

Problem W

Simple Beam With Trapezoidal Loads

Steel

$E = 29000$ ksi

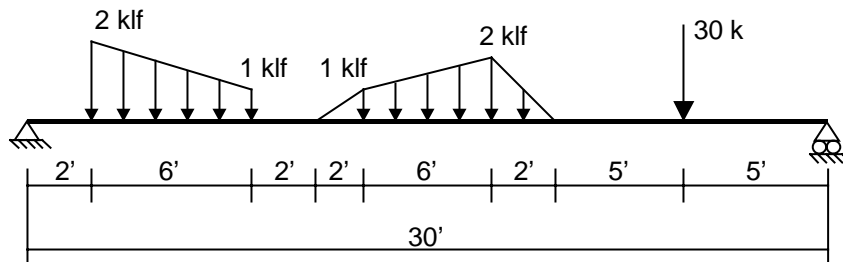
Poissons Ratio = 0.3

Beam = W21X50

To Do


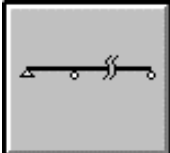


Determine midspan deflection of beam. Ignore the self weight of the beam.
Create the model as follows:


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.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem W Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 
2. From the **File** menu select **New Model From Template...** This displays the Model Templates dialog box.
3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.
4. In this dialog box:
 - Type **1** in the Number of Spans edit box.
 - Type **30** in the Span Length edit box.
 - Click the **OK** button.
5. Click the “X” in the top right-hand corner of the 3-D View window to close it.
6. Click the drop down box in the status bar to change the units to kip-in. 
7. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
8. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.
9. In this dialog box:
 - Verify 29000 is entered in the Modulus of Elasticity edit box.
 - Verify .3 is entered in the Poisson’s Ratio edit box.
 - Accept the other default values.
 - Click the **OK** button twice to exit all dialog boxes.
10. Click the drop down box in the status bar to change the units to kip-ft. 
11. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.
12. In this dialog box:
 - Type **0** in the Self Weight Multiplier edit box.
 - Click the **Change Load** button.

- Click the **OK** button.
13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
 14. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.
 15. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.
 16. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:
 - Scroll down and click on the W21X50 section.
 - Click the **OK** button three times.
 17. Select the frame element.
 18. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
 19. Highlight W21X50 in the Frame Sections area and click the **OK** button.
 20. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignment.
 21. Select the frame element.
 22. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
 23. In this dialog box:
 - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z 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.
 24. Select the frame element.
 25. From the **Assign** menu select **Frame Static Loads...** and then **Trapezoidal...** from the submenu to display the Trapezoidal Span Loads dialog box.

26. In this dialog box:
- In the Load Type and Direction area verify that the Forces option is selected and that the Global Z 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 **2** in the second Distance edit box and type **-2** in the second Load edit box
 - Type **8** in the third Distance edit box and type **-1** in the third Load edit box
 - Type **8** in the fourth Distance edit box and type **-1** in the fourth Load edit box
 - Click the **OK** button.

27. Select the frame element.

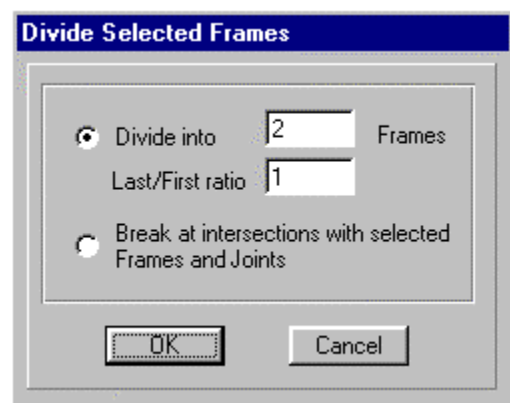
28. From the **Assign** menu select **Frame Static Loads...** and then **Trapezoidal...** from the submenu to display the Trapezoidal Span Loads dialog box.

29. In this dialog box:

- 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.

30. Select the frame element.

31. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box. Verify that the dialog box appears as shown in the figure and click the **OK** button. The frame element is broken into two elements with a joint at the center. Note that the frame loading did not change.




32. Click the **Show Undeformed Shape** button  to remove the displayed frame static load assignments.

33. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.


- In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.

- Click the **OK** button.

34. Click the **Run Analysis** button  to run the analysis.

35. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

36. Right click on the center joint to see its displacement in feet.

37. Click the drop down box in the status bar to change the units to kip-in. 

38. Right click on the center joint to see its displacement in inches.