

# Problem I

## Prestressed Concrete Beam

### Concrete

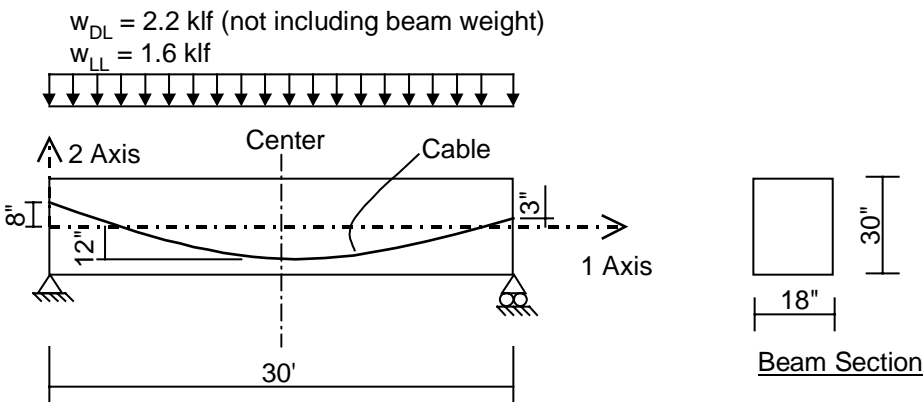
$E = 4400$  ksi, Poissons Ratio = 0.2

$f'_c = 6$  ksi

Cable Tension = 200 kips


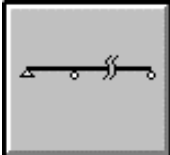

### To Do


Determine the moment diagram for a DL + LL + PRESTRESS loading combination. Compare the results using 4 output segments and using 30 output segments.



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.


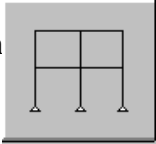



## **Problem I Solution**




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. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
7. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.
8. In this dialog box:
  - Verify 0.15 is entered in the Weight per Unit Volume edit box.
  - Click the **OK** button twice to exit all dialog boxes.
9. Click the drop down box in the status bar to change the units to kip-in. 
10. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
11. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.
12. In this dialog box:
  - Type **4400** in the Modulus of Elasticity edit box.
  - Verify .2 is entered in the Poisson’s Ratio edit box.
  - Verify 60 is entered in the Reinforcing Yield Stress, fy edit box.
  - Type **6** in the Concrete Strength, fc edit box.

- Type **60** in the Shear steel Yield Stress, *fys* edit box.
  - Type **6** in the Concrete Shear Strength, *fcs* edit box.
  - Accept the other default values.
  - Click the **OK** button twice to exit all dialog boxes.
13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
14. In this dialog box:
- With the default section, FSEC1, highlighted, click the **Modify/Show Section** button to display the Rectangular Section dialog box.
  - In this dialog box:
    - Select CONC from the Material drop-down box.
    - Type **30** in the Depth (t3) edit box.
    - Type **18** in the Width (t2) edit box.
    - Click the **OK** button twice to exit all dialog boxes.
15. Select the frame element by clicking on it.
16. From the **Assign** menu select **Frame** and then **Prestress...** from the submenu to display the Frame Prestressing Patterns dialog box.
17. In this dialog box:
- Type **200** in the Cable Tension edit box.
  - In the Cable Eccentricities area type **8** in the Start edit box.
  - In the Cable Eccentricities area type **12** in the Middle edit box.
  - In the Cable Eccentricities area type **3** in the End edit box.
  - Click the **OK** button.
18. Click the drop down box in the status bar to change the units to kip-ft. 
19. From the **Define** menu select **Static Load Cases...** This will display the Define Static Load Case Names dialog box.
20. In this dialog box:

- Type **DL** in the Load edit box.
  - Click the **Change Load** button
  - Type **LL** in the Load edit box.
  - Select LIVE from the Type drop-down box.
  - Type **0** in the Self weight Multiplier box.
  - Click the **Add New Load** button.
  - Type **PRESTRES** in the Load edit box.
  - Select OTHER from the Type drop-down box.
  - Click the **Add New Load** button.
  - Click the **OK** button.
21. From the **Define** menu select **Load Combinations....** This will display the Define Load Combinations dialog box.
22. In this dialog box:
- Click the **Add New Combo** button to display the Load Combination Data dialog box.
  - In this dialog box:
    - Accept the default load combination name, COMB1
    - Accept the default load combination type, Add.
    - Type **COMB1: DL + LL + Prestress** in the Title edit box.
    - Verify the DL Load Case is selected in the Case Name drop-down box.
    - Verify that 1 is entered in the Scale factor edit box.
    - Click the **Add** button.
    - Select LL Load Case from the Case Name drop-down box.
    - Click the **Add** button.
    - Select PRESTRES Load Case from the Case Name drop-down box.
    - Click the **Add** button.

- Click the **OK** button twice to exit all dialog boxes.
23. Select the frame element.
  24. 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.
  25. In this dialog box:
    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-2.2**.
    - Click the **OK** button.
  26. Select the frame element.
  27. 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.
  28. In this dialog box:
    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-1.6**.
    - Click the **OK** button.
  29. Select the frame element.
  30. From the **Assign** menu select **Frame Static Loads...** and then **Prestress...** from the submenu to display the Frame Prestress Loads dialog box.
  31. In this dialog box:
    - Select PRESTRES from the Load Case Name drop-down box.
    - Type **1** in the Scale Factor edit box..
    - Click the **OK** button.
  32. Select the frame element.
  33. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

34. In this dialog box:
- Type **4** in the Number of Segments edit box.
  - Click the **OK** button.
35. Click the **Show Undeformed Shape** button  to remove the displayed frame output segment assignments.
36. 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.
37. Click the **Run Analysis** button  to run the analysis.
38. 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.
39. Click the **Member Force Diagram for Frames** button , (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.
40. In this dialog box:
- Select COMB1 Combo from the Load drop-down box.
  - Select the Moment 3-3 option in the Component area.
  - Uncheck the Fill Diagram check box.
  - Check the Show Values on Diagram check box.
  - Click the **OK** button to display the moment diagram.
- Note: You may want to print this moment diagram for comparison with the one obtained when 30 output segments are specified. To print the moment diagram select **Print Graphics** from the **File** menu.*
- Note: For load combinations, when force diagrams are plotted, exact values are only calculated at the ends of each output segment. These exact values are plotted and then they are connected with straight lines.*
41. Click the **Lock/Unlock Model** button  on the main toolbar to unlock the model. Click the **OK** button when asked if it is OK to delete.

42. Select the frame element.
43. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.
44. In this dialog box:
  - Type **30** in the Number of Segments edit box.
  - Click the **OK** button.
45. Click the **Show Undeformed Shape** button  to remove the displayed frame output segment assignments.
46. Click the **Run Analysis** button  to run the analysis.
47. 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.
48. Click the **Member Force Diagram for Frames** button , (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.
49. In this dialog box:
  - Verify that the COMB1 Combo is selected in the Load drop-down box.
  - Verify that the Moment 3-3 option is selected in the Component area.
  - Verify that the Show Values on Diagram check box is checked.
  - Click the **OK** button to display the moment diagram.