

Problem I

Prestressed Concrete Beam

Concrete

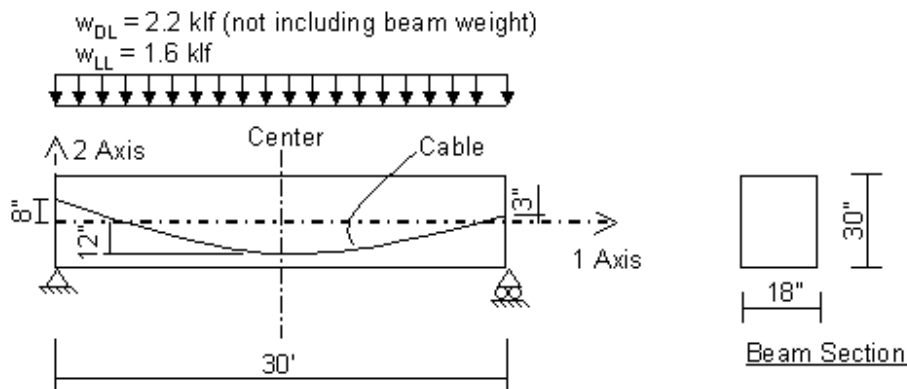
$E = 4400$ ksi, Poissons Ratio = 0.2

$f_c = 6$ ksi

Cable Tension = 200 kips

To Do

Determine the moment diagram for a DL + LL + PRESTRESS loading combination. Compare the results using 4 output segments and using 30 output segments.



CSI Solution Demonstrates Use of these Features

- Response Combinations
- Output Stations
- Prestressing

Problem I Solution

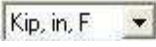
1. Click the **File** menu > **New Model** command to display the **New Model** form.

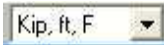
2. Click the drop-down box to set the units to .




3. Click the **Beam** button to display the **Beam** form. In that form:


- Type **1** in the *Number of Spans* edit box.
- Type **30** in the *Span Length* 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 **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 **4400** in the *Modulus of Elasticity* edit box.
 - Verify .2 is entered in the *Poisson's Ratio* edit box.
 - Type **6** in the *Specified Conc Comp Strength, f'c* edit box.
 - Verify 60 is entered in the *Bending Reinf Yield Stress, fy* edit box.
 - Type **60** in the *Shear Reinf. Yield Stress, fys* 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 **30** in the *Depth (t3)* edit box.
 - Type **18** in the *Width (t2)* edit box.
 - Click the **OK** buttons on the **Rectangular Section** and **Frame Properties** forms to close all forms.
 11. 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.
 - Type **PRESTRES** in the *Load Name* edit box.

- Select *OTHER* from the *Type* drop-down box.
 - Click the **Add New Load** button.
 - Click the **OK** button.
12. Select the line (frame) object by clicking on it.
13. 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.
14. Click the **Draw menu > Frame/Cable/Tendons** command to display the **Properties of Object** form. In that form,
- Click the *Line Object Type* drop-down list and select the *Tendon* option.
 - Click on the joint at one end of the beam and then click on the joint at the other end of the beam to draw the tendon. The **Tendon Data for Line Object 3** form will display when you release the mouse button.
15. With the **Tendon Data for Line Object 3** form displayed, click the **Parabolic Calculator** button to display the **Define Parabolic Tendon Layout for Line Object 3** form. In that form,
- Select 1 from the *Quick Start* drop-down list in the lower left corner of the form.
 - Click the **Quick Start** button to update the *Tendon Layout Data* spreadsheet. An additional row of data will be added to the spreadsheet. In the spreadsheet area,
 - Type **8** in *Coord 2* cell of the first row of data.
 - Ensure that **-12** is in the *Coord 2* cell in the second row of data.
 - Type **3** in the *Coord 2* cell in the third row of data.
 - Click the **Refresh** button to update the spreadsheet area and the graphical display area.
 - Ensure that the *Use Calculated Results for This Tendon* check box is checked.
 - Click the **Done** button to close the **Define Parabolic Tendon Layout for Line Object 3** form and redisplay the **Tendon Data for Line Object 3** form.
16. With the **Tendon Data for Line Object 3** form redisplayed, click the **Add** button in the *Tendon Loads* area of the form to display the **Tendon Load** form. In that form,
- Select the PRESTRES load case from the *Load Case Name* drop-down list.
 - Type **200** in the *Force* edit box.
 - Ensure that all of the *Friction and Anchorage Losses* and *Other Loss Parameters* are zero. If necessary, type zero in the *Curvature Coefficient*, *Wobble Coefficient*, *Anchorage Set Slip*, *Elastic Shortening Stress*, *Creep Stress*, *Shrinkage Stress* and *Steel Relaxation Stress* edit boxes.
 - Click the **OK** buttons on the **Tendon Load** form and the **Tendon Data for Line Object 3** form to close those forms.
17. Click the drop-down box in the status bar to change the units to .


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 the *DEAD* Case 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* Case from the *Case Name* drop-down box.
 - Click the **Add** button.
 - Select *PRESTRES* Case 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 close all forms.
19. Select the line (frame) object.
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 **2.2**.
 - Click the **OK** button.
21. Select the line (frame) object.
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 **1.6**.
 - Click the **OK** button.
23. Select the line (frame) object.
24. Click the **Assign menu > Frame/Cable/Tendon > Output Stations** command to display the **Assign Output Station Spacing** form. In that form:
 - Type **4** in the *Min Number Stations* edit box.
 - Click the **OK** button.
25. Click the **Show Undeformed Shape** button  to remove the displayed frame output station assignments.
26. 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.
27. 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.
 - Verify that the *PRESTRES* analysis case is set to *Run* in the *Action* list.
 - Click the **Run Now** button to run the analysis.
28. 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.
29. 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: You may want to print this moment diagram for comparison with the one obtained when 20 output stations are specified. To print the moment diagram, click the **File menu > Print Graphics** command.

Note: For response combinations, when force diagrams are plotted, exact values are only calculated at the ends of each output station. Those exact values are plotted and then they are connected with straight lines.

30. Click the **Lock/Unlock Model** button  to unlock the model. Click the **OK** button when asked if it is ok to delete.
31. Select the line (frame) object.
32. Click the **Assign menu > Frame/Cable/Tendon > Output Stations** command to display the **Assign Output Station Spacing** form. In that form:
- Type **20** in the *Min Number Stations* edit box.
 - Click the **OK** button.

33. Click the **Show Undeformed Shape** button  to remove the displayed frame output station assignments.
34. 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.
35. 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
36. Click the **Display menu > Show Forces/Stresses > Frames/Cables/Tendons** command to display the **Member Force Diagram for Frames** form. In that form:
 - Verify that the *COMB1* is selected in the *Case/Combo Name* drop-down box.
 - Verify that the *Moment 3-3* option is selected in the *Component* area.
 - Verify that the *Show Values on Diagram* check box is checked.
 - Click the **OK** button to display the moment diagram.