

# Problem J

## Beam On Elastic Foundation

### Concrete

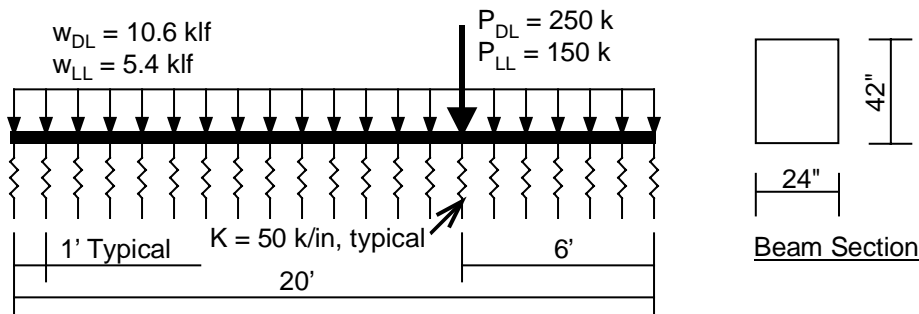
$E = 3120 \text{ ksi}$

Poissons Ratio = 0.2

### To Do


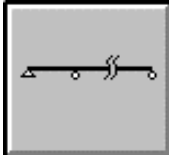

Determine the moment diagram under combined dead plus live loads and the maximum downward displacement.

**Note:** Dead load shown does not include beam self weight.




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.

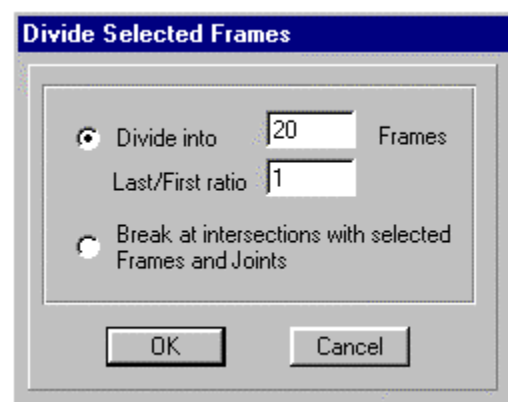
## **Problem J Solution**

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 **20** in the Span Length edit box.
  - Uncheck the Restraints check 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 **3120** in the Modulus of Elasticity edit box.
  - Verify .2 is entered in the Poisson’s Ratio 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 **42** in the Depth (t3) edit box.
      - Type **24** in the Width (t2) edit box.
      - Click the **OK** button twice to exit all dialog boxes.





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






16. Select the frame element by clicking on it.
17. From the **Edit** menu select **Divide Frames...** to display the Divide Select frames dialog box.
18. Fill in the dialog box as shown in the adjacent figure and click the **OK** button.
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.
  - 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** 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.
    - Click the **OK** button twice to exit all dialog boxes.
23. Select all of the frame elements by “windowing”.
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 **-10.6**.
  - Click the **OK** button.
26. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).
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 **-5.4**.
  - Click the **OK** button.
29. Click the **Show Undeformed Shape** button  to remove the displayed frame uniform load assignments.
30. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
31. In this dialog box:
  - Check the Labels box in the Joints area.
  - Click the **OK** button.
32. Select joint 16 (6 feet from the right end) by clicking on it.
33. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
34. In this dialog box:
  - Select DL from the Load Case Name drop-down box.
  - Type **-250** in the Force Global Z edit box in the Loads area.
  - Click the **OK** button.
35. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).
36. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
37. In this dialog box:
  - Select LL from the Load Case Name drop-down box.
  - Type **-150** in the Force Global Z edit box in the Loads area.
  - Click the **OK** button.
38. Click the **Show Undeformed Shape** button  to remove the displayed joint load assignments.

39. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
40. In this dialog box:
  - Uncheck the Labels box in the Joints area.
  - Click the **OK** button.
41. Click the drop down box in the status bar to change the units to kip-in. 
42. Select all of the joint elements by “windowing”.
43. From the **Assign** menu select **Joint** and then **Springs...** from the submenu to display the Joint Springs dialog box.
44. In this dialog box:
  - Type **50** in the Translation 3 edit box.
  - Click the **OK** button.
45. Click the **Show Undeformed Shape** button  to remove the displayed frame output segment assignments.
46. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
47. In this dialog box:
  - Uncheck the UX, UY, RX and RZ check boxes leaving just the UZ and RY boxes checked.
  - Click the **OK** button.
48. Click the **Run Analysis** button  to run the analysis.
49. 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.
50. 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.
51. 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: If the font size is too small for you to read the moment values use the following procedure to increase the font size. From the **Options** menu select **Preferences**, click on the Dimensions Tab if it is not already visible, type in a new (larger) font size in the Minimum Graphic Font Size edit box (usually about 6 points is sufficient), click the **OK** button and then click the **Refresh Window** button  on the main toolbar.*

*Note: You can right click on any of the frame elements to see details of the moment diagram for that element.*

52. Click the drop down box in the status bar to change the units to kip-in. 
53. Click the **Display Static Deformed Shape** button  (or select **Show Deformed Shape...** from the **Display** menu). The Deformed Shape dialog box appears.
54. In this dialog box:
  - Select COMB1 Combo from the Load drop-down box.
  - Click the **OK** button.
55. Right click on the joint at the far right end of the beam to view its deflection.