

Problem M

Flat Plate In The X-Y Plane With A Twist

Concrete

$E = 3600$ ksi, Poissons Ratio = 0.2

Available Degrees of Freedom

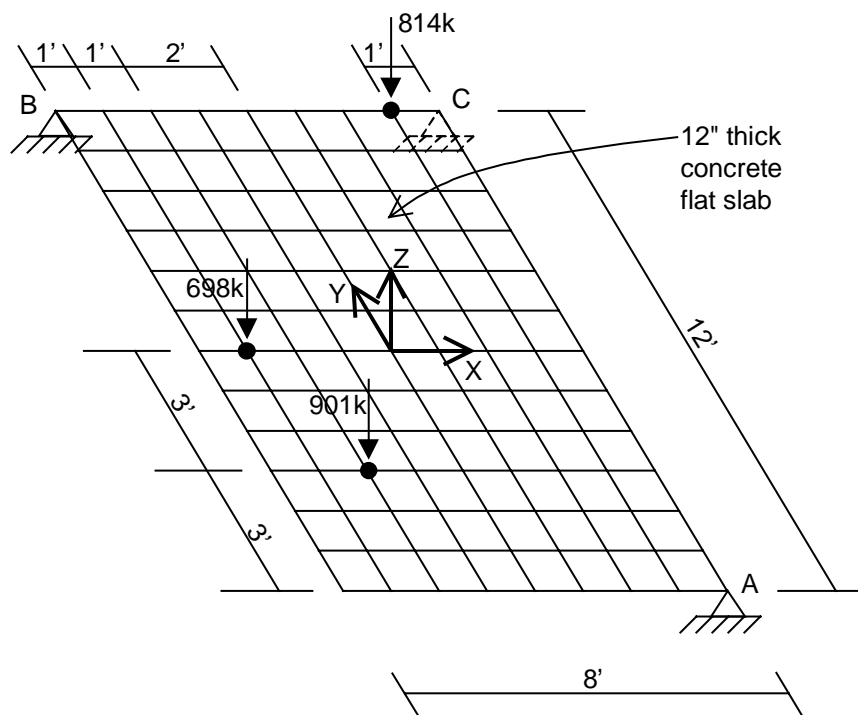
UZ, RX, RY

Supports

Joints A, B and C have Z-direction supports, as shown.





To Do

Determine support reactions at joints A, B and C. Explain the apparently odd results for the reaction at joint C.







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 M Solution


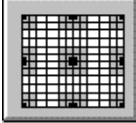



1. Click the drop down box in the status bar to change the units to kip-ft. 
2. From the **File** menu select **New Model...** This displays the Coordinate System Definition dialog box.
3. In this dialog box
 - Select the Cartesian Tab.
 - In the Number of Grid Spaces area type **2** in the X direction edit box.
 - In the Number of Grid Spaces area type **2** in the Y direction edit box.
 - In the Number of Grid Spaces area type **0** in the Z direction edit box.
 - In the Grid Spacing area verify the X direction spacing is 4.
 - In the Grid Spacing area type **6** the Y direction edit box.
 - Click the **OK** button.
4. Click the “X” in the top right-hand corner of the 3-D View window to close it.
5. Verify that the Snap to Joints and Grid Points button  on the side toolbar is depressed.
6. Click the **Draw Rectangular Shell Element** button  on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
7. Click on upper left-hand corner grid intersection (coordinates are (-4, 6, 0)) and then click on the lower right-hand grid intersection (coordinates are (4, -6, 0)) to draw a shell element over the entire structure.
8. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.
9. Click on the shell element to select it.
10. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.
11. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.
12. Select the joints that are labeled “A”, “B” and “C” in the problem statement.



13. From the **Assign** menu select **Joints** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.
14. In this dialog box:
 - Uncheck the Translation 1 and Translation 2 check boxes.
 - Verify that the Translation 3 check box is checked.
 - Verify that the Rotation about 1, 2 and 3 check boxes are *not* checked.
 - Click the **OK** button.
15. Click the **Show Undeformed Shape** button  to remove the display of joint restraints and reset the window display (and title).
16. Click the drop down box in the status bar to change the units to kip-in. 
17. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
18. Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
19. In this dialog box:
 - Verify that the Modulus of Elasticity is 3600.
 - Verify that the Poisson's Ratio is 0.2.
 - Click the **OK** button twice to exit the dialog boxes.
20. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.
21. In this dialog box:
 - Click the Modify/Show Section button to display the Shell Sections dialog box.
 - In this dialog box:
 - Verify that the selected material is CONC.
 - Verify that both the Membrane and the Bending thicknesses are 12.
 - Verify that the Shell option is selected in the Type area.
 - Click the **OK** button twice to exit all dialog boxes.
22. Click the drop down box in the status bar to change the units to kip-ft. 

23. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
24. In this dialog box:
 - Check the Labels box in the Joints area.
 - Click the **OK** button.

Note: If the font size is too small for you to read the joint labels 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.
25. Select joint 106 (coordinates (3, 6, 0)) by clicking on it.
26. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
27. In this dialog box:
 - Type **-814** in the Force Global Z edit box in the Loads area.
 - Click the **OK** button.
28. Select joint 16 (coordinates (-3, 0, 0)) by clicking on it.
29. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
30. In this dialog box:
 - Type **-698** in the Force Global Z edit box in the Loads area.
 - Click the **OK** button.
31. Select joint 32 (coordinates (-2, -3, 0)) by clicking on it.
32. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
33. In this dialog box:
 - Type **-901** in the Force Global Z edit box in the Loads area.
 - Click the **OK** button.
34. Click the **Show Undeformed Shape** button  to reset the window display.

35. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
36. In this dialog box:
- Uncheck the Labels box in the Joints area.
 - Click the **OK** button.
37. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
- In this dialog box click the **Plane Grid XY Plane** button  to set the available degrees of freedom.
 - Click the **OK** button.
38. Click the 3D View button  on the main toolbar to switch to a 3-D View.
39. Click the **Run Analysis** button  to run the analysis.
40. 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.
41. Click the **Joint Reaction Forces** button  on the main toolbar to display the Joint Reaction Forces dialog box.
42. In this dialog box:
- Verify that the Reactions option is selected in the Type area.
 - Click the **OK** button and view the support reactions.

Note: The reaction at the joint labeled “C” in the problem statement is zero (0). The reason for this apparently odd result is that the resultant of all the applied loads lies on a line connecting the support points labeled “A” and “B”, and thus by simple statics the reaction at support point “C” must be zero. Note that you could move the support point labeled “C” anywhere on the structure (except on the line connecting support points “A” and “B”, since this would result in an unstable structure) and the resulting reactions would not change.