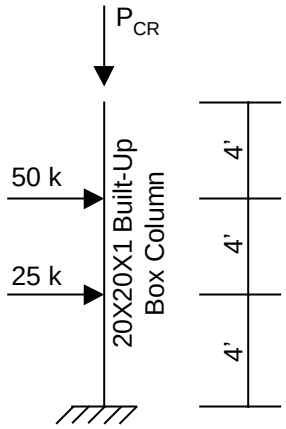
Problem P

Critical Buckling Load

Steel
E =29000 ksi, Poissons Ratio = 0.3

To Do
Use P-Delta option and iteration to determine the critical buckling load for this built-up column. Hint: $P_{CRITICAL}$ is between 15,480 and 15,490 kips.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem P Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model…**. This displays the Coordinate System Definition dialog box.

3. In this dialog box:

   - Select the Cartesian Tab.
   - Type **0** in the X Direction Number of Grid Spaces edit box.
   - Type **0** in the Y Direction Number of Grid Spaces edit box.
   - Type **1** in the Z Direction Number of Grid Spaces edit box.
   - Type **12** in the Z Direction Grid Spacing edit box.
   - Click the **OK** button.

4. Click the **Quick Draw Frame Element** button  on the side toolbar.

5. Click once on the grid line in the 3-D View window to draw the frame element.

6. Click the **Pointer** button  on the side tool bar to exit draw mode and enter select mode.

7. Click on the bottom joint in the 3-D View window to select it.

8. From the **Assign** menu, choose **Joint**, and then **Restraints…** from the submenu. This will display the Joint Restraints dialog box.

9. In this dialog box:

   - Click the **fixed base fast restraint** button  to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.
   - Click the **OK** button.

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

11. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

12. In this dialog box:

- Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.
- Click the **OK** button twice to exit the dialog boxes.

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
14. In the Click To area, click the drop-down box that saysAdd I/Wide Flange and then click on the Add Box/Tube item to display the Box/Tube Section dialog box.
15. In this dialog box:
    - Type **BOX** in the Section Name edit box.
    - Type **20** in the Outside Depth (t3) edit box.
    - Type **20** in the Outside Width (t2) edit box.
    - Type **1** in the Flange Thickness (tf) edit box.
    - Type **1** in the Web Thickness (tw) edit box.
    - Click the **OK** button twice to exit all dialog boxes.
16. Click the drop down box in the status bar to change the units to kip-ft. 

17. Click on the frame element to select it.
18. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
19. In this dialog box:
    - Click on BOX in the Frame Sections area to highlight it.
    - Click the **OK** button.
20. Click the **Show Undeformed Shape** button  to remove the displayed frame element assignments.
21. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.
22. In this dialog box:
    - Type **LAT** in the Load edit box.
    - Select OTHER from the Type drop-down box.

- Type **0** in the Self Weight Multiplier edit box.
- Click the **Change Load** button.
- Type **AXIAL** in the Load edit box.
- Click the **Add New Load** button.
- Click the **OK** button.

23. Click on the frame element to select it.

24. From the **Assign** menu select **Frame Static Loads…** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

25. In this dialog box:

    - Select LAT from the Load Case Name drop-down box.
    - Select Global X from the Direction drop-down box in the Load Type and Direction area.
    - Type **.3333** in the first Distance edit box and type **25** in the first Load edit box.
    - Type **.6667** in the second Distance edit box and type **50** in the second Load edit box.
    - Click the **OK** button.

26. Select the top joint.

27. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

28. In this dialog box:

    - Select AXIAL from the Load Case Name drop-down box.
    - Type **-1** 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 force assignments.

30. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

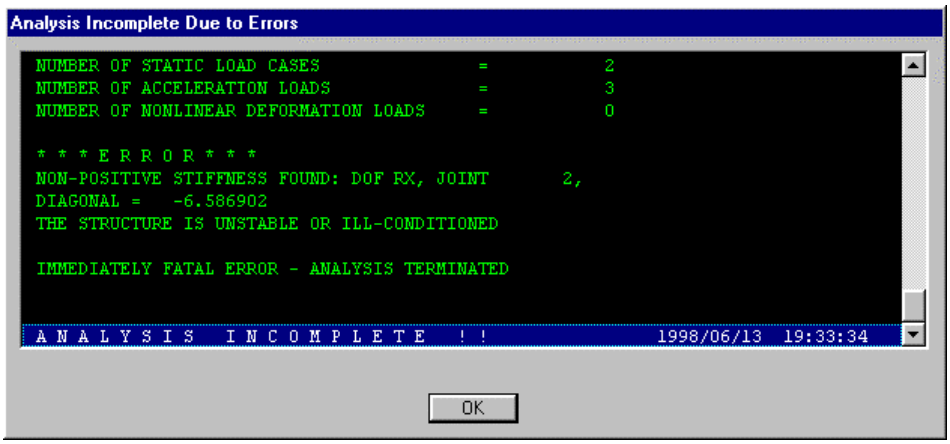
    - Check the Include P-Delta check box, if it is not already checked.

- Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
- In this dialog box:
  - Type **5** in the Maximum Iterations edit box.
  - Select AXIAL from the Load Case drop-down box.
  - Type **15485** in the Scale Factor area.

    Note: Since the hint says $P_{CRITICAL}$ is between 15,480 and 15,490 kips, we will start midway between these two values at 15485 kips.
  - Click the **Add** button.
  - Click the **OK** button twice to exit all dialog boxes.

31. Click the **Run Analysis** button to run the analysis.

32. When the analysis is complete check the messages in the Analysis window. There should be an error message similar to that shown below indicating the structure is unstable. This indicates that 15,485 kips is larger than the critical buckling load. Click the **OK** button to close the Analysis window.



33. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
- Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
- In this dialog box:
  - Highlight the AXIAL/15485 Load Case/Scale Factor..
  - Type **15484** in the Scale Factor area.
  - Click the **Modify** button.

- Click the **OK** button twice to exit all dialog boxes.

34. Click the **Run Analysis** button to run the analysis.

35. When the analysis is complete check the messages in the Analysis window. Again there should be an error message indicating the structure is unstable. This indicates that 15,484 kips is larger than the critcal buckling load. Click the **OK** button to close the Analysis window.

36. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
    - In this dialog box:
        - Highlight the AXIAL/15484 Load Case/Scale Factor..
        - Type **15483** in the Scale Factor area.
        - Click the **Modify** button.
        - Click the **OK** button twice to exit all dialog boxes.

37. Click the **Run Analysis** button to run the analysis.

38. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). The analysis run successfully. Thus the critical buckling load is approximately 15484 kips. Click the **OK** button to close the Analysis window.