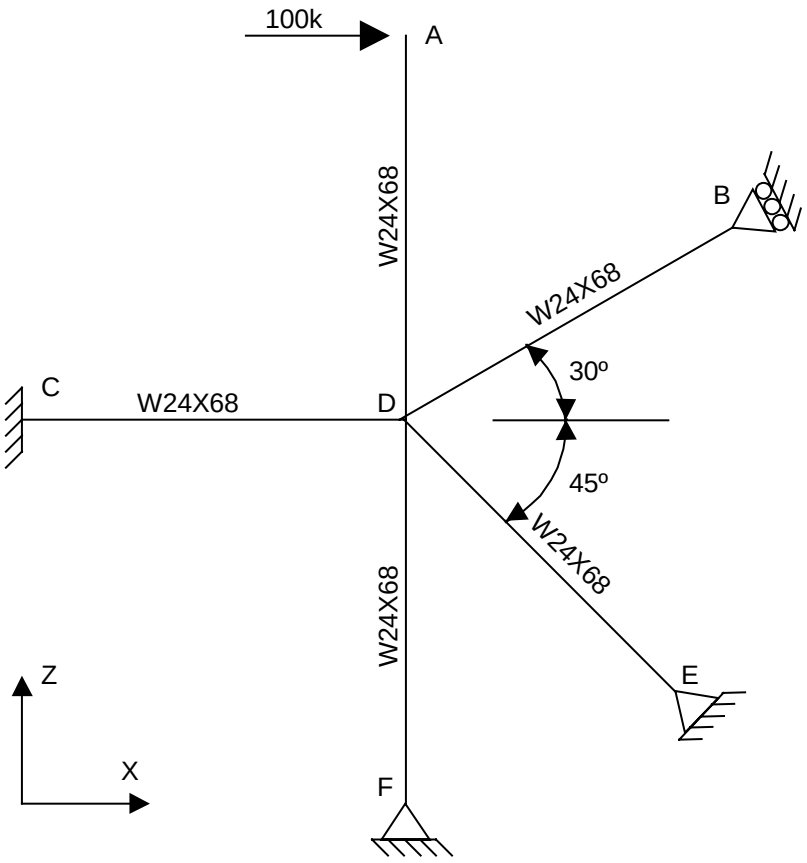
Problem D

Inclined Supports

Steel
E =29000 ksi, Poissons Ratio = 0.3
All members are 10 feet long.

To Do
Determine support reactions.
Determine X-direction displacements
at joints A and B.

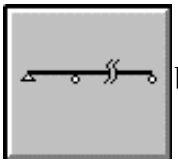


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem D 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 **2** in the Number of Spans edit box.
   - Type **10** in the Span Length edit box.
   - Uncheck the Restraints 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 Frames area.
   - Click the **OK** button.

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

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

10. 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.

11. 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 to exit all dialog boxes.

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

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. Highlight Sections.pro and click the **Open** button.

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 W24X68 section.
    - Click the **OK** button three times.

17. Select frame elements 1 and 2.

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

19. Highlight W24X68 in the Frame Sections area and click the **OK** button.

20. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.

21. Select frame element 2 by clicking on it.

22. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

23. In this dialog box:

    - Click the Radial Tab.
    - Choose the Y Axis option in the Rotate about area. Note it will rotate about the Y-axis at the origin.
    - Type **45** in the Angle edit box in the Increment Data area.
    - Type **1** in the Number edit box in the Increment Data area.
    - Click the **OK** button.

24. Select frame element 2 by clicking on it.

25. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

26. In this dialog box:
    - Click the Radial Tab.
    - Type **90** in the Angle edit box in the Increment Data area.
    - Click the **OK** button.
27. Select frame element 2 by clicking on it.
28. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
29. In this dialog box:
    - Click the Radial Tab.
    - Type **270** in the Angle edit box in the Increment Data area.
    - Click the **OK** button.
30. Select frame element 2 by clicking on it.
31. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
32. In this dialog box:
    - Click the Radial Tab.
    - Type **330** in the Angle edit box in the Increment Data area.
    - Click the **OK** button.
33. Select frame element 2 by clicking on it.
34. Press the Delete key on the keyboard to delete this member.
35. Click the **Refresh Window** button  to refresh the drawing.
36. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.
37. In this dialog box:
    - Check the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

38. Select joint 4.

39. From the **Assign** menu select **Joint** and then **Local Axes…** from the submenu to display the Joint Local Axis dialog box.

40. In this dialog box:

    - Type **-45** in the about Y' edit box.

    - Press the **OK** button.

41. Select joint 7.

42. From the **Assign** menu select **Joint** and then **Local Axes…** from the submenu to display the Joint Local Axis dialog box.

43. In this dialog box:

    - Type **-120** in the about Y' edit box.

    - Press the **OK** button.

44. Select joint 1.

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

46. In this dialog box:

    - Check all six boxes in the Restraints in Local Directions area.

    - Click the **OK** button.

47. Select joints 4 and 5.

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

49. In this dialog box:

    - In the Restraints in Local Directions area uncheck the three Rotation boxes and leave the three Translation boxes checked.

    - Click the **OK** button.

50. Select joint 7.

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

52. In this dialog box:

    - In the Restraints in Local Directions area uncheck the Translation 3 box and leave the Translation 1 and Translation 2 boxes checked.
    - Click the **OK** button.

53. Select joint 6.

54. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

55. In this dialog box:

    - Type **100** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

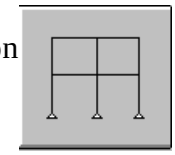
56. Click the **Show Undeformed Shape** button  to remove the displayed joint force assignments.

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

58. In this dialog box:

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

59. 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.

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

61. 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.

62. Right click on the joints labeled A and B in the problem statement to see their displacements.

63. Click the **Joint Reaction Forces** button  to display the Joint Reaction Forces dialog box.

64. In this dialog box:

- Verify that the Reactions option is selected in the Type area.

- Click the **OK** button.

65. The reactions are displayed on the screen. You can right click on any joint to see the reactions at that joint or you can just read the reactions on the screen. If the text is too small to read, you can zoom in, or you can change the minimum font size as described in the note below.

    *Note: To change the minimum font size select **Preferences** from the **Options** menu and make sure the Dimensions Tab is selected. In the Minimum Graphic font Size edit box input a new size, maybe 5 or 6 points. Click the **OK** button.*