8.12 P – Delta Analysis

- Consider a column of length L that has two concentrated loads applied at the top of the column: a vertical load P and a horizontal load H.

- According to a linear elastic analysis, the reactions at the base of the column for these two loads will be a vertical reaction of magnitude P, a horizontal reaction of magnitude H and a moment equal to H*L as shown in the figure below.

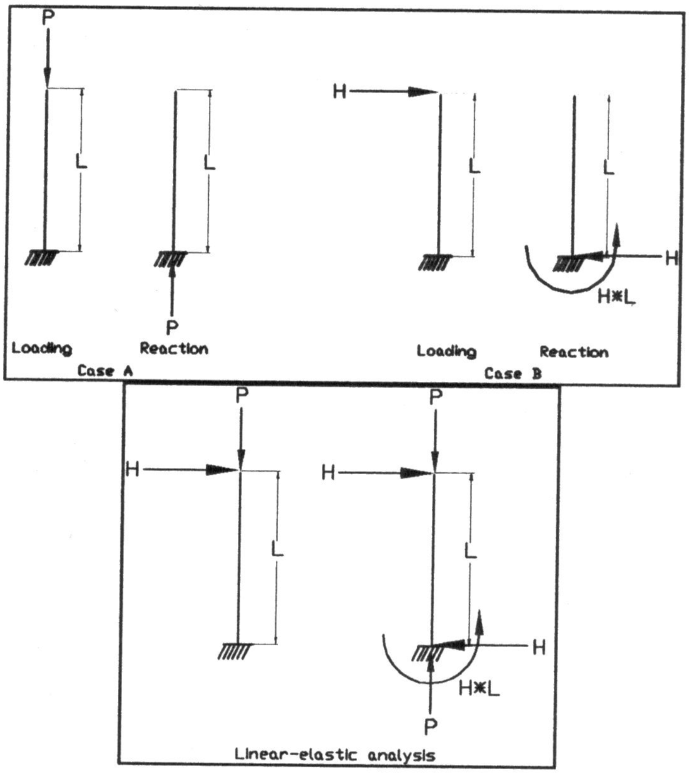

Figure 8. 10

- This is to say that the result of Loads A and B acting simultaneously is equivalent to the result of Load A plus the result of Load B.
- This logic represents a linear combination, which can be created in STAAD.*Pro* using the *Define Combinations* tab.
- This method of load combination could be more accurately termed "result combination", because it does not truly analyze a combined load case. It simply instructs the program to combine the results of multiple load cases.
- The implicit assumption with this type of load combination is that the effect of the combined loading is equivalent to the sum of the effects of the individual loads.
- This may or may not be a valid assumption, and it warrants consideration on the part of the design professional.
- The linear-elastic type of analysis is not permitted with some design codes, including the ACI code. There is an extra effect called the P – Delta effect which must be taken into account when designing according to the ACI code.
- In a real structure, the horizontal force H might be caused by a wind load or earthquake load, causing the column to deflect a distance Δ.

The taller the column, the greater the distance Δ for a given force H.

- The vertical force P might represent a dead load or a live load. So, in reality, these load cases would act simultaneously, not independent of each other.
- During this simultaneous action of the two loads, while the column is deflecting due to the action of the horizontal load, the position of the vertical load P shifts a distance Δ so that the vertical load, instead of acting axially along the column, now induces a moment reaction at the base of the column equal to P * Δ.

- The total moment reaction at the base of the column is now (H * L) + (P * Δ) as shown in the figure below. However, the additional component of moment, P * Δ, is not apparent in a linear –elastic analysis.

