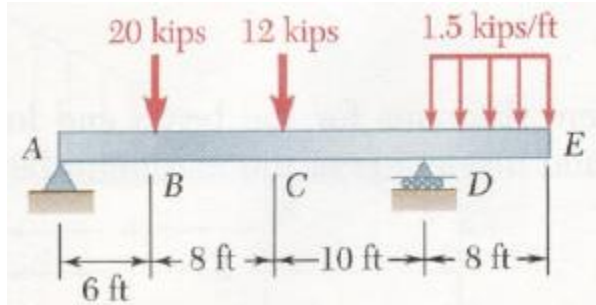


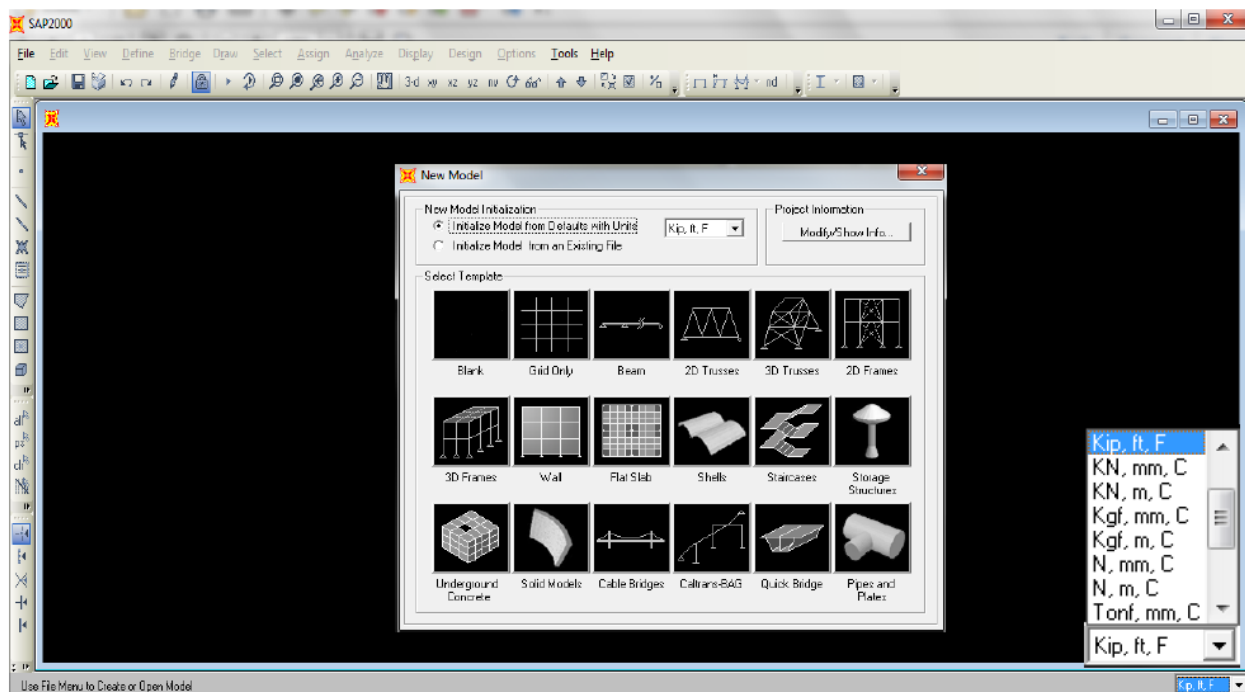
## Tutorial on Shear and Moment diagrams with distributed forces



Draw the shear and bending moment diagram for the beam and loading shown

**Step 1:** Define Dimensions (bottom right), set to **Kip, ft, F**.

**Step 2:** File → New Model → Grid only



**Step 3:** In Quick Grid Lines menu input the following: Number of Gridlines: X = 5, Y = 1, Z =1. Grid Spacing: X = 6, Y = 1, Z =1. We are working in XZ coordinate system; therefore, any value except for zero can be put in the Y direction.

