Problem R

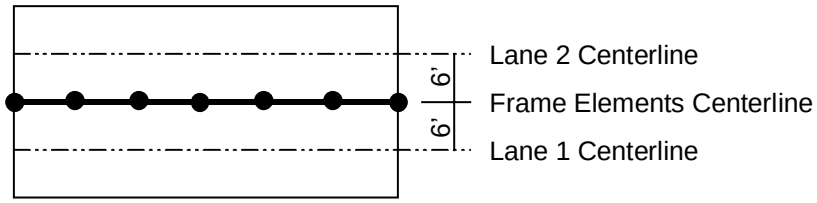
Bridge With Moving Load

Concrete Material Properties
E =5000 ksi, Poissons Ratio = 0.2

Member Properties

Column	Girder
A = 40 ft^2	A = 35 ft^2
I = 400 ft^3	I = 500 ft^3
AS = 30 ft^2	AS = 12 ft^2

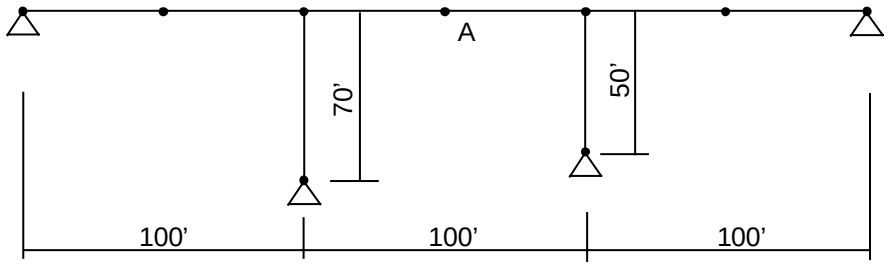
Moving Load
Number of Lanes = 2



Check for worst case of HS20-44 truck load and HS20-44L lane load applied to each lane simultaneously.

Use the Exact method of calculation.

To Do
Set the number of ouput segments for girder elements to 2. Review the moving load influence line for vertical displacement at joint A in Lane 1. Review the moving load girder M33 moments in Lane 1. Set the number of ouput segments for the girder elements to 10. Review the same influence line and moments.

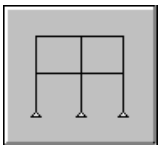


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem R 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 **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box

   - Type **1** in the Number of Stories edit box.
   - Type **3** in the Number of Bays edit box.
   - Type **70** in the Story Height edit box.
   - Type **100** in the Bay Width 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. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Labels box in the Joints area.
   - Check the Labels box in the Frames area.
   - Click the **OK** button.

8. Select column elements 1 and 4. Press the delete key on the keyboard to delete these elements.

9. Click the **Refresh Window** button  to refresh the drawing.

10. Select joints 2, 3, 5 and 8.

11. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

12. In this dialog box:

    - Verify that the Translation 1, 2 and 3 check boxes are checked. If they are not checked, check them.
    - Click the **OK** button.

13. Select joint 5.

14. From the **Edit** menu select **Move...** to display the **Move Selected Points** menu.

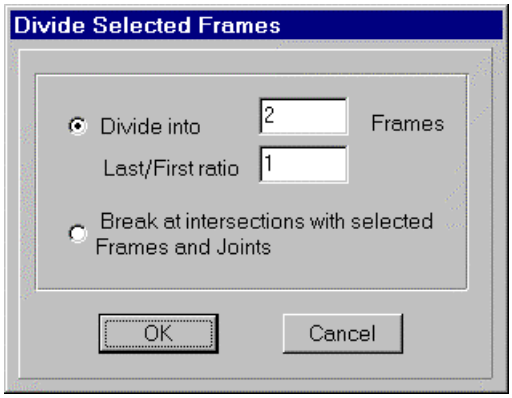
15. In this dialog box:

    - Type **20** in the Delta Z edit box.
    - Verify that 0 is entered in the Delta X and Delta Y edit boxes.
    - Click the **OK** button.

16. Select frame elements 5, 6 and 7.

17. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.

18. Verify that the dialog box appears as shown in the figure and click the **OK** button.



19. Select frame elements 8 through 13 (i.e., the girder elements).

20. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

21. In this dialog box:

    - Type **2** in the Number of Segments edit box.
    - Click the **OK** button.

