

## Problem W

### Simple Beam With Trapezoidal Loads

#### Steel

$E = 29000$  ksi

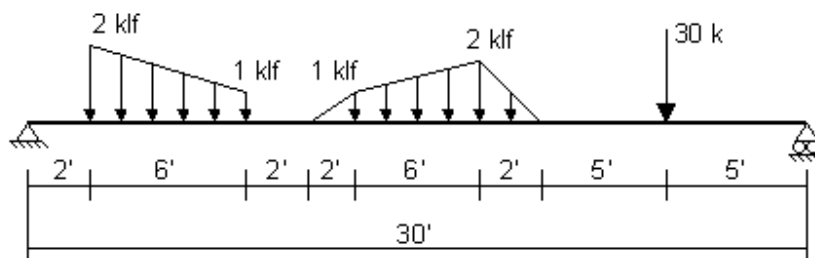
Poissons Ratio = 0.3

Beam = W21X50

#### To Do

Determine midspan deflection of beam. Ignore the self weight of the beam.  
Create the model as follows:

1. From the File menu, choose New Model From Template. Select the Beam template in the upper left hand corner. Set the number of spans to one.
2. Define the frame section properties.
3. Apply the loads to the beam.
4. Use the Divide Frames option in the Edit Menu to break the beam into two elements with a joint at the center.



### CSI Solution Demonstrate Use of These Features


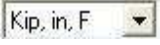
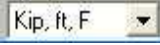

- Divide Frames
- Trapezoidal Loads (Distributed Loads)

### Problem W Solution

1. Click the **File menu > New Model** command to display the **New Model** form.

2. Click the drop-down box to set the units to .

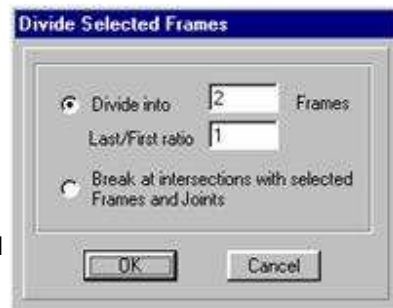


3. Click on the **Beam** button  to display the **Beam** form. In that form:
  - Type **1** in the *Number of Spans* edit box.
  - Type **30** in the *Span Length* edit box.
  - Click the **OK** button.
4. Click the “X” in the top right-hand corner of the 3-D View window to close it.
5. Click the drop-down box in the status bar to change the units to .
6. Click the **Define menu > Materials** command to display the **Define Materials** form.
7. Click on STEEL in the *Materials* area to highlight (select) it, and then click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:
  - Verify 29000 is entered in the *Modulus of Elasticity* edit box.
  - Verify 0.3 is entered in the *Poisson's Ratio* edit box.
  - Accept the other default values.
  - Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to exit all forms.
8. Click the drop-down box in the status bar to change the units to .
9. Click the **Define menu > Load Cases** command to display the **Define Loads** form. In that form:
  - Type **0** in the *Self Weight Multiplier* edit box.
  - Click the **Modify Load** button.
  - Click the **OK** button.
10. Select the line (beam) object.
11. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form.
12. Highlight *W21X50* in the *Properties* area and click the **OK** button.
13. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignment.
14. Select the line (beam) object.
15. Click the **Assign menu > Frame/Cable/Tendon Loads > Point** command to display the **Frame Point Loads** form. In that form:
  - In the *Load Type and Direction* area, verify that the *Forces* option is selected and that the *Gravity* direction is selected.
  - In the *Point Loads* area, select the *Absolute Distance From End I* option.
  - In the *Point Loads* area, type **25** in the first *Distance* edit box and type **30** in the first *Load* edit box

- Click the **OK** button.
16. Select the line (beam) object.
17. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
- In the *Load Type and Direction* area, verify that the *Forces* option is selected and that the *Gravity* direction is selected.
  - In the *Trapezoidal Loads* area, select the *Absolute Distance From End I* option.
  - In the *Trapezoidal Loads* area, type **2** in the first *Distance* edit box and type **2** in the first *Load* edit box
  - Type **8** in the second *Distance* edit box and type **1** in the second *Load* edit box
  - Click the **OK** button.
18. Select the line (beam) object.
19. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
- In the *Trapezoidal Loads* area, type **10** in the first *Distance* edit box and type **0** in the first *Load* edit box
  - Type **12** in the second *Distance* edit box and type **1** in the second *Load* edit box
  - Type **18** in the third *Distance* edit box and type **2** in the third *Load* edit box
  - Type **20** in the fourth *Distance* edit box and type **0** in the fourth *Load* edit box
  - Click the **OK** button.


20. Select the line (beam) object.

21. Click the **Edit menu > Divide Frames** command to display the **Divide Selected Frames** form. Verify that the form appears as shown in the figure click the **OK** button. The frame object broken into two objects with a joint at center. Note that the frame loading did change.

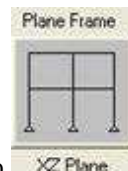


and  
is  
the  
not


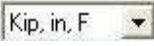
22. Click the **Show Undeformed Shape**

button  to remove the displayed frame static load assignments.

23. Click the **Analyze menu > Set Analysis Options** command to display the **Analysis Options** form. In that form:



- Click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
- Click the **OK** button.

24. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:
- Click on *MODAL* in the *Case Name* list to highlight it.
  - Click the **Run/Do Not Run Case** button.
  - Click the **Run Now** button.
25. When the analysis is complete, check the messages in the **SAP Analysis Monitor** window (there should be no warnings or errors) and then click the **OK** button to close the window.
26. Right click on the center joint to see its displacement in feet.
27. Click the drop-down box in the status bar to change the units to .
28. Right click on the center joint to see its displacement in inches.