Quick Grid Lines

Cartesian Cylindrical

Coordinate System Name  
GLOBAL

Number of Grid Lines

X direction 5

Y direction 1

Z direction 1

Grid Spacing

X direction 6

Y direction 1

Z direction 1

First Grid Line Location

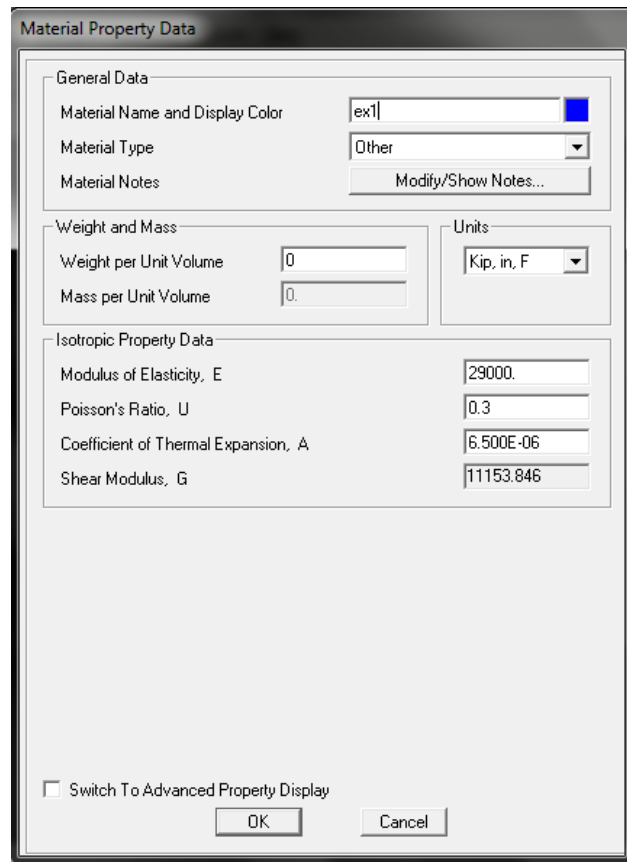
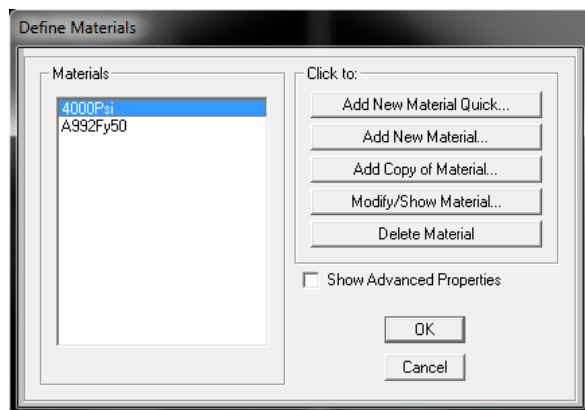
X direction 0.

Y direction 0.

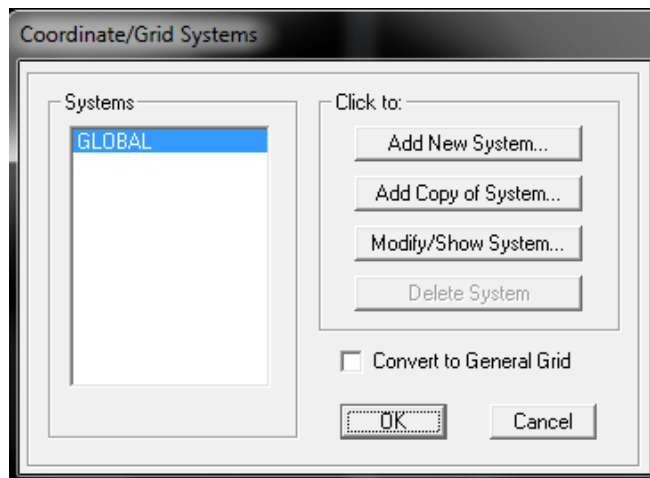
Z direction 0.

