

Problem F

Wall Resisting Hydrostatic Pressure

Concrete

$E = 3600$ ksi, Poissons Ratio = 0.2

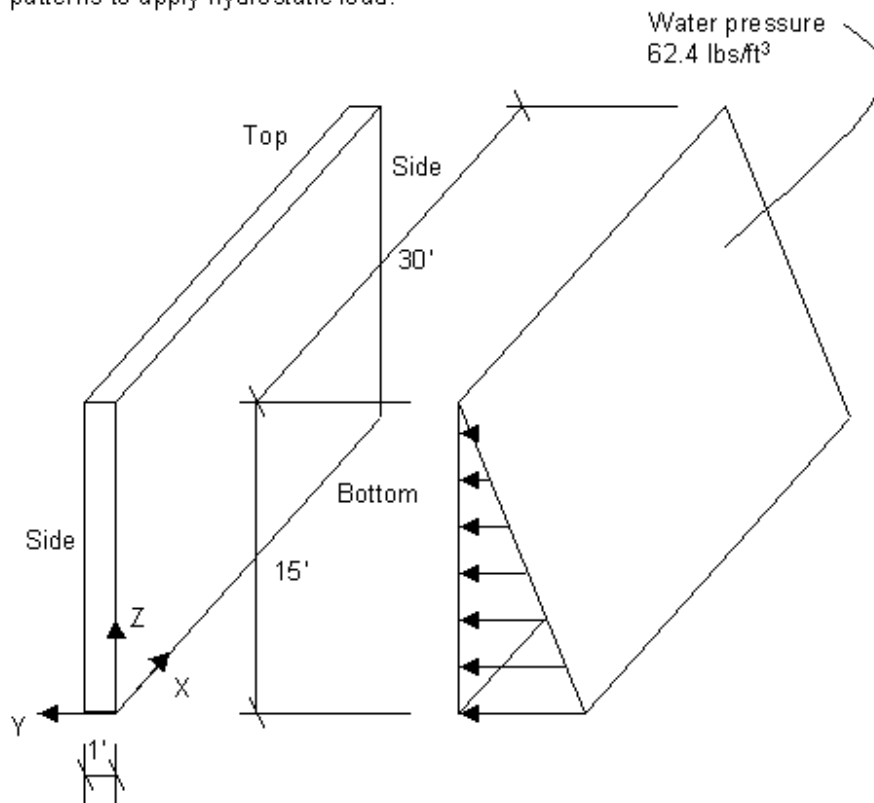
Boundary Conditions

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.

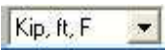


CSI Solution Demonstrates Use of These Features

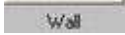
- Hydrostatic Loading
- Joint Patterns

Problem F Solution

1. Click the **File menu > New Model** command to display the **New Model** form.

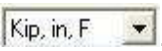
2. Click the drop-down box to set the units to .



3. Click on the **Wall** template  button to display the **Shear Wall** form. In that form:

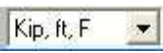
- Type **30** in the *Number of Divisions, X* edit box.
- Type **15** in the *Number of Divisions, Z* edit box.
- Type **1** in the *Division Space, X* edit box.
- Type **1** in the *Division Space, Z* edit box.
- Click the **OK** button.

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. 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 and Poisson’s ratio is 0.2.
- Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to close the forms.

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

8. Select all of the support joints at the bottom of the wall by “windowing.”

9. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:

- 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.

10. Click the **Define menu > Joint Patterns** command to display the **Define Pattern Names** form. In that form:







- Type **HYDRO** in the edit box in the *Patterns* area.
- Click the **Add New Pattern Name** button.
- Click the **OK** button.

11. Click the **Select All** button  on the side tool bar.

12. Click the **Assign menu > Joint Patterns** command to display the **Pattern Data** form. In that form:

- Select HYDRO from the *Pattern Name* drop-down list.

Note: Press the F1 key on the keyboard for context sensitive help on the form 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.
13. Click the **Select All** button .
14. Click the **Assign menu > Area Loads > Surface Pressure (All)** command to display the **Area Surface Pressure Load** form. In that form:
- Select the *By Joint Pattern* option.
 - Select HYDRO from the *Pattern* drop-down list.
 - Type **.0624** in the *Multiplier* edit box.
 - Click the **OK** button.
15. Click the **Show Undeformed Shape** button  to remove the displayed joint force assignments.
16. 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.
17. 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.
18. Right click on the center joint at the top of the wall and note the Y-direction displacement.
19. Click the **Lock/Unlock Model** button  and click the resulting **OK** button to unlock the model.
20. Select the joints along the sides of the model by “windowing” each side separately.
21. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:
- 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.
22. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form, click the **Run Now** button to run the analysis.

23. 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.
24. Right click on the center joint at the top of the wall and note the Y-direction displacement.