CONCRETE DESIGN SOFTWARE LIBRARY

DESIGN AND INVESTIGATION OF REINFORCED CONCRETE COLUMN SECTIONS

5420 Old Orchard Road, Skokie, Illinois 60077-1083
Telephone: (847) 966-6200
Fax: (847) 966-9781
Website: www.portcement.org

PCACOL

This computer program (including software deign, programming structure, graphics, manual, and on-line help system) was created by the Computer Programs Unit, Portland Cement Association.

While the Portland Cement Association has taken every precaution to utilize the existing state-of-the-art and to assure the correctness of the analytical solution techniques used in this program, the responsibilities for modeling the structure, inputting data, applying engineering judgment to evaluate the output, and implementing engineering drawings remain with the structural engineer of record. Accordingly, Portland Cement Association does and must disclaim any and all responsibility for defects or failures of structures in connection with which this program is used.

© 1998, 1989, 1990, 1992, 1998 Portland Cement Association. All Rights Reserved.

PCACOL is a Trademark of the Portland Cement Association

Microsoft and Windows are registered trademarks of the Microsoft Corporation.

Appendix

Conversion Factors—English to Metric

To convert from	To	Multiply by
in	mm	25.4
ft	m	0.3048
kip/in^2 (ksi)	MPa (N/mm^2)	6.895
kip	kN	4.448
kip-ft	kN-m	1.356
lb/ft	kg/m	1.488164

Conversion Factors—Metric to English

To convert from	To	Divide by
mm	in	25.4
m	ft	0.3048
MPa (N/mm^2)	kip/in^2 (ksi)	6.895
kN	kip	4.448
kN-m	kip-ft	1.356
kg/m	lb/ft	1.488164

Table of Contents

Chapter 1—Introduction

Terrms 1-1
Conventions 1-2
System Requirements 1-3
Program Capacity 1-3
Installing PCACOL 1-4
Running PCACOL 1-4

Chapter 2—Method of Solution

A - Definitions and Assumptions 2-2
B - Sign Convention 2-6
C - Section Investigation 2-8
D - Section Design 2-11
E - Moment Magnification (Slenderness Effects) for Braced Columns ... 2-12
F - Moment Magnification (Slenderness Effects) for Sway Columns 2-15
G - Output Notation 2-18

Chapter 3—The PCACOL Window

Using the Menus 3-3
Menu Conventions 3-3
Using a Dialog Box 3-4
Working with a List Box 3-7
File Menu 3-8
Input Menu 3-9
Solve Menu 3-11
View Menu 3-12
Help Menu 3-13

Chapter 4—Operating the Program

Creating a New Data File 4-1
Opening an Existing Data File 4-2
Saving the Data 4-3
Importing Data 4-4
Printing the Results 4-5
General Information 4-6
Material Properties 4-7
Section|Rectangular 4-8
Section|Circular 4-9
Reinforcement |All Sides Equal 4-10
Reinforcement|Equal Spacing 4-12
Reinforcement|Sides Different 4-14
Reinforcement|Irregular 4-16
Reinforcement|Design Criteria 4-17
Reinforcement|Confinement 4-18
Slenderness|Factors 4-19
Slenderness|Design Column 4-20
Slenderness|Columns Above and Below 4-22
Slenderness|X-Beams 4-23
Slenderness|Y-Beams 4-24
Loads|Factored 4-25
Loads|Service 4-26
Loads|Control Points 4-27
Loads|Axial Loads 4-28
Loads|Load Combinations 4-29
Flipping/Rotating the Section 4-30

Chapter 4—Operating the Program (continued)

Executing a Run 4-31
Viewing the Results 4-32
Changing the Startup Defaults 4-33
Changing the Rebar Database 4-34
Superimposing Diagrams 4-36
Viewing the Diagram 4-37

