

# Problem P

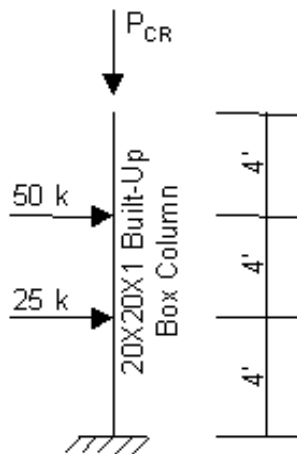
## Critical Buckling Load

### Steel

$E = 29000$  ksi, Poissons Ratio = 0.3

### To Do

Use the P-Delta option and iteration to determine the critical buckling load for this built-up column.



## CSI Solution Demonstrates Use of These Features

- Buckling Analysis
- P-Delta

## Problem P Solution


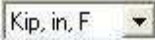
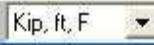


1. Click the **File menu > New Model** command to display the **New Model** form.



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




3. Click the **Grid Only** button to display the **New Coord/Grid System** form. In that form:

- Select the Cartesian Tab.

- Type **1** in the *X Direction* edit box in the *Number of Grid Lines* area.
  - Type **1** in the *Y Direction* edit box in the *Number of Grid Lines* area.
  - Type **2** in the *Z Direction* edit box in the *Number of Grid Lines* area.
  - Type **12** in the *Z Direction* edit box in the *Grid Spacing* area.
  - Click the **OK** button.
4. Click the “X” in the top right-hand corner of the 3-D View window to close that view.
  5. Click the **Set XZ View** button  to switch to a XZ view.
  6. Click the drop-down box in the status bar to change the units to .
  7. 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.
  8. Click the **Define menu > Frame Sections** command to display the **Frame Properties** form.
  9. In the *Choose Property Type to Add* area, click the drop-down list that reads *Add I/Wide Flange* and then click on the *Add Box/Tube* item.
  10. Click the **Add New Property** button to display the **Box/Tube Section** form. In that form:
    - 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** buttons on the **Box/Tube Section** and **Frame Properties** forms to exit all forms.
  11. Click the drop-down box in the status bar to change the units to .
  12. Click the **Quick Draw Frame/Cable** button  on the side toolbar to display the **Properties of Object** form. Verify that the *Line Object Type* is *Straight Frame*. Then, highlight (select) *BOX* from the *Property* drop-down list.
  13. Click once on the grid line located along the Z axis to draw a frame object.
  14. Click the **Set Select Mode** button  on the side toolbar to exit Draw mode and enter Select mode.
  15. Click on the point located at the origin (0, 0, 0) to select it.
  16. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:

- 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.
17. Click the **Define menu > Load Cases** command to display the **Define Loads** form. In that form:
- Type **LAT** in the *Load Name* edit box.
  - Select *OTHER* from the *Type* drop-down list.
  - Click the **Modify Load** button.
  - Type **AXIAL** in the *Load Name* edit box.
  - Click the **Add New Load** button.
  - Click the **OK** button.
18. Click on the frame object to select it.
19. Click the **Assign menu > Frame/Cable/Tendon Loads > Point** command to display the **Frame Point Loads** form. In that form:
- Select *LAT* from the *Load Case Name* drop-down list.
  - Select *X* from the *Direction* drop-down list in the *Load Type and Direction* area.
  - Verify that the *Relative Distance from End-I* option is selected in the *Point Loads* 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.
20. Select the point at the top of the column (0, 0, 12).
21. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
- Select *AXIAL* from the *Load Case Name* drop-down list.
  - Type **-1** in the *Force Global Z* edit box in the *Loads* area.
  - Click the **OK** button.
22. Click the **Show Undeformed Shape** button  to remove the displayed joint force assignments.
23. 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 **Delete Case** button.
  - Highlight (select) *DEAD* in the *Case Name* list and click the **Modify/Show Case** button to display the **Analysis Case Data** form. In that form:
    - Type **LAT** in the *Analysis Case Name* edit box.
    - In the *Analysis Type* area, select the *Nonlinear* option.

- In the *Other Parameters* area, click on the **Modify/Show** button to display the **Nonlinear Parameters** form. In that form:
    - Select the *P-Delta* option in the *Geometric Nonlinearity Parameters* area.
    - Click the **OK** buttons on the **Nonlinear Parameters** and **Analysis Case Data** forms to return to the **Analysis Cases** form.
24. In the **Analysis Cases** form:
- Highlight (select) *AXIAL* in the *Case Name* list and click the **Modify/Show Case** button to display the **Analysis Case Data** form. In that form:
    - Select *Buckling* from the *Analysis Case Type* drop-down box.
    - In the *Stiffness to Use* area, select the *Stiffness at End of Nonlinear Case* option, and verify that *LAT* is shown in the drop-down list.
    - Type **2** in the *Number of Buckling Modes* edit box.
    - Click the **OK** buttons on the **Analysis Case Data** and **Analysis Cases** forms to exit all forms.
25. Click on the frame object to select it.
26. Click the **Assign menu > Frame/Cable/Tendon > Automatic Frame Subdivide** command to display the **Assign Automatic Frame Subdivide** form. In that form:
- Select the *Auto Subdivide Frame* option.
  - Check the *Minimum Number of Segments* box and type **4** in the edit box.
  - Click the **OK** button.
27. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:
- Verify that all analysis cases are 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 and then click the **OK** button to close the window.
29. Click the **Display menu > Show Tables** command to display the **Choose Tables for Display** form. In that form:
- Click the + (plus) symbol associated with *Structure Output* item.
  - Click the + (plus) symbol associated with *Other Output* Items.
  - Click the *Table: Buckling Factors* item to select it.
  - Click the **OK** button to display the **Buckling Factors** table. The product of the Scale Factors shown times the applied axial load gives the critical buckling load. Click the **Done** button to close the form.