Problem L

Periodic Loading

Steel

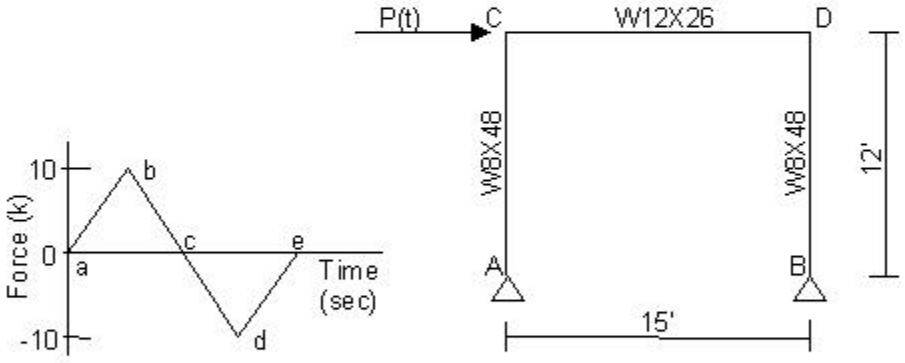
E = 29000 ksi, Poissons Ration = 0.3
Pinned base
All beam-column connections are rigid

Joint Masses

Lumped mass at joints C and D is 0.02 kip-sec^2 / in

Loading

The load P(t), applied to Joint C, is a periodic load. Three different loading cases (functions) are defined for P(t). The three loading functions, which have periods of 0.25, 0.50, and 1.00 second, respectively, are shown in the chart and graph below. Assume 5% damping for all loading.



Point	Force (k)	Time Funcion 1 (sec)	Time Function 2 (sec)	Time Function 3 (sec)
a	0	0	0	0
b	10	0.0625	0.125	0.25
c	0	0.125	0.25	0.5
d	-10	0.1875	0.375	0.75
e	0	0.25	0.5	1

Note: The periods of time functions 1, 2, and 3 are 0.25, 0.5, and 1 second, respectively.

To Do

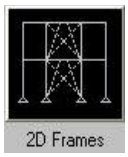
1. Verifty the natural period of the structure is approximately 0.50 second.
2. Determine displacement at joint D for the three periodic functions.

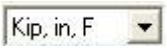
CSI Solution Demonstrates Use these Features

- Mode Shapes
- Modal Time History Analysis (Periodic)

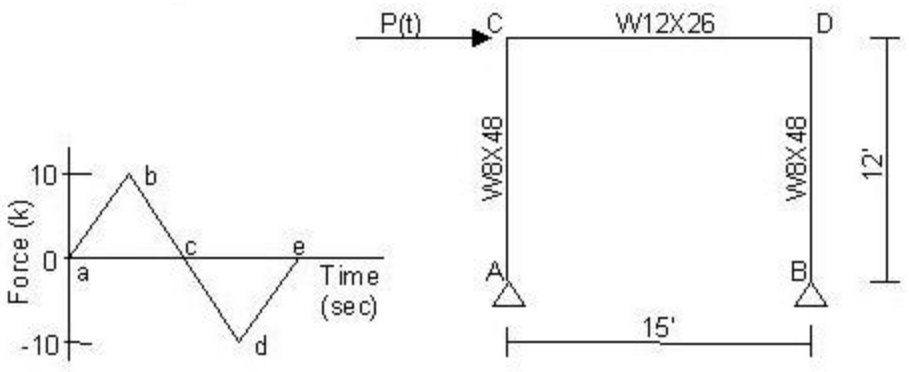
Problem L Solution

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

3. Click the **2D Frame** button 

to access the ***2D Frames*** form. In that form:
   - Select *Portal* in the *2D Frame Type* drop-down list.
     - Type **1** in the *Number of Stories* edit box.
     - Type **1** in the *Number of Bays* edit box.
     - Type **15** in the *Bay Width* edit box.
     - Click the + (plus) symbol beside the *Beams* or *Columns* drop-down list to access the ***Frame Properties*** form.
       - Click the **Import New Property** button to access the ***Import Frame Section Property*** form. Select *Steel* in the *Frame Section Property Type* drop-down list and click the **I/Wide Flange** button to access the ***Section Property File*** form.
       - Select the SECTIONS.PRO file and click the **Open** button to display a ***database*** form of available sections.
         - On the ***database*** form, click the + (plus) symbol beside the drop-down list to access the ***Define Materials*** form.
           - Highlight the **A992Fy50** definition in the *Materials* display list and click the **Modify/Show Material** button to access the ***Material Property Data*** form.
             - Change the Units to 