Chapter 5— Examples

Example 1 - Design of a Short Column 5-2
Example 2 - Capacity of a Braced Column 5-11
Example 3 - Capacity of a Sway Column 5-22

Chapter 6—The IRRCOL Modulle

Main Menu 6-2
Edit Menu 6-3
View Menu 6-4
Help Menu 6-5
Defining the Drawing Area 6-7
Graphical Editing of the Section or Opening 6-8
Graphical Editing of the Reinforcement 6-9
Manual Editing of the Section or Opening 6-12
Manual Editing of the Reinforcement 6-13
Zooming and Panning the Image 6-14
Example 4 - Capacity of an Irregular Section 6-15

APPENDIX

Conversion Factors—English to Metric A-1
Conversion Factors—Metric to English A-2

Chapter 1

Introduction

Terms

The following terms are used throughout this manual. A brief explanation is given to help familiarize yourself with them.

Windows	refers to the Microsoft Windows environment version 3.1 or higher.
[ ]	indicates metric equivalent
Click on	means to position the cursor on top of a designated item or location and press and release the left-mouse button (unless instructed to use the right-mouse button).
Double-click on	means to position the cursor on top of a designated item or location and press and release the left-mouse button twice in quick succession.

Conventions

To help you locate and interpret information easily, the PCACOL manual adheres to the following text format.

Style	Meaning
Italic	indicates a glossary item, or emphasizes a given word or phrase.
Bold	indicates the name of a menu or a menu item command such as **File** or **Save**.
`Mono-space`	indicates something you should enter with the keyboard. For example `"c:\*.txt".`
KEY + KEY	indicates a key combination. The plus sign indicates that you should press and hold the first key while pressing the second key, then release both keys. For example, "ALT + F" indicates that you should press the "ALT" key and hold it while you press the "F" key. then release both keys.

System Requirements

To use PCACOL, a personal computer running Microsoft Windows V3.1 is required. PCACOL needs less than 1.0 MB of disk space to install.

Program Capacity

- 500 reinforcing bars within a section.
- 50 exterior points that define the geometric outline of the cross section (IRRCOL module only).
- 50 interior points that define an opening in the cross section (IRRCOL module only).
- 160 factored load entries, each consisting of an axial load, a moment about the x-axis, and a moment about the y-axis.
- 10 service load entries, each consisting of dead, live, and lateral axial loads, moments at column top about the x and y axes, and moments at column bottom about the x and y axes.

Installing PCACOL

1. Execute Windows and insert the PCACOL diskette into the proper floppy disk drive.
2. From the Program Manager **File** menu, selct **Run**.
3. Enter `a:pcasetup` or `b:pcasetup` (depending on the drive the diskette is in) in the Command Line text box, then choose ok.
4. Follow the instructions displayed on the screen.
5. After the setup program displays a successful installation message, the lst dialog box displayed asks whether you would like to view the README.TXT file. This file contains information supllemental to the manual.

Running PCACOL

Select the “PCACOL” icon from within the “PCA Programs” group window and press ENTER (or simply double-click on the icon).

Chapter 2

Method of Solution

PCACOL enables the user to compute the capacity of (investigate) or design reinforced concrete column sections. The section under consideration may be subject to a combination of loadings consisting of combined axial loads and uniaxial or biaxial moments.

In PCACOL, the section may be rectangular or circular with any reinforcement layout. Material properties for the concrete and reinforcing steel are user-controlled. The applied loading consists of tensile or compressive axial loads acting alone or combined with uniaxial or biaxial bending.

At the user's option, the program may investigate the slenderness effects on a column using the approximate method described in the design code. If applicable, the program computes the magnification factors to be applied to the moment(s).

With the IRRCOL add-on module, PCACOL can investigate irregularly-shaped sections. An irregular section is defined as a closed polygon and may even contain an opening (hollow section).

PCACOL conforms to the provisions of the following desing specifications:
Building Code Requirements for Structural Concrete (ACI 318-95), published by the American Concrete Institute, Farmington Hills, MI.

