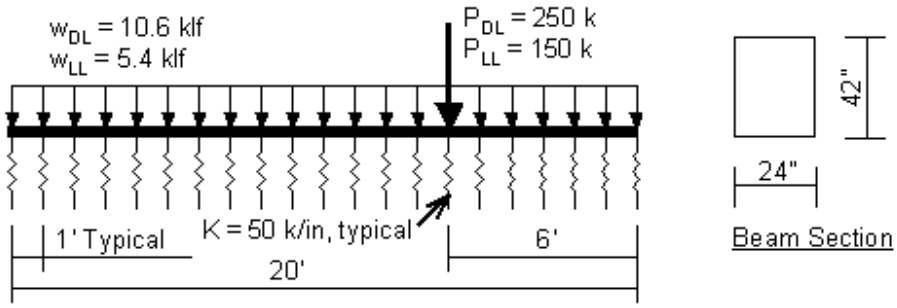
Problem J

Beam On Elastic Foundation

Concrete
E = 3120 ksi
Poissons Ratio = 0.2

To Do
Determine the moment diagram under combined dead plus live loads and the maximum downward displacement.

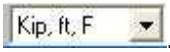
Note: Dead load shown does not include beam self weight.



CSI Solution Demonstrates Use of These Features

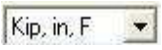
- Divide Frames
- Response Combinations
- Springs

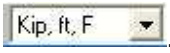
Problem J Solution

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 **Beam** button 

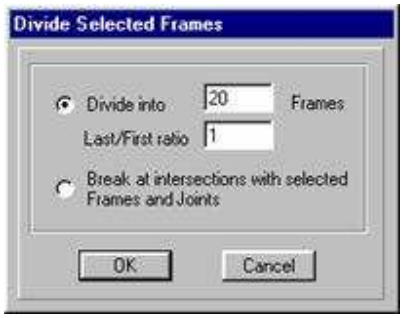
to display the ***Beam*** form. In that form:
    - Type **1** in the *Number of Spans* edit box.
    - Type **20** in the *Span Length* edit box.
    - Uncheck the *Restraints* check box
    - Click the **OK** button.

4. Click the “X” in the top right-hand corner of the 3-D View window to close it.
5. Click the **Define menu > Materials** command to display the ***Define Materials*** form.
6. Click on CONC in the *Materials* area to highlight it (select it), and then click the **Modify/Show Material** button to display the ***Material Property Data*** form. In that form:
   - Verify *0.15* is entered in the *Weight per Unit Volume* edit box.
   - Click the **OK** buttons on the ***Material Property Data*** and ***Define Materials*** forms to close all forms.
7. Click the drop-down box in the status bar to change the units to 

.
8. Click the **Define menu > Materials** command to display the ***Define Materials*** form.
9. Click on CONC in the *Materials* area to highlight it (select it), and then click the **Modify/Show Material** button to display the ***Material Property Data*** form. In that form:
   - Type **3120** in the *Modulus of Elasticity* edit box.
   - Verify .2 is entered in the *Poisson’s Ratio* edit box.
   - Accept the other default values.
   - Click the **OK** buttons on the ***Material Property Data*** and ***Define Materials*** forms to close all forms.
10. Click the **Define menu > Frame Sections** command to display the ***Frame Properties*** form. In that form:
    - Click the drop-down box that reads *Add I/Wide Flange* and then click on the *Add Rectangular* item.
    - Click the **Add New Property** button to display the ***Rectangular Section*** form. In that form:
      - Type **CONBEAM** in the *Section Name* edit box.
      - Select CONC from the *Material* drop-down box.
      - Type **42** in the *Depth (t3)* edit box.
      - Type **24** in the *Width (t2)* edit box.
      - Click the **OK** buttons on the ***Rectangular Section*** and ***Frame Properties*** forms to close all forms.
11. Select the line (frame) object by clicking on it.
12. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the ***Frame Properties*** form. In that form:
    - Click on CONBEAM in the *Properties* list box to highlight it.
    - Click the **OK** button.
13. Click the drop-down box in the status bar to change the units to 

.
14. Select the line (frame) object by clicking on it.
15. Click the **Edit menu > Divide Frames** command to display the ***Divide Selected Frames*** form.