OK Cancel

**Step 4: Define Materials Properties:** From the menu select Define → Materials. Select add new material and then under Material Type select others. Put 0 for Weight per unit vol. and in the Modulus of Elasticity put 29000 since it is not given assume it to be steel.



**Step 5: Spacing:** right click on mouse then select Edit Grid Data → on the pop up window select Modify/Show System on the Define Grid System Data window under display Grid As and select spacing modify the spacing for A, B, C, and D select ok



**Define Grid System Data**

Edit Format

System Name: GLOBAL Units: Kip, in, F

Grid Lines: Quick Start...

X Grid Data

	Grid ID	Spacing	Line Type	Visibility	Bubble Loc.	Bubble Loc.
1	A	6	Primary	Show	End	
2	B	8	Primary	Show	End	
3	C	10	Primary	Show	End	
4	D	8	Primary	Show	End	
5	E	0	Primary	Show	End	
6						
7						
8						

Y Grid Data

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

Z Grid Data

	Grid ID	Spacing	Line Type	Visibility	Bubble Loc.	Bubble Loc.
1	Z1	0	Primary	Show	End	
2						
3						
4						
5						
6						
7						
8						

Display Grids as: ☐ Ordinates ☒ Spacing

☐ Hide All Grid Lines

☐ Glue to Grid Lines

Bubble Size: 0.6667

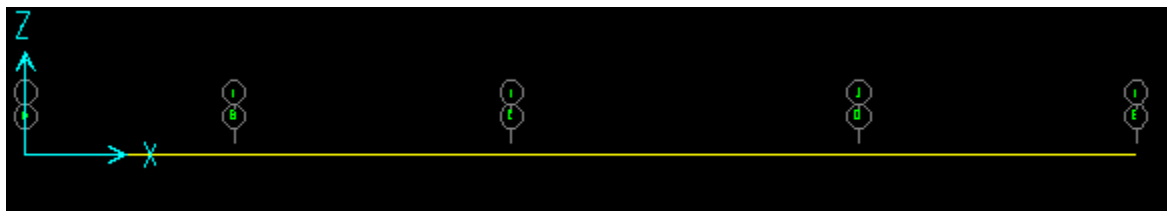
Reset to Default Color

Reorder Ordinates

OK Cancel

**Step 6: Drawing Frame Elements:** Draw → Draw Frame/ Cable/ Tendon, or select the Frame/ Cable/ Tendon button on the left side of the menu interface to start drawing the truss. Once finished drawing the frame elements, hit Esc button or click on the Set Select Mode button located on the top left of the interface menu.(it's one member 32ft long)

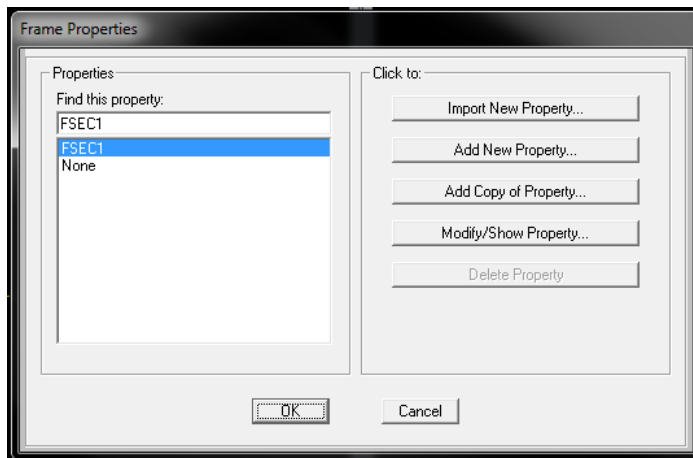
(XZ plane)