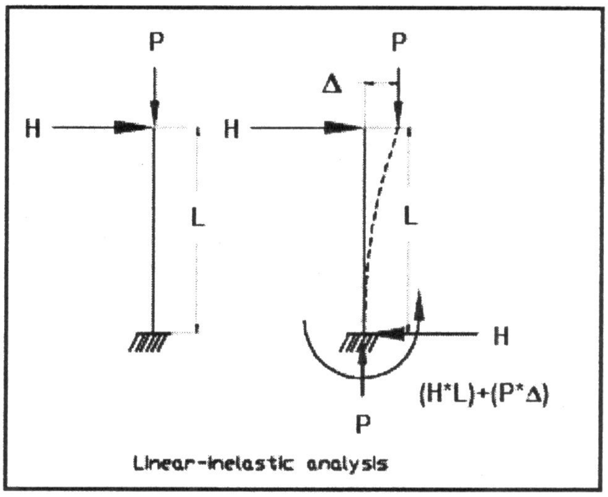

Figure 8. 11

- When considering the equations of static equilibrium, the quantity (P * Δ) is not actually seen in the "applied load" side of the equation, but appears in the reaction side of the equation.

- This is a *linear* – *inelastic* analysis. In this type of analysis, it is not correct to simply take the combination of the results of Load A plus the results of Load B.

- The results of Load A just give a reaction P.

- The results of Load B just gives a reaction H * L.

- Looking at these two load cases in isolation, the P – Delta effect never becomes apparent.

- It is only when these two load cases act simultaneously that the P – Delta effect is produced. Consequently, the traditional linear-elastic load combination, where results are just added up, is not going to reveal the P – Delta value.

- The ACI code indicates that in the design of a column, the slenderness effect can be accounted for using two different methods.
- One method is called the moment magnifier approach, which uses some code-based equations to approximate these second order effects.
- The other method is to perform a P – Delta Analysis.
- The next step in the example model will be to create a third *load case* that is a combination of the first two *load cases*.
- In this example, an alternate method of combining loads will be used, one that correctly accounts for the P-Delta effect by applying the horizontal and vertical loads simultaneously.
- There are actually a couple of ways to achieve this in STAAD.*Pro*.
- One way would be to put both loads in a single load case, instead of creating separate load cases for the horizontal and vertical loads, as was done in this model.
- Although it is possible, this is not a very convenient method, because of all the different *load cases* that would be required to correctly model all of the required load combinations.
- This method would also be undesirable from the standpoint that it is often necessary to evaluate a structure for individual load cases as part of the overall structural evaluation/design. Combining multiple forces into each load case would make this evaluation impossible.
- Instead of requiring all the loads on the structure to be jumbled into a single load case in order to carry out a P – Delta Analysis, STAAD.*Pro* provides another type of primary load that "looks like" a load combination.

- It is called a *Repeat Load*, and it is a primary load where the program is instructed to create a new load case whose constituents are derived from the various existing *load cases* with any necessary load factors applied to them.

- Using the *Repeat Load* command is a two-step process. First, a new *Repeat load case* must be created, and then the constituent *load cases* and their respective factors must be identified and associated with the new *Repeat Load* case.

- Click on **Load Cases Details** in the *Load & Definition* dialog, and then click the **Add** button.

- A *Repeat Load* is actually a *primary load*, and the *Primary* tab is active by default in the *Add New:Load Cases* dialog.

- Type the name **Loads 1 + 2** in the *Title* field.

- Leave the *Loading Type* set to *None* by default, since this new *load case* will not be associated with any code-based load types.

- Click the **Add** button, but do not close this dialog yet.

- Click on the expression **3: Loads 1 + 2** in the *Load & Definition* dialog.

This is the way to tell STAAD.*Pro* that the next component is to be added to this load case.

Note that the *Add New:Load Cases* dialog automatically changes to the *Add New:Load Items* dialog.

- Click the **Repeat Load** tab in the *Add New:Load Items* dialog.

The *Repeat Load* tab contains two items: *Repeat Load* and *Reference Load*. The *Repeat Load* item is active by default.

The left side of this dialog lists the existing *Available Load Cases*.

The right side displays the *Repeated Load Definition*.

Loads can be moved back and forth between the *Available Load Cases* on the left and the *Repeated Load Definition* on the right using the arrow buttons.