.
             - Verify that the *Modulus of Elasticity* is 29000 and *Poisson's Ratio* is 0.3.
           - Click the **OK** buttons on the ***Material Property Data*** and ***Define Materials*** forms to close the forms and return to the ***database*** form.
         - Ensure that the **A992Fy50** definition is selected in the Materials drop-down list.
         - Scroll down the list of sections, and while holding down the Ctrl key, click on **W8X48** and **W12X26** to select them.
         - Click the **OK** button to close the ***database*** form and display the ***I/Wide Flange Section*** form.
     - Click the **OK** buttons on the ***I/Wide Flange Section*** and ***Frame Properties*** forms to return to the ***2D Frames*** form.
   - Select the **W12X26** section in the *Beams* drop-down list.
   - Select the **W8X48** section in the *Columns* drop-down list.
   - Click the **OK** button on the ***Frame Properties*** form to close the form and display the template model in the program windows.
4. Click the "X" in the top right-hand corner of the 3-D View window to close it.
5. Select the joints labeled C and D in the problem statement.



6. Click the **Assign menu > Joint > Masses** command to access the ***Joint Masses*** form. In that form:
   - Type **.02** in the *Direction 1* edit box.
   - Type **.02** in the *Direction 3* edit box.
   - Click the **OK** button.
7. Click the **Show Undeformed Shape** button to remove the displayed joint mass assignments.
8. Select the joint labeled C in the problem statement.
9. Click the **Assign menu > Joint Loads > Forces** command to access the ***Joint Forces*** form. In that form:
   - Type **1** in the *Force Global X* edit box.
   - Click the **OK** button.
10. Click the **Show Undeformed Shape** button to remove the displayed joint force assignments.
11. Click the **Define menu > Functions > Time History** command to access the ***Define Time History Functions*** form. In that form:
   - In the *Choose Function Type to Add* area, click the drop-down list that reads *Sine Function* and then click on the *User Function* item.
   - Click the **Add New Function** button to access the ***Time History Function Definition*** form. In that form:
     - Accept the default *FUNC1* Function Name.
     - Type **.0** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
     - Type **.0625** in the *Time* edit box, type **10** in the *Value* edit box, and click the **Add** button.
     - Type **.125** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
     - Type **.1875** in the *Time* edit box, type **-10** in the *Value* edit box, and click the **Add** button.
     - Type **.25** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
     - Click the **OK** button to return to the ***Define Time History Functions*** form.
   - Click the **Add New Function** button to access the ***Time History Function Definition*** form. In that form:
     - Accept the default *FUNC2* Function Name.
     - Type **.0** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
     - Type **.125** in the *Time* edit box, type **10** in the *Value* edit box, and click the **Add** button.

- Type **.25** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
- Type **.375** in the *Time* edit box, type **-10** in the *Value* edit box, and click the **Add** button.
- Type **.5** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
- Click the **OK** button to return to the ***Define Time History Functions*** form.

- Click the **Add New Function** button to access the ***Time History Function Definition*** form. In that form:
  - Accept the default *FUNC3 Function Name*.
  - Type **.0** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
  - Type **.25** in the *Time* edit box, type **10** in the *Value* edit box, and click the **Add** button.
  - Type **.5** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
  - Type **.75** in the *Time* edit box, type **-10** in the *Value* edit box, and click the **Add** button.
  - Type **1** in the *Time* edit box, type **0** in the *Value* edit box, and click the **Add** button.
  - Click the **OK** buttons on the ***Time History Function Definition*** and ***Define Time History Functions*** forms to close all of the forms.

12. Click the **Define menu > Load Cases** command to access the ***Define Load Cases*** form. In that form:
    - Highlight (select) *MODAL* in the *Case Name* list and click the **Modify/Show Case** button to display the ***Analysis Case Data - Modal*** form. In that form:
      - Type **4** in the *Maximum Number of Modes* edit box.
      - Click the **OK** button to return to the ***Load Cases*** form.
    - Click the **Add New Load Case** button to access the ***Load Case Data - Linear Static*** form. In that form:
      - Accept the default *Load Case Name*, *ACASE1*.
      - Select **Time History** from the *Load Case Type* drop-down list.
      - In the *Time History Motion Type* area, select the **Periodic** option.
      - In the *Loads Applied* area, verify that *FUNC1* is selected in the *Function* drop-down list and then click on the **Add** button.
      - In the *Time Step Data* area, type **25** in the *Number of Output Time Steps* edit box.
      - In the *Time Step Data* area, type **.01** in the *Output Time Step Size* edit box.
      - Click the **OK** button to return to the ***Define Load Cases*** form.
    - Click the **Add New Load Case** button to display the ***Load Case Data - Linear Static*** form. In that form:
      - Accept the default *Load Case Name*, *ACASE2*.
      - Select **Time History** from the *Load Case Type* drop-down list.
      - In the *Time History Motion Type* area, select the **Periodic** option.
      - In the *Loads Applied* area, select *FUNC2* from the *Function* drop-down list and click on the **Add** button.
      - In the *Time Step Data* area, type **50** in the *Number of Output Time Steps* edit box.