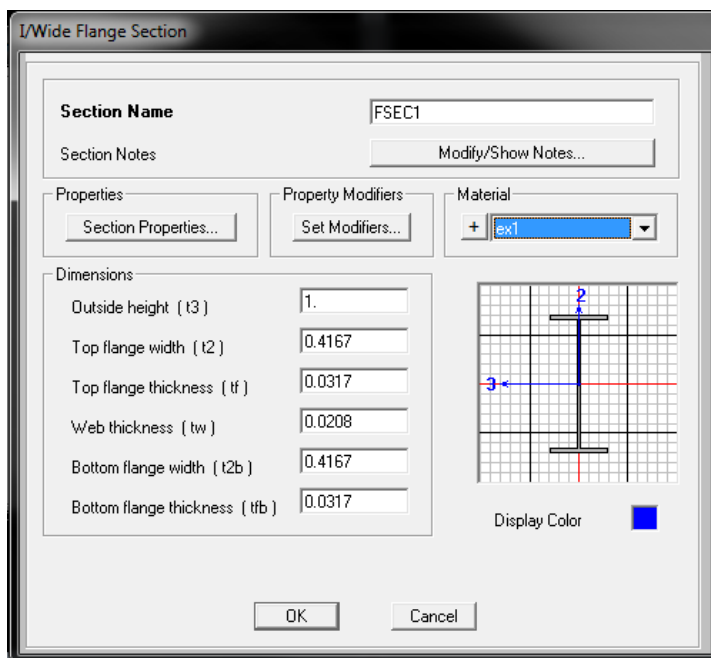
**Step 7: Drawing Joints:** Draw → Draw Special Joint, or select the Special Joint button on the left side of the menu interface to start drawing. Draw joints where forces are going to be applied



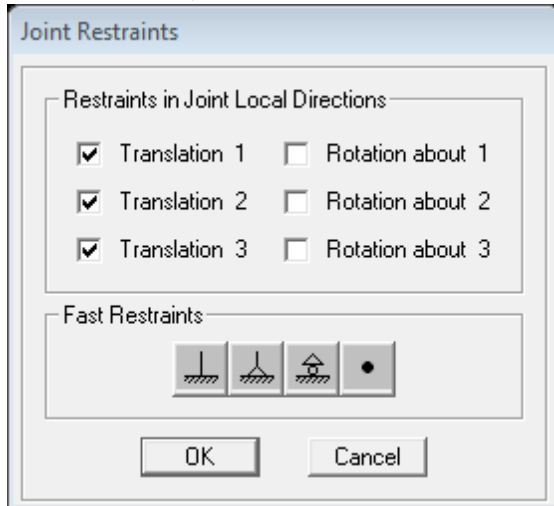
**Step 8 :** Left click on the frame then select: Assign → Frame → Frame Sections, on the pop up window select **FSEC1** and press Modify/Show Property



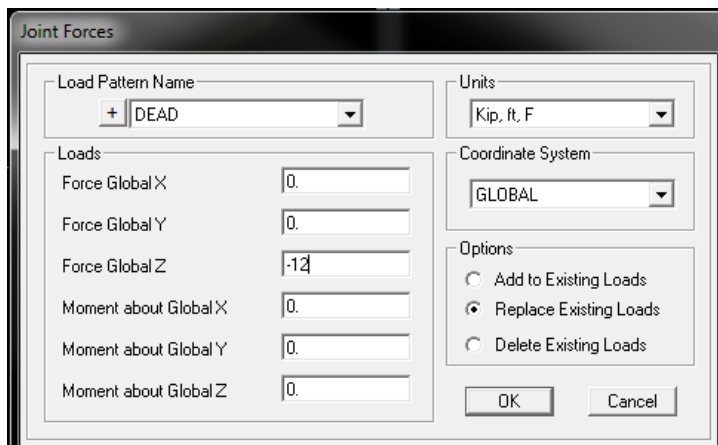
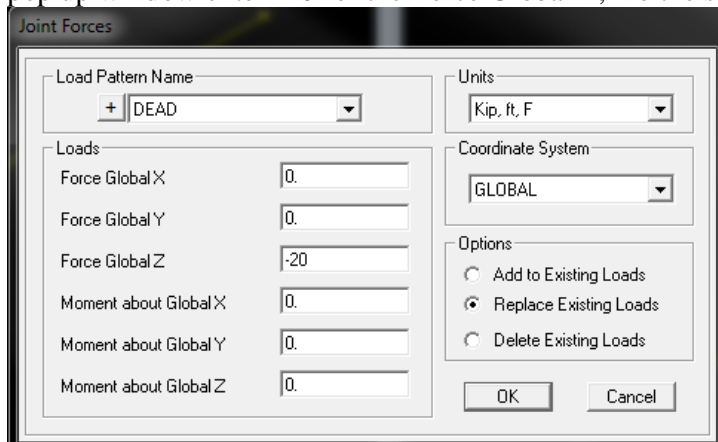
Under material select **ex1** and press ok



