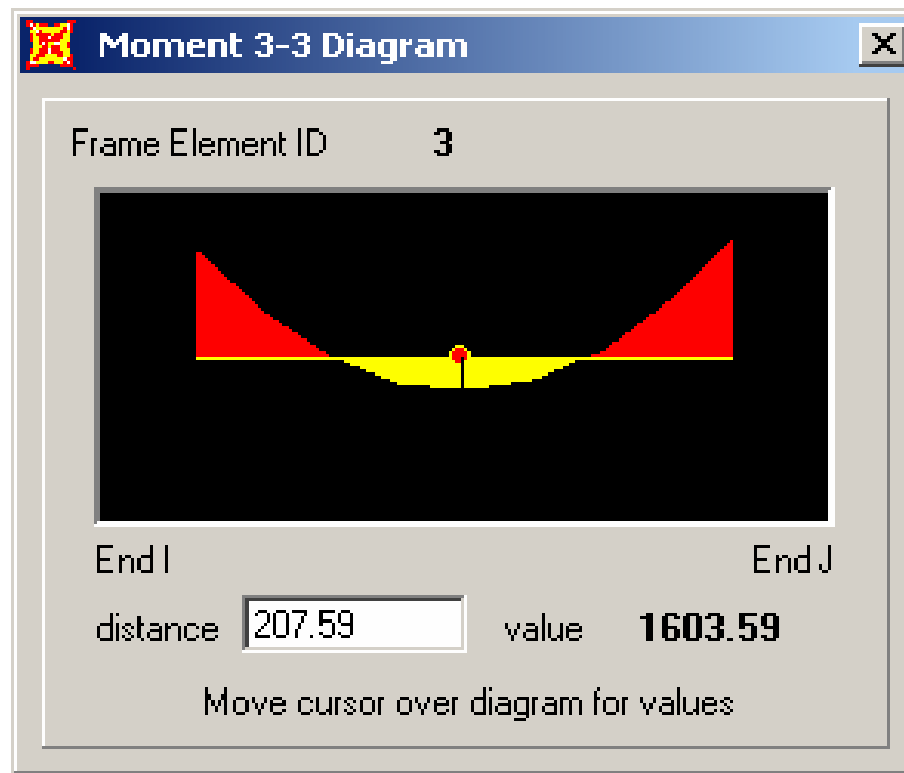
Step 10. Analysis Results

- 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.