The *Factor* field is available to apply factors to individual load cases that comprise the *Repeated Load Definition.*

- Click on **1: Pressure Load** in the *Available Load Cases* list.
- Click the **single right arrow** [>] button to move the load to the *Repeated Load Definition* list.
- Since the design will be based on the ACI code, the loads should be factored.
- Apply a dead load factor of **1.2** in the *Factor* field.
- Click on **2: Lateral Load** in the *Available Load Cases* list.
- Click the **single right arrow** [>] button.
- Enter a *Factor* of **1.6**, and click the **Add** button.
- Click the **Close** button.

The new *Repeat Load* case is shown in the *Load & Defintions* dialog. The syntax is *load case 1 with a factor of 1.2 load case 2 with a factor of 1.6.*

- Remember to always use the *Repeat Load* specification, rather than the *Load Combination* specification, any time a P – Delta analysis is to be performed.
- A copy of this model is already saved in this state in the dataset, and is named Dataset 8_7.std.
- Click **File | Close** to return to the Start Page.
- Click **No** when asked if you want to save.

8.13 Providing Analysis Instructions

- Open the file named **Dataset 8_7.std**.

The next step is to issue the analysis instructions.

- Click on the **Analysis/Print** tab in the Page Control.
- Click on the **PDelta Analysis** tab in the *Analysis/Print Commands* dialog.
- The *PDelta Analysis* page includes a field labeled *Number of Iterations*, and a field labeled *Converge*.
- If a *Number of Iterations, n* is specified, STAAD.*Pro* will iterate *n* times.
- An alternative to specifying a *Number of Iterations* is to use the *Converge* option. See the following commentary for additional information about the *Converge* option, but take special note of the "word of caution" below.

When the *Converge* checkbox is selected, STAAD.*Pro* will continue to iterate and compare joint displacements with a convergence displacement tolerance.

The default convergence displacement tolerance is equal to the maximum span of the structure divided by 120. Note that this default value was not intended to suggest an "optimum" value. It was merely put in place to allow the engineer to apply his or her own value based on engineering judgment.

To specify a different value for the convergence displacement tolerance, use the SET DISPLACEMENT *f* command in the input file. Refer to section 5.5 of the Technical Reference manual.

The *Converge* command has the option of specifying a maximum number of iterations, "*m*". If "*m*" is specified, the analysis will stop after that iteration even if convergence has not been achieved. If convergence is achieved in less than "*m*" iterations, the analysis is terminated.

- A word of caution about the use of the CONVERGE option: it is possible that a model using the CONVERGE option may have 2 early iterations with results close enough to be deemed converged. However, if the same analysis was changed to not use CONVERGE but instead to specify many more iterations, occasionally buckling would be detected. Experience shows that it generally takes 5 to 35 iterations to reach buckling failure. So in this day and age where computing power and speed is so abundant, good practice dictates avoiding the use of the CONVERGE feature and instead using the option to set the *Number of Iterations* high enough to prove that the structure is stable for a given load case.
- Enter **35** in the *Number of Iterations* field.
- Leave all other options at their default settings, and click the **Add** button to add the P – Delta Analysis command to the input instructions.
- Click the **Close** button to dismiss the *Analysis/Print Commands* dialog.
- A copy of this model is already saved in this state in the dataset, and is named Dataset 8_8.std.
- Click **File | Close** to return to the Start Page.
- Click **No** when asked if you want to save.

8.14 Running the Analysis

- Open the file named **Dataset 8_8.std**.
- The model is now ready to analyze.
- Click **Analyze | Run Analysis...**.
- The program should be able to run the analysis and generate results. The message *Analysis Successfully Completed* should appear in the lower portion of the *STAAD Analysis and Design* dialog, followed by some messages indicating that the program created some results files.
- Click the **Go to Post Processing Mode** radio button, and then click **Done**.
- Click **OK** to accept the three load cases shown in the *Selected* list on the *Results Setup* dialog.
- Keep the current model open for use in the next section.