**Step 9: Define Structural Supports:** Left click on the left support select: Assign → Joint → Restraints, Pin Do the same for the other supports.



**Step 10: Loads:** Left click on the joint select: Assign → Joint → Joint loads → Forces, in the pop up window enter -20 for the Force Global Z, Do the same for the other Joint and enter -12.



**Step 11: Distributed load:** Left click on the frame: select: Assign → Frame Loads → Distributed loads. On the put up window select **Absolute Distance from End-I** in the Trapezoidal loads enter in the distance boxes 24, 32, 0, 0 and in the Load boxes enter 1.5, 1.5, 0, 0.

**Frame Distributed Loads**

Load Pattern Name: + DEAD Units: Kip, ft, F

Load Type and Direction: ☒ Forces ☐ Moments  
 Coord Sys: GLOBAL Direction: Gravity

Options: ☐ Add to Existing Loads ☒ Replace Existing Loads ☐ Delete Existing Loads

Trapezoidal Loads:

	1.	2.	3.	4.
Distance	24.	32.	0.	0.
Load	1.5	1.5	0.	0.

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

Uniform Load: Load 0.

OK Cancel

**Step 12: Show loads:** Select: Display → Show load Assign → Joint... on the pop up window make sure the **show loading values** is selected click ok

**Show Joint Loads**

Load Pattern Name: DEAD

Resolve Forces in this Coordinate System: Coord System: None, (display as defined)

Load Type: ☒ Forces ☐ Displacements

☒ Show Loading Values

OK Cancel

**Show Frame Loads**

**Load Pattern Name** DEAD

**Load Type**

- ☒ Span Loading (Forces)
  - Coord System None, (display as defined)
- ☐ Span Loading (Moments)
  - Coord System None, (display as defined)
- ☐ Gravity Multipliers
  - Coord System GLOBAL
- ☐ Temperature Contours
- ☐ Temperature Values
- ☐ Temperature Gradient 2-2 Contours
- ☐ Temperature Gradient 2-2 Values
- ☐ Temperature Gradient 3-3 Contours
- ☐ Temperature Gradient 3-3 Values
- ☐ Deformation Loads
- ☐ Target Forces
- ☐ Strain Load Values
  - Component
- ☐ Tendon Applied Load Data
- ☐ Tendon Calculated Load Data
- ☐ Span Wave Loads
  - Load Step
  - Coord System None, (display as defined)
- ☐ Open Structure Wind Loads
  - Coord System Frame Local