16. Fill in the form as shown in the adjacent figure and click the **OK** button.



17. Click the **Define menu > Load Cases** command to display the ***Define Loads*** form. In that form:
    - Type **LIVE** in the *Load Name* edit box.
    - Select *LIVE* from the *Type* drop-down box.
    - Click the **Add New Load** button.
    - Click the **OK** button.
18. Click the **Define menu > Combinations** command to display the ***Define Response Combinations*** form. In that form:
    - Click the **Add New Combo** button to display the ***Response Combination Data*** form. In that form:
        - Accept the default *Response Combination Name*, *COMB1*.
        - Accept the default *Combination Type*, *Linear Add*.
        - Verify that *DEAD* is selected in the *Case Name* drop-down box.
        - Verify that *1* is entered in the *Scale Factor* edit box.
        - Click the **Add** button.
        - Select *LIVE* from the *Case Name* drop-down box.
        - Click the **Add** button.
        - Click the **OK** buttons on the ***Response Combination Data*** and ***Define Response Combinations*** forms to exit all forms.
19. Select all of the line (frame) objects by "windowing."
20. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the ***Frame Distributed Loads*** form. In that form:
    - Verify that the *Load Case Name* is *DEAD*.
    - In the *Load Type and Direction* area, verify that the *Forces* option is selected and that the *Gravity* direction is selected.
    - In the *Uniform Load* area, type **10.6**.
    - Click the **OK** button.
21. Click the **Get Previous Selection** button  (or the **Select menu > Get Previous Selection** command).
22. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the ***Frame Distributed Loads*** form. In that form:
    - Select *LIVE* from the *Load Case Name* drop-down box.
    - In the *Uniform Load* area type **5.4**.
    - Click the **OK** button.

23. Click the **Show Undeformed Shape** button  to remove the displayed frame uniform load assignments.

24. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the ***Display Options*** form. In that form:
    - Check the *Labels* box in the *Joints* area.
    - Click the **OK** button.

25. Select joint 16 (6 feet from the right end) by clicking on it.

26. Click the **Assign menu > Joint Loads > Forces** command to display the ***Joint Forces*** form. In that form:
    - Verify that the *Load Case Name* shown is *DEAD*.
    - Type **-250** in the *Force Global Z* edit box in the *Loads* area.
    - Click the **OK** button.

27. Click the **Get Previous Selection** button  (or the **Select menu > Get Previous Selection** command).

28. Click the **Assign menu > Joint Loads > Forces** command to display the ***Joint Forces*** form. In that form:
    - Select *LIVE* from the *Load Case Name* drop-down box.
    - Type **-150** in the *Force Global Z* edit box in the *Loads* area.
    - Click the **OK** button.

29. Click the **Show Undeformed Shape** button  to remove the displayed joint load assignments.

30. Click the **Set Display Options** button  on the main toolbar (or the **View menu > Set Display Options** command) to display the ***Display Options for Active Window*** form. In that form:
    - Uncheck the *Labels* box in the *Joints* area.
    - Click the **OK** button.

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

32. Select all of the objects by “windowing.”

33. Click the **Assign menu > Joint > Springs** command to display the ***Joint Springs*** form. In that form:
    - Type **50** in the *Translation 3* edit box.
    - Click the **OK** button.

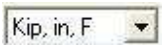
34. Click the **Analyze menu > Set Analysis Options** command to display the ***Analysis Options*** form. In that form:
    - Uncheck the *UX*, *UY*, *RX* and *RZ* check boxes leaving just the *UZ* and *RY* boxes checked.

- Click the **OK** button.

35. 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.
    - Verify that the *LIVE* analysis case is set to *Run* in the *Action* list.
    - Click the **Run Now** button to run the analysis.
36. 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.
37. Click the **Display menu > Show Forces/Stresses > Frames/Cables/Tendons** command to display the ***Member Force Diagram for Frames*** form. In that form:
    - Select *COMB1* from the *Case/Combo Name* drop-down box.
    - Select the *Moment 3-3* option in the *Component* area.
    - Uncheck the *Fill Diagram* check box.
    - Check the *Show Values on Diagram* check box.
    - Click the **OK** button to display the moment diagram.

Note: To change the font size, click the **Options menu > Preferences > Dimensions/ Tolerances** command to display the ***Dimensions/Tolerances Preferences*** form. Type in a new font size in the *Minimum Graphic Font Size* edit box (usually about 6 points is sufficient) and click the **OK** button.

Note: Right click on any of the frame objects to view details of the moment diagram for that member.

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

.
39. Click the **Show Deformed Shape** button  (or the **Display menu > Show Deformed Shape** command) to display the ***Deformed Shape*** form. In that form:
    - Select *COMB1* from the *Case/Combo Name* drop-down box.
    - Click the **OK** button.
40. Right click on the joint at the far right end of the beam to view its deflection.