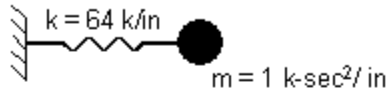


Problem Y

Response Spectrum Analysis For Single Degree of Freedom System

System



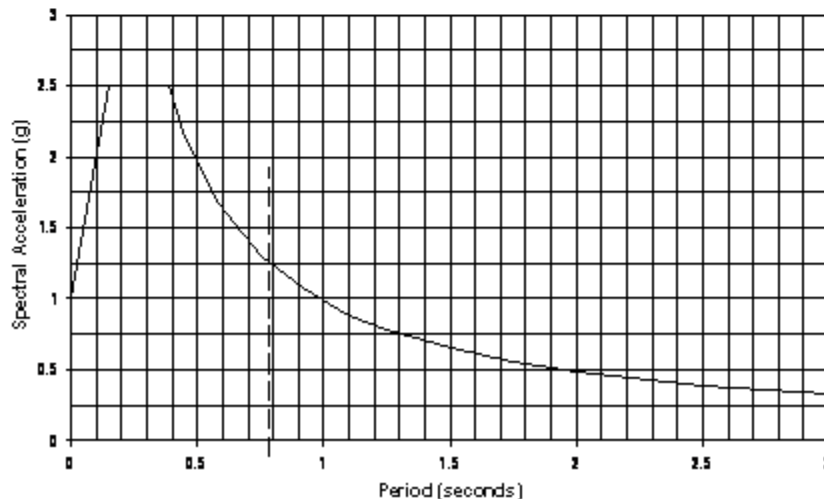
To Do

Perform a response spectrum analysis of this single degree of freedom system using the built-in 1994 UBC S1 spectrum. Compare the period with the calculated period below. Compare the spring force with the response spectrum below.

$$T = 2\pi \sqrt{\frac{m}{k}} = 2\pi \sqrt{\frac{1}{64}} = 0.7854 \text{ seconds}$$

$$\text{Note: } 1.25 \text{ g} \times \frac{386.4 \text{ in/sec}^2}{1 \text{ g}} = 483 \text{ in/sec}^2$$

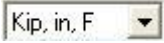
1994 UBC S1 Spectrum



CSI Solution Demonstrates Use of These Features

- Draw Special Joint
- Dynamic Analysis
- Mode Shapes
- Response Spectrum Analysis

Problem Y Solution

1. Click the **File menu > New Model** command to access the **New Model** form.
2. Verify that the units shown in the status bar are .



3. Click on the **Grid Only** button to access the **Quick Grid Lines** form. In that form:

- Select the *Cartesian* Tab.
- In the *Number of Grid Lines* area:
 - Type **1** in the *X direction* edit box.
 - Type **1** in the *Y direction* edit box.
 - Type **1** in the *Z direction* edit box.
- Click the **OK** button.

4. Click the “X” in the upper right-hand corner of the window labeled 3-D View to close it.



5. Click the **Draw Special Joint** button or the **Draw menu > Draw Special Joint** command to access the **Properties of Object** form.

6. Click on the axes intersection at the origin to enter a joint.



7. Click the **Set Select Mode** button to exit Draw mode and enter Select mode.
8. Select the joint by clicking on it.
9. Click the **Assign menu > Joint > Springs** command to access the **Joint Springs** form. In that form:
 - Type **64** in the *Translation 1* edit box.
 - Click the **OK** button.

10. Select the joint by clicking on it.


11. Click the **Assign menu > Joint > Masses** command to access the **Joint Masses** form. In that form:

- In the *Masses in Local Directions* area type **1** in the *Direction 1* edit box.
- Click the **OK** button.

12. Click the **Define menu > Functions > Response Spectrum** command to access the **Define Response Spectrum Functions** form. In that form:

- In the *Choose Function Type to Add* area, select the *UBC94 Spectrum* from the drop-down list.
- Click the **Add New Function** button to access the **Response Spectrum UBC 94 Function Definition** form. In that form:
 - Type **SPEC1** in the *Function Name* edit box.
 - In the *Parameters* area type **1** in the *Seismic Zone Factor, Z* edit box. This variable is being scaled up in excess of the required code value to give a peak ground acceleration of

1g.

- Select 1 from the *Soil Type* drop-down list.
 - Click the **OK** buttons on the **Response Spectrum UBC 94 Function Definition** and **Define Response Spectrum Functions** forms to exit all forms.
13. Click the **Define menu > Load Cases** command to access the **Define Load Cases** form. In that form:
- Click the **Add New Load Case** button to access the **Load Case Data** form. In that form:
 - Type **RS** in the *Load Case Name* edit box.
 - In the *Analysis Case Type* area, select *Response Spectrum* from the drop-down list.
 - Accept the default *Modal Combination* option, CQC.
 - Accept the default *Directional Combination* option, SRSS. Note that this option is irrelevant in this example since the response spectrum is run in only one direction.
 - In the *Loads Applied* area:
 - Verify that *U1* is shown in the *Load Name* drop-down list.
 - Verify that *SPEC1* is shown in the *Function* drop-down list.
 - Type **386.4** in the *Scale Factor* edit box.
 - Click the **Add** button.
 - Verify that *Modal Damping* is set to *Constant at 0.05*.
 - Click the **OK** button.
 - Click on MODAL in the *Load Case Name* list to highlight it.
 - Click the **Modify/Show Load Case** button to access the **Load Case Data - Modal** form. In that form:
 - Verify that the *Eigen Vectors* option is selected in the *Type of Modes* area.
 - In the *Number of Modes* area, type **1** in the *Maximum Number of Modes* edit box.
 - Click the **OK** buttons on the **Load Case Data - Modal** and **Define Load Cases** forms to exit all forms.
14. Click the **Analyze menu > Set Analysis Options** command to access the **Analysis Options** form.
- Uncheck all of the *Available DOFs* check boxes except for UX.
 - Click the **OK** button.
15. Click the **Run Analysis** button  to access the **Set Load Cases to Run** form. In that form:
- Click the **Run Now** button.
16. 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.
17. Click the **Display menu > Show Forces/Stresses > Joints** command to access the **Joint Reaction Forces** form. Click the **OK** button. If the axes make it difficult to read the spring force, click the **View menu > Show Axes** command to toggle the axes display off.