- In the *Time Step Data* area, type **.01** in the *Output Time Step Size* edit box.
- Click the **OK** button to return to the ***Define Load Cases*** form.

- Click the **Add New Load Case** button to access the ***Load Case Data*** form. In that form:
  - Accept the default *Load Case Name*, *ACASE3*.
  - Select **Time History** from the *Load Case Type* drop-down list.
  - In the *Time History Motion Type* area, select the ***Periodic*** option.
  - In the *Loads Applied* area, select *FUNC3* from the *Function* drop-down list and click on the **Add** button.
  - In the *Time Step Data* area, type **100** in the *Number of Output Time Steps* edit box.
  - In the *Time Step Data* area, type **.01** in the *Output Time Step Size* edit box.
  - Click the **OK** buttons on the ***Load Case Data*** and ***Define Load Cases*** forms to close all forms.

13. Click the **Analyze menu > Set Analysis Options** command to access the ***Analysis Options*** form.

- In that form click the **Plane Frame XZ Plane** button 

to set the available degrees of freedom.
- Click the **OK** button.

14. Click the **Run Analysis** button to display the ***Set Load Cases to Run*** form. In that form:
- Verify that the *DEAD* load case is set to *Run* in the *Action* list.
- Verify that the *MODAL* load case is set to *Run* in the *Action* list.
- Verify that the *ACASE1* load case is set to *Run* in the *Action* list.
- Verify that the *ACASE2* load case is set to *Run* in the *Action* list.
- Verify that the *ACASE3* load case is set to *Run* in the *Action* list.
- Click the **Run Now** button to run the analysis.

15. When the analysis is complete, check the messages in the ***SAP Analysis Monitor*** window (there should be no warnings or errors). Note in the messages that the first mode period is about 0.5 second. Click the **OK** button to close the window. Note again in the window title on the screen that the first mode period is about 0.5 second.

16. Select the joint labeled D in the problem statement.

17. Click the **Display menu > Show Tables** command to access the ***Choose Tables to Display*** form. In that form:

- Click on the **Select Load Cases** button to access the ***Select Load Cases*** form. In that form:
  - Click the **Clear All** button.
  - Click on the *ACASE1* case to highlight it.
  - Hold down the shift key on the keyboard and click on the *ACASE3* case. The ACASE1,

ACASE2, and ACASE3 cases should all be highlighted (selected) now.

- Click the **OK** button to close the form.

- In the *ANALYSIS RESULTS* area, click on the + (plus) associated with the *Joint Output* item to display the *Displacements* item.
- Click the + (plus) associated with *Displacements* item to display the *Table: Joint Displacements* and *Table: Joint Displacements (Absolute)* items.
- Click on the *Table: Joint Displacements* and *Table: Joint Displacements (Absolute)* items.
- Click the **OK** button to display the ***Joint Displacements*** form.

  Note that the maximum displacement occurs for ACASE2, as would be expected because the period of Function 2 is close to the first mode period of the structure.

- Click the **Done** button to close the table.

We have viewed the envelopes of the joint displacement at joint D. Now we will view the time histories of the displacement.

18. Click the **Display menu > Show Plot Functions** command to access the ***Plot Function Trace Display Definition*** form. In that form:
    - Click the **Define Plot Functions** button to access the ***Plot Functions*** form. In that form:
      - Highlight Joint 4.
      - Click the **Modify/Show Plot Function** button to access the ***Joint Plot Function*** form. In that form:
        - Verify that the *Displ* option is selected in the *Vector Type* area.
        - Verify that the *UX* option is selected in the *Component* area.
        - Click the **OK** button on the ***Joint Plot Function*** and ***Plot Functions*** form to return to the ***Plot Function Trace Display Definition*** form.
    - Verify that *ACASE1* is selected in the *Load Case* drop-down list.
    - Click on *Joint 4* in the *List of Functions* list box to select it.
    - Click the **Add** button to move Joint 4 to the *Vertical Functions* list box.
    - Click the **Display** button to display the time history.
      - Click the **OK** button to close the time history display and return to the ***Plot Function Trace Display Definition*** form.
    - Select ACASE2 in the *Load Case* drop-down list.
    - Click the **Display** button to display the time history.
      - Click the **OK** button to close the time history display and return to the ***Plot Function Trace Display Definition*** form.
    - Select ACASE3 in the *Load Case* drop-down list.
    - Click the **Display** button to display the time history.
      - Click the **OK** button to close the time history display and return to the ***Plot Function Trace Display Definition*** form.

- Click the **Done** button to close the ***Plot Function Trace Display Definition*** form.