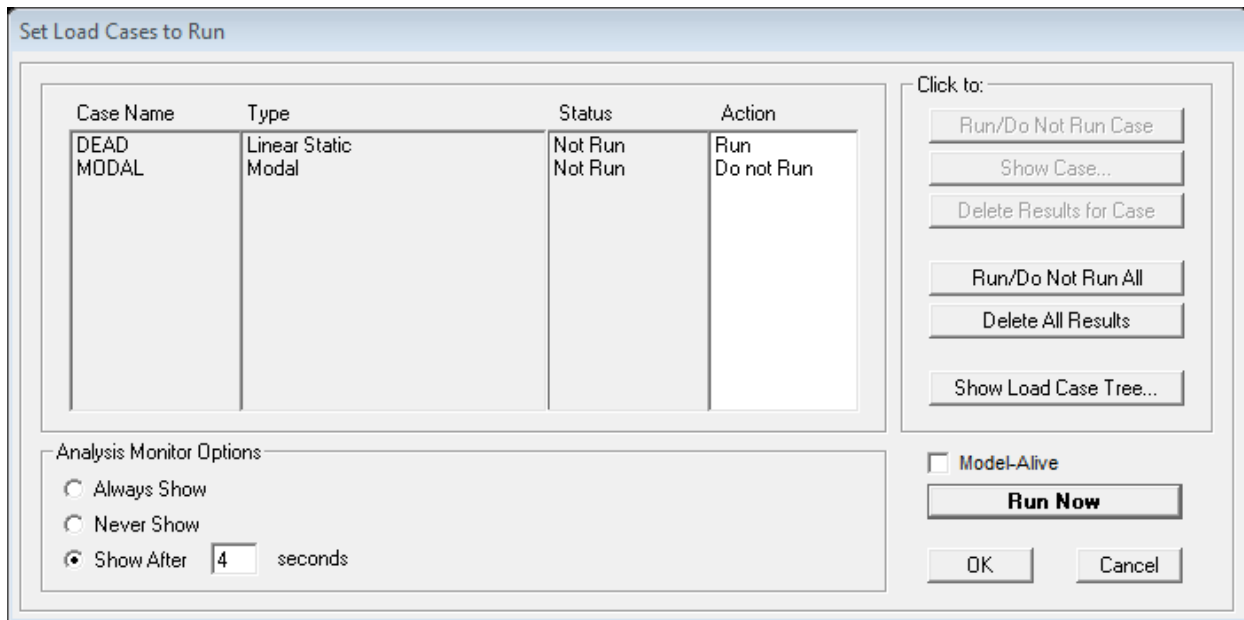
☒ Show Joint Loads with Span Loads

☒ Show Span Loading Values

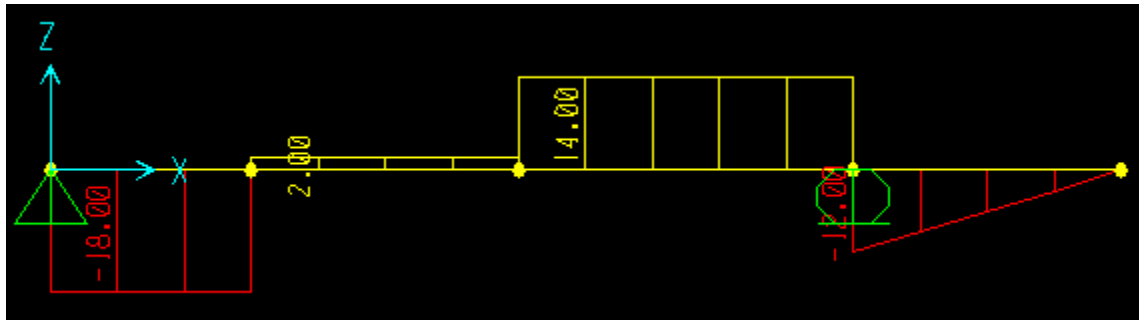
OK Cancel



**Step 13: Run Analysis:** Choose Analyze → Run Analysis. Set MODAL to Do Not Run and click the Run Now button.



**Step 14: Display Shear Diagram:** Display → Show Forces/ Stresses → Frames/ Cables. Under Component select Shear 2-2, under Options select Show Values on Diagram and hit OK.



**Step 15: Display Moment Diagram:** Display → Show Forces/ Stresses → Frames/ Cables. Under Component select Moment 3-3, under Options select Show Values on Diagram and hit OK.

