Problem M

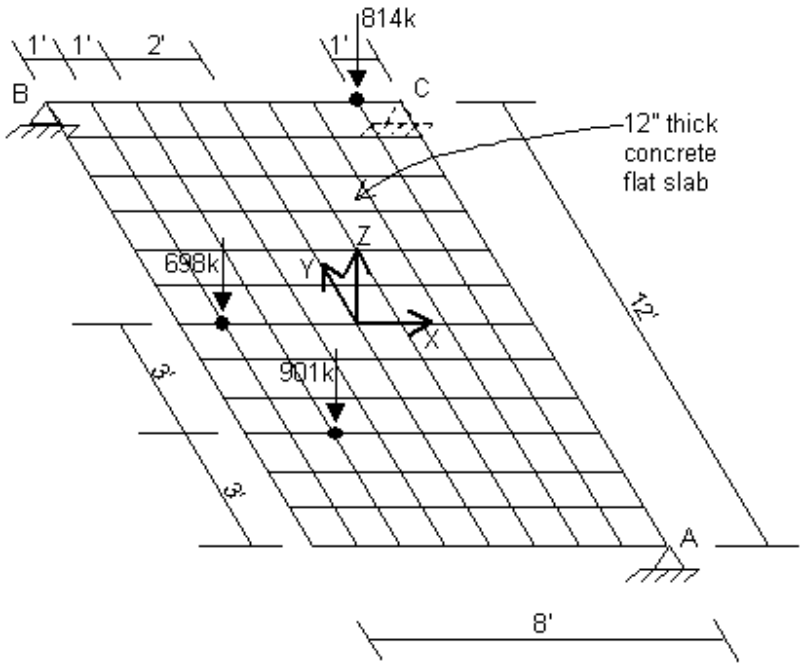
Flat Plate In The X-Y Plane With A Twist

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.



CSI Solution Demonstrates Use of These Features

- Mesh Area Objects
- "Trick" Problem

Problem M Solution

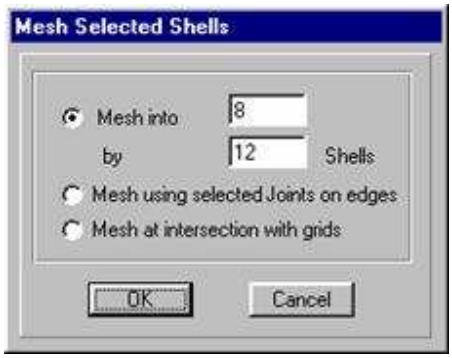
1. Click the **File menu > New Model** command to display the ***New Model*** form.
2. Click the drop-down box to set the units to Kip, ft, F.

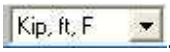
3. Click on the **Grid Only** button 

to display the ***New Coord/Grid System*** form. In that form:
    - Select the Cartesian Tab.
    - In the *Number of Grid Lines* area, type **3** in the *X direction* edit box.
    - In the *Number of Grid Line*s area, type **3** in the *Y directio*n edit box.
    - In the *Number of Grid Lines* area, type **1** in the *Z direction* edit box.
    - In the *Grid Spacing* area, type **4** the *X direction* edit box.
    - 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. Click the drop-down box in the status bar to change the units to 

.
6. Click the **Define menu > Materials** command to display the ***Define Materials*** form.
7. Highlight the CONC material and click the **Modify/Show Material** button to display the ***Material Property Data*** form. In that form:
    - Verify that the *Modulus of Elasticity* is 3600.
    - Verify that the *Poisson's Ratio* is 0.2.
    - Click the **OK** buttons on the ***Material Property Data*** and ***Define Materials*** forms to close the forms.
8. Click the **Define menu > Area Sections** command to display the ***Area Sections*** form. In that form:
    - Click the **Modify/Show Section** button to display the ***Area Section Data*** form. In that form:
        - Verify that the selected *Material Name* is CONC.
        - Verify that the *Shell* option is selected in the *Area Type* area.
        - 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** buttons on the ***Area Section Data*** and ***Area Sections*** forms to close all forms.
9. Verify that the **Snap to Points and Grid Intersections** button  on the side toolbar is depressed.
10. Click the **Draw Rectangular Area** button  on the side toolbar or the **Draw menu > Draw Rectangular Area** command to display the ***Properties of Object*** form. In that form:
    - Verify that the selected Property is *ASEC1*.

12. Click on upper left-hand corner grid intersection (Point “B” in the problem statement) and then click on the lower right-hand grid intersection (Point “A”) to draw an area object over the entire structure.
13. Click the **Set Select Mode** button  to exit the Draw mode and enter Select mode.
14. Click on the area object to select it.
15. Click the **Edit menu > Mesh Areas** command to display the ***Mesh Selected Shells*** form.
16. Fill in this form as shown in the adjacent figure and click the **OK** button.



17. Select the joints that are labeled “A,” “B” and “C” in the problem statement.
18. Click the **Assign menu > Joint > Restraints** command to display the ***Joint Restraints*** form. In that form:
    - 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.
18. Click the **Show Undeformed Shape** button  to remove the display of joint restraints and reset the window display (and title).
19. Click the drop-down box in the status bar to change the units to  .

20. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the ***Display Options for Active Window*** form. In that form:
    - Check the *Labels* box in the *Joints* area.
    - Click the **OK** button.
21. Select joint 106 (coordinates 7, 12, 0) by clicking on it.
22. Click the **Assign menu > Joint Loads > Forces** command to display the ***Joint Forces*** form. In that form:
    - Type **-814** in the *Force Global Z* edit box in the *Loads* area.
    - Click the **OK** button.
23. Select joint 16 (coordinates 1, 6, 0) by clicking on it.
24. Click the **Assign menu > Joint Loads > Forces** command to display the ***Joint Forces*** form. In that form:
    - Type **-698** in the *Force Global Z* edit box in the *Loads* area.
    - Click the **OK** button.

25. Select joint 32 (coordinates 2, 3, 0) by clicking on it.
26. Click the **Assign menu > Joint Loads > Forces** command to display the ***Joint Forces*** form. In that form:
    - Type **-901** in the *Force Global Z* edit box in the *Loads* area.
    - Click the **OK** button.
27. Click the **Show Undeformed Shape** button  to reset the window display.
28. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the ***Display Options for Active Window*** form. In that form:
    - Uncheck the *Labels* box in the *Joints* area.
    - Click the **OK** button.
29. Click the **Analyze menu > Set Analysis Options** command to display the ***Analysis Options*** form. In that form:
    - Click the **Plane Grid XY Plane** button 

to set the available degrees of freedom.
    - Click the **OK** button.
30. Click the **Set Default 3D View** button 

to switch to a 3-D View.
31. Click the **Run Analysis** button  to display the ***Set Analysis Cases to Run*** form. In that form:
    - Highlight (select) *MODAL* in the *Case Name* list and click the **Run/Do Not Run Case** button.
    - Verify that the *DEAD* analysis case is set to *Run* in the *Action* list.
    - Click the **Run Now** button to run the analysis.
32. When the analysis is complete check the messages in the ***SAP Analysis Monitor*** window (there should be no warnings or errors) and then click the **OK** button to close the window.
33. Click the **Display menu > Show Forces/Stresses > Joints** command to display the ***Joint Reaction Forces*** form. In that form:
    - 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 virtually 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” because this would result in an unstable structure) and the resulting reactions would not change.