22. Click the **Show Undeformed Shape** button  to remove the displayed output segment assignments so that you can see the frame element labels again.

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

24. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

25. 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.

26. In this dialog box:

    - Type **5000** in the Modulus of Elasticity edit box.
    - Type **.2** in the Poisson's Ratio edit box, if it is not already entered.
    - Click the **OK** button twice to exit all dialog boxes.

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

28. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

29. In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add General item to display the Property Data dialog box.

30. In this dialog box:

    - Type **40** in the Cross Sectional (Axial) Area edit box.
    - Type **400** in the Moment Of Inertia About 3 Axis edit box.
    - Type **30** in the Shear Area in 2 Direction edit box.
    - Click the **OK** button to display the General Section dialog box.
    - In this dialog box:
        - Type **COLUMN** in the Section Name edit box.
        - Select CONC from the Material drop-down box.
        - Click the **OK** button to return to the Define Frame Sections dialog box.

31. In the Click To area, click the drop-down box that says Add General and then click on the Add General item to display the Property Data dialog box.

32. In this dialog box:

    - Type **35** in the Cross Sectional (Axial) Area edit box.
    - Type **500** in the Moment Of Inertia About 3 Axis edit box.
    - Type **12** in the Shear Area in 2 Direction edit box.
    - Click the **OK** button to display the General Section dialog box.
    - In this dialog box:

- Type **GIRDER** in the Section Name edit box.
- Select CONC from the Material drop-down box.
- Click the **OK** button twice to exit all dialog boxes.

33. Select frame elements 8 through 13 (girders).

34. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

35. In this dialog box:

    - Click on GIRDER in the Frame Sections area to highlight it.
    - Click the **OK** button.

36. Select the two column elements.

37. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

38. In this dialog box:

    - Click on COLUMN in the Frame Sections area to highlight it.
    - Click the **OK** button.

39. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments so that you can see the frame element labels again.

40. From the **Define** menu select **Moving Load Cases** and then select **Lanes...** to display the Define Bridge Lanes dialog box.

41. In this dialog box:

    - Click the **Add New Lane** button to display the Lane Data dialog box.
    - In this dialog box:
        - Accept the default Lane Name, LANE1.
        - Type **8** in the Frame edit box.
        - Type **-6** in the Eccentricity edit box.
        - Click the **Add** button.
        - Type **9** in the Frame edit box.

- Click the **Add** button.
- Type **10** in the Frame edit box.
- Click the **Add** button.
- Type **11** in the Frame edit box.
- Click the **Add** button.
- Type **12** in the Frame edit box.
- Click the **Add** button.
- Type **13** in the Frame edit box.
- Click the **Add** button.
- Click the OK button to return to the Define Bridge Lanes dialog box. In this dialog box:

- Click the **Add New Lane** button to display the Lane Data dialog box.
- In this dialog box:
  - Accept the default Lane Name, LANE2.
  - Type **8** in the Frame edit box.
  - Type **6** in the Eccentricity edit box.
  - Click the **Add** button.
  - Type **9** in the Frame edit box.
  - Click the **Add** button.
  - Type **10** in the Frame edit box.
  - Click the **Add** button.
  - Type **11** in the Frame edit box.
  - Click the **Add** button.
  - Type **12** in the Frame edit box.
  - Click the **Add** button.

- Type **13** in the Frame edit box.
- Click the **Add** button.
- Click the OK button twice to exit all dialog boxes.

42. From the **Define** menu select **Moving Load Cases** and then select **Vehicles...** to display the Define Vehicles dialog box.

43. In this dialog box:

- In the Click To area click the drop-down box and select Add Standard Vehicle to display the Standard Vehicle Data dialog box.
- In this dialog box:
  - In the Data Definition area select HSn-44 in the Vehicle Type drop-down box.
  - Type **20** in the Scale Factor edit box if it is not already there.
  - Click the **OK** button to return to the Define Vehicles dialog box.
- In the Click To area click the drop-down box and select Add Standard Vehicle to display the Standard Vehicle Data dialog box.
- In this dialog box:
  - In the Data Definition area select HSn-44L in the Vehicle Type drop-down box.
  - Type **20** in the Scale Factor edit box if it is not already there.
  - Click the **OK** button twice to exit all dialog boxes.

