Problem L

Periodic Loading

Steel

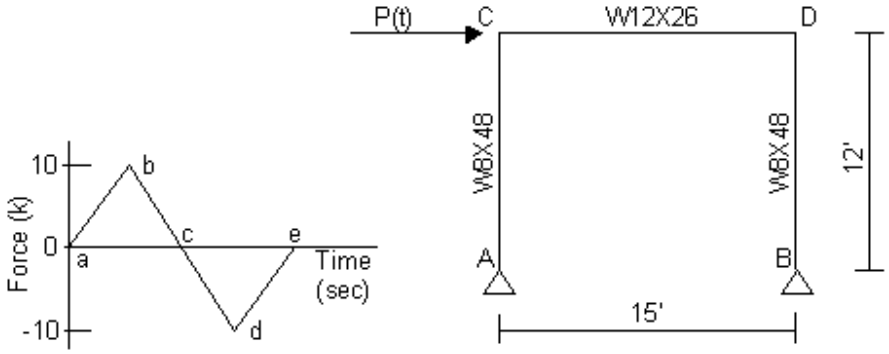
E=29000 ksi, Poissons Ratio = 0.3
Pinned base
All beam-column connections are rigid

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 seconds respectively, are shown in the chart and graph below. Assume 5% damping for all loading.



Point	Force (k)	Time Function 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 period of time functions 1, 2 and 3 is 0.25, 0.5 and 1 seconds respectively.

To Do

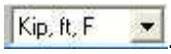
1. Verify natural period of structure is approximately 0.50 seconds.
2. Determine displacement at joint D for the three periodic functions.

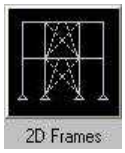
CSI Solution Demonstrates Use these Features

- Mode Shapes
- Modal Time History Analysis (Periodic)

Problem L Solution

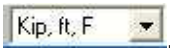
1. Click the **File menu > New Model** command to display the ***New Model*** form.

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

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

to display 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 **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 STEEL 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 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.
7. Click the **Define menu > Frame Sections** command to display the ***Frame Properties*** form.
8. In the *Choose Property Type to Add* area, click the drop-down box that reads *Import I/Wide Flange* and then click on the *Import I/Wide Flange* item.
9. In the *Click to* area, click the **Add New Property** button to display the ***Section Property File*** form. In that form:
    - Locate the Sections.pro file, which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.
    - A form appears with a list of all wide flange sections in the database. In that form:
        - Scroll down and click on the *W8X48* section.
        - Scroll down to the *W12X26* section, and click on it while holding down the Ctrl key on the keyboard.
        - Click the **OK** buttons on the database form, the ***I/Wide Flange Sections*** form and the ***Frame Properties*** forms to close all forms.
10. Select the beam object.
11. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the ***Frame Properties*** form. In that form:
    - Click on *W12X26* in the *Properties* area to highlight it.
    - Click the **OK** button.
12. Select the two column objects.

13. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the ***Frame Properties*** form. In that form:
    - Click on *W8X48* in the *Properties* area to highlight it.
    - Click the **OK** button.
14. Select the joints labeled C and D in the problem statement.
15. Click the **Assign menu > Joint > Masses** command to display 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.
16. Click the drop-down box in the status bar to change the units to 

.
17. Click the **Show Undeformed Shape** button  to remove the displayed joint mass assignments.
18. Select the joint labeled C in the problem statement.
19. Click the **Assign menu > Joint Loads > Forces** command to display the ***Joint Forces*** form. In that form:
    - Type **1** in the *Force Global X* edit box.
    - Click the **OK** button.
20. Click the **Show Undeformed Shape** button  to remove the displayed joint force assignments.
21. Click the **Define menu > Functions > Time History** command to display the ***Define Time History Functions*** form. In that form:
    - In the *Choose Function Type to Add* area, click the drop-down box that reads *Sine Function* and then click on the *User Function* item.
    - Click the **Add New Function** button to display 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 display 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 display 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.

22. Click the **Define menu > Analysis Cases** command to display the ***Analysis 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:
        - In the Number of Modes area, type **4** in the *Maximum Number of Modes* edit box.
        - Click the **OK** button to return to the ***Analysis Cases*** form.
    - Click the **Add New Case** button to display the ***Analysis Case Data - Linear Static*** form. In that form:
        - Accept the default *Analysis Case Name*, *ACASE1*.
        - Select *Time History* from the *Analysis Case Type* drop-down box.
        - 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 box and 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 ***Analysis Cases*** form.

- Click the **Add New Case** button to display the ***Analysis Case Data*** form. In that form:
  - Accept the default *Analysis Case Name*, *ACASE*2.
  - Select *Time History* from the *Analysis Case Type* drop-down box.
  - In the *Time History Motion Type* area, select the *Periodic* option.
  - In the *Loads Applied* area, select *FUNC2* from the *Function* drop-down box 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 ***Analysis Cases*** form.
- Click the **Add New Case** button to display the ***Analysis Case Data*** form. In that form:
  - Accept the default *Analysis Case Name*, *ACASE3*.
  - Select *Time History* from the *Analysis 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 box 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 ***Analysis Case Data*** and ***Analysis Cases*** forms to close all forms.

23. Click the **Analyze menu > Set Analysis Options** command to display 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.

24. Click the **Run Analysis** button to display the ***Set Analysis Cases to Run*** form. In that form:
    - Verify that the *DEAD* analysis case is set to *Run* in the *Action* list.
    - Verify that the *MODAL* analysis case is set to *Run* in the *Action* list.
    - Verify that the *ACASE1* analysis case is set to *Run* in the *Action* list.
    - Verify that the *ACASE2* analysis case is set to *Run* in the *Action* list.
    - Verify that the *ACASE3* analysis case is set to *Run* in the *Action* list.
    - Click the **Run Now** button to run the analysis.
25. 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 .5 second.
26. Select the joint labeled D in the problem statement.
27. Click the **Display menu > Show Tables** command to display the ***Choose Tables to Display*** form. In that form:
    - Click on the **Select Analysis Case** button to display the ***Select Output 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.

28. Click the **Display menu > Show Plot Functions** command to display the ***Plot Function Trace Display Definition*** form. In that form:

- Click the **Define Plot Functions** button to display the ***Plot Functions*** form. In that form:
    - Highlight Joint 4.
    - Click the **Modify/Show Plot Function** button to display 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 *Analysis Case* drop-down box.
- 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 *Analysis Case* drop-down 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 ACASE3 in the *Analysis Case* drop-down 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.
- Click the **Done** button to close the ***Plot Function Trace Display Definition*** form.