Design of Concrete Structures (A23.3-94), published by the Canadian Standards Association, Ontario, Canada

A - Definitions and Assumptions

1. The analysis of the reinforced concrete section conforms to the provisions of the Strength Design Theory.
   a) All conditions of strength satisfy the applicable conditions of equilibrium and strain compatibility.
   b) Strain in the concrete and in the reinforcement is directly proportional to the distance from the neutral axis.
   c) The maximum usable (ultimate) strain at the extreme concrete compression fiber is assumed equal to 0.003 (0.0035 for CSA) unless otherwise specified by the user.
   d) A uniform rectangular concrete stress block is used. The maximum uniform concrete compressive stress, f_c, is $0.85\, f'_c$ and the block depth is $\beta_1 c$, where c is the distance from the extreme compression fiber to the neutral axis and β_1 is described in item 4 below. Both f_c and β_1 may be modified by the user. For CSA, f_c is taken as:

      $f_c = (0.85 - 0.0025\, f'_c)\ f'_c \geq 0.67\ f'_c$, where f'_c is in MPa

   e) Concrete displaced by the reinforcement in compression is deducted from the compression block.

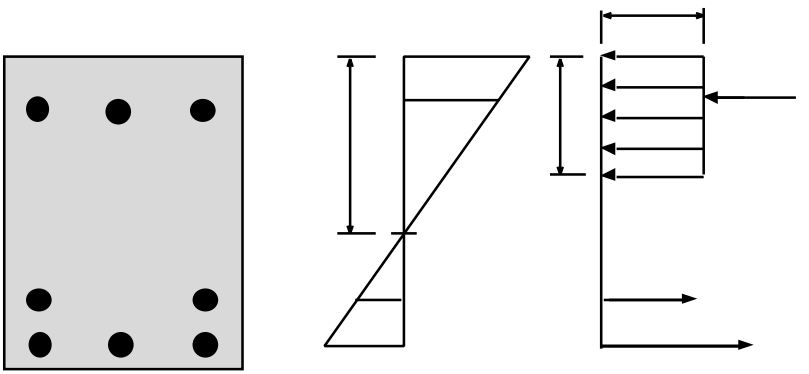


Figure 2-1 — Analysis of Reinforced Concrete Section

f) For the reinforcing steel, the elastic-plastic stress-strain distribution is used. Stress in the reinforcing steel below the yield strength, f_y, is directly proportional to the strain. For strains greater than that corresponding to the yield strength, the reinforcement stress remains constant and equal to f_y. If the rupture strain for steel is specified, for strains greater than the specified rupture strain, the reinforcement stress equals zero.

g) Tensile strength of concrete is neglected.

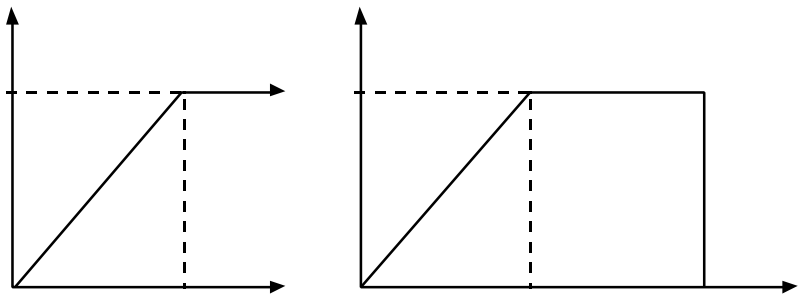


Figure 2-2 — Elastic-Plastic Stress-Strain Reinforcement Curve a) with $\varepsilon_{rupture}$ = infinity, b) $\varepsilon_{rupture}$ specified

2. The modulus of elasticity of concrete, E_c, is computed as follows (unless otherwise specified by the user):

   $E_c = 57{,}000\sqrt{f'_c}$ where f'_c and E_c are in psi.

