

Problem F

Wall Resisting Hydrostatic Pressure

Concrete

$E = 3600$ ksi, Poissons Ratio = 0.2

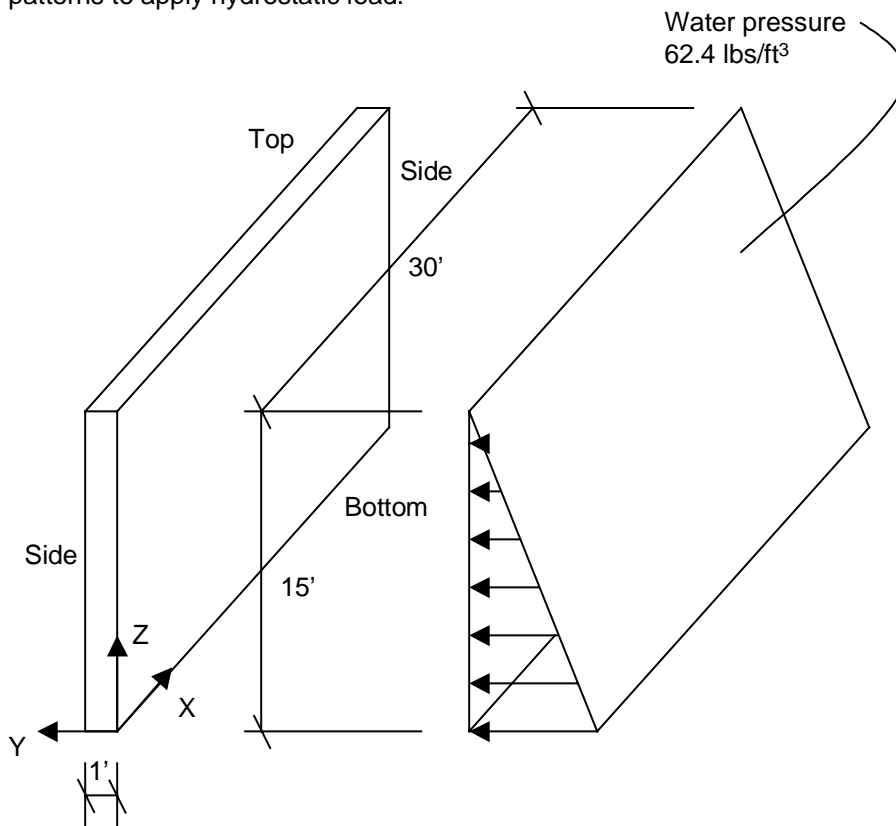
Boundary Conditions

Case 1: Wall clamped at bottom only.

Case 2: Wall clamped at bottom and sides.


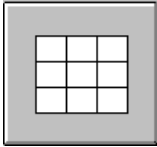



To Do


Determine maximum Y-direction displacements at top of wall for Case 1 and Case 2 support conditions. Use joint patterns to apply hydrostatic load.





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 F 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 **Shear Wall** template  button to display the Shear Wall dialog box.
4. In this dialog box
 - Type **30** in the Number of Spaces Along X edit box.
 - Type **15** in the Number of Spaces Along Z edit box.
 - Type **1** Space Width Along X edit box.
 - Type **1** Space Width Along Z edit 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 drop down box in the status bar to change the units to kip-in. 
7. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
8. In this dialog box:
 - Verify that the modulus of elasticity is 3600 and poisson’s ratio is 0.2.
 - Click the **OK** button twice to exit the dialog boxes.
9. Click the drop down box in the status bar to change the units to kip-ft. 
10. Select all of the support joints at the bottom of the wall by “windowing”.
11. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
12. In this dialog box:
 - Click the fixed base fast restraint button  to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.

- Click the **OK** button.
13. From the **Define** menu select **Joint Patterns...** to display the Define Pattern Names dialog box.
 14. In this dialog box:
 15. Type **HYDRO** in the edit box in the Patterns area.
 16. Click the **Add New Pattern Name** button
 17. Click the **OK** button.
 18. Click the **Select All** button  on the side tool bar.
 19. From the **Assign** menu select **Joint Patterns...** to display the Pattern Data dialog box.
 20. In this dialog box:
 - Select HYDRO from the Pattern Name drop-down box.

Note: Press the F1 key on the keyboard for context sensitive help on the dialog box illustrating the definition of the Constants. When finished reading the help, click the "X" in the top right-hand corner of the Help window to close it.

 - Type **-1** in the Constant C edit box.
 - Type **15** in the Constant D edit box.
 - Click the **OK** button.
 21. Click the **Select All** button  on the side tool bar.
 22. From the **Assign** menu select **Shell Static Loads...** and select **Pressure...** from the submenu to display the Shell Pressure Loads dialog box.
 23. In this dialog box:
 - Select the By Joint Pattern option.
 - Select HYDRO from the Pattern drop-down box.
 - Type **.0624** in the Multiplier edit box.
 - Click the **OK** button.
 24. Click the **Show Undeformed Shape** button  to remove the displayed joint force assignments.

25. Click the **Run Analysis** button  to run the analysis.
26. 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.
27. Right click on the center joint at the top of the wall and note the Y-direction displacement.
28. Click the **Lock/Unlock Model** button  and click the resulting **OK** button to unlock the model.
29. Select the joints along the sides of the model by “windowing” each side separately.
30. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
31. In this dialog box:
 - Click the fixed base fast restraint button  to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.
 - Click the **OK** button.
32. Click the **Run Analysis** button  to run the analysis.
33. 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.
34. Right click on the center joint at the top of the wall and note the Y-direction displacement.