44. From the **Define** menu select **Moving Load Cases** and then select **Vehicles Classes...** to display the Define Vehicle Classes dialog box.

45. In this dialog box:

- Click the **Add New Class** button to display the Vehicle Class Data dialog box.
- In this dialog box:
  - Accept the default Vehicle Class Name, VECL1
  - Verify that HSn441 is in the Vehicle Name drop-down box.
  - Enter **1** in the Scale Factor edit box if it is not already there.
  - Click the **Add** button.

- Select HSn442 in the Vehicle Name drop-down box.
- Click the **Add** button.
- Click the **OK** button twice to exit all dialog boxes.

46. From the **Define** menu select **Moving Load Cases** and then select **Bridge Responses...** to display the Bridge Response Request dialog box.

47. In this dialog box:

- In the Type of Results area check all four of the check boxes if they are not already checked.
- In the Method of Calculation area select the Exact option.
- Click the **OK** button.

48. From the **Define** menu select **Moving Load Cases** and then select **Moving Load Cases...** to display the Define Moving Load Cases dialog box.

49. In this dialog box:

- Click the **Add New Load** button to display the Moving Load Case Data dialog box.
- In this dialog box:
  - Accept the default Moving Load Case Name, MOVE1.
  - Click the **Add New Assign** button to display the Moving Load Case Assignment Data dialog box.
  - In this dialog box:
    - ✓ In the Assignment Lanes area click on LANE1 in the Select Lanes From list box to highlight (select) it.
    - ✓ Hold down the Ctrl key on the keyboard and click on LANE2 to add it to the selection.
    - ✓ Click the **Add** button to transfer these items to the Selected Lanes list box.
    - ✓ Click the **OK** button three times to exit all dialog boxes.

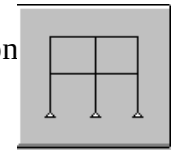
50. Note that the joint labeled A in the problem statement is labeled 10 on the screen.

51. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

52. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

53. 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.

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

55. 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.

56. From the **Display** menu select **Show Influence Lines...** and then **Joints...** to display the Show Joint Influence Line dialog box.

57. In this dialog box:

    - Select LANE1 from the Lane drop-down box if it is not already showing.
    - Type **10** in the Joint ID edit box.
    - In the Vector Type area select the Displacement option if it is not already selected.
    - In the Component area select the U3 option (vertical displacement).
    - Click the **OK** button to display the influence line.

    Note: This influence line is constructed with two output segments specified for the girder elements. Points for the influence line are calculated at the ends of each output segment. These points are then connected by straight lines. You can clearly see the segments in the influence line.

58. Click the **Member Force Diagram For Frames** button to display the Member Force Diagram For Frames dialog box.

59. In this dialog box:

    - Select MOVE1 Moving Load from the Load drop-down box.
    - In the Component area select the Moment 3-3 option.

- Uncheck the Fill Diagram check box.
- Check the Show Values On Diagram check box.
- Click the **OK** button to display the moment diagram.

Note: This moment diagram is constructed with two output segments specified for the girder elements. Points for the moment diagram are calculated at the ends of each output segment. These points are then connected by straight lines. You can clearly see the segments in the moment diagram.

60. Click the **Lock/Unlock Model** button  and click the resulting **OK** button to unlock the model.

61. Select frame elements 8 through 13 (i.e., the girder elements).

62. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

63. In this dialog box:
    - Type **10** in the Number of Segments edit box.
    - Click the **OK** button.

64. Click the **Show Undeformed Shape** button  to remove the displayed output segment assignments.

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

66. 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.

67. From the **Display** menu select **Show Influence Lines...** and then **Joints...** to display the Show Joint Influence Line dialog box.

68. In this dialog box click the OK button to display the influence line.

    Note: This influence line is much sharper than the previous one.

69. Click the **Member Force Diagram For Frames** button  to display the Member Force Diagram For Frames dialog box.

70. In this dialog box click the **OK** button to display the moment diagram.

    Note: This moment diagram is much sharper than the previous one.