   $E_c = 4{,}700\sqrt{f'_c}$ where f'_c and E_c are in MPa.

   For the CSA standard, $E_c = 4{,}500\sqrt{f'_c}$, where f'_c and E_c are in MPa.

3. The modulus of elasticity of reinforcing steel, E_s, is taken as 29,000 ksi unless otherwise specified by the user.
4. The ratio of the concrete compression block depth to the distance between the extreme compression fiber and the neutral axis, β_1, is computed as follows (unless otherwise specified by the user):

   $0.65 \le \beta_1 = 1.05 - 0.05 f'_c \le 0.85$, where f'_c is in ksi

   $0.65 \le \beta_1 = 1.0643 - 0.007143 f'_c \le 0.85$, where f'_c is in MPa

   For CSA: $\beta_1 = 0.97 - 0.0025 f'_c \ge 0.67$, where f'_c is in MPa

5. Stress in the reinforcement is computed based on the strain at the centroid of each reinforcing bar.

6. All moments are referenced to the geometric centroid of the gross concrete section (neglecting the reinforcement).

7. For the ACI code, the nominal (unreduced) strength of the section is first computed on the basis of the strength of materials. This is then reduced to the design strength using the capacity reduction factors.

   The following reduction factors are used for the ACI code:
   ϕ_a is the reduction factor for pure axial compression
   ϕ_b is the reduction factor for flexure and axial tension
   ϕ_c is the reduction factor for flexure and axial compression

   For compressive design axial loads less than the smaller of $0.1 f'_c A_g$ and the balanced load, the program interpolates between ϕ_c and ϕ_b.

   For the CSA standard, the following factors are used:
   ϕ_s is the reduction factor for pure axial compression
   ϕ_b is the material resistance factor for steel

φ_c is the material resistance factor for concrete

8. If the reinforcement ratio falls below the code-specified minimum of 1%, the program has two options: consider the section as *Structural* or *Architectural*. For Architectural, the maximum concrete stress, f_c, is multiplied by the reinforcement ratio. This produces the same effect as reducing the concrete area to achieve a reinforcement ratio of 1%. For Structural, the specified concrete stress is used.

9. Under the Design option, the reinforcement ratio cannot be less than 0.5%. Under the Investigation option, any value can be used for the reinforcement ratio.

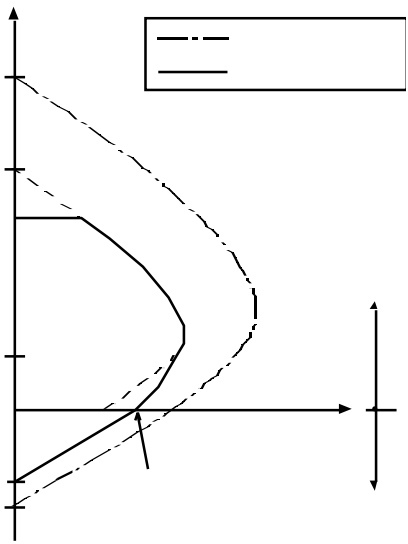


Figure 2-3 — Reduction Factors for Flexure and Axial Load per ACI code

B - Sign Convention

1. Looking at the section in plan, the positive x-axis points to the right and the positive y-axis points up.

2. Positive axial loads are compressive and negative axial loads are tensile.

3. The right-hand rule is used to determine the compression and tension faces of the section. A positive moment about the x-axis, *Mx*, produces tension at the top face of the section and compression at the bottom face. A positive moment about the y-axis, *My*, produces tension at the left face of the section and compression at the right face.

4. If service loads are input, moments at the upper (top) and lower (bottom) ends of the column are needed. Top and bottom moments of opposite signs produce single curvature bending. Top and bottom moments of the same sign produce double curvature bending.

   In the global coordinate system, the section at the upper end of the column coincides with PCACOL's local coordinate system where positive moments about the y-axis produce tension on the left side of the section. However, the global coordinate system produces an opposite effect on the section at the lower end of the column where positive moments about the y-axis produce tension on the right side of the section. Since PCACOL assumes the reinforcement pattern to be constant throughout the column height, the program internally changes the sign of the service moments at the lower end to convert from the global coordinate system to the local.

5. For uniaxial runs, the factored moments considered always produce positive bending on the section as described in item 3 above. This is valid for the cases when the section is symmetrical about the run axis. If the column bends in double curvature and the section is not symmetrical, a biaxial investigation run is recommended.

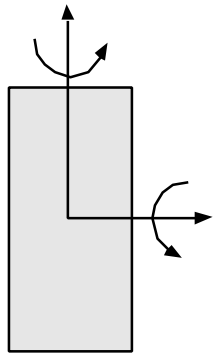


Figure 2-4 — PCACOL's positive local coordinate system

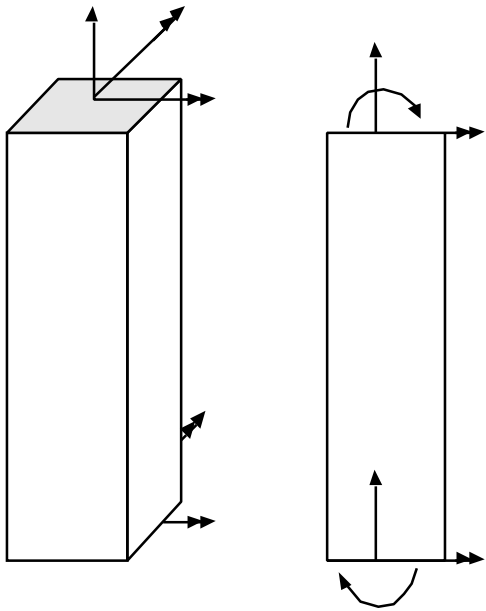


Figure 2-5 — PCACOL's positive global system

C - Section Investigation

1. The computations performed when investigating a section depend on the selected load mode:
   a) Factored loads - for the axial load of each load point, the moment capacity and the ratio of design-to-applied moment are computed. For a biaxial run, the computed *Mx* and *My* moment capacities are at the same angle as that produced by the applied *Mx* and *My* moments.
   b) Service loads - the program computes the factored loads using the input load combinations. If slenderness effects are to be checked and the column is found to be slender, the applied moments are magnified according to Procedures E and F. For each factored load, the same computations described in (a) above are performed.
   c) Control points - for several key points on the interaction diagram, the axial load, corresponding moment capacity and the corresponding neutral axis depth are computed. The key points are: pure compression, maximum compression, point where steel stress is zero, point where steel stress is 0.5 f_y, balanced point, pure flexure and pure tension.
   d) Axial loads – for each input axial load, the moment capacity and the corresponding neutral axis depth are computed.

2. The program also computes the interaction diagram (uniaxial runs) or the three-dimensional failure surface (biaxial runs) of the input section. The values of pure compressive axial load capacity and pure tensile axial load capacity are computed. These two values set the range within which the moment capacities are computed. A predetermined number of axial load values (within this range) are computed.
   a) For uniaxial runs, the moment capacity about the selected axis only is computed. The moment capacity about the orthogonal axis is ignored. To compute the moment capacity at a certain level of axial load, the neutral axis angle is held constant, parallel to the selected axis. The neutral axis depth is adjusted to arrive at the desired axial load capacity. This is done for all the predetermined values of axial load.

b) For biaxial runs, the same predetermined values of axial load are utilized. For each level of axial load, the section is rotated in 10-degree increments from 0 degrees to 360 degrees and the *Mx* and *My* moment capacities are computed. Thus for each level of axial load, an *Mx*-*My* contour is developed. Repeating this for the entire range of axial loads, the three-dimensional failure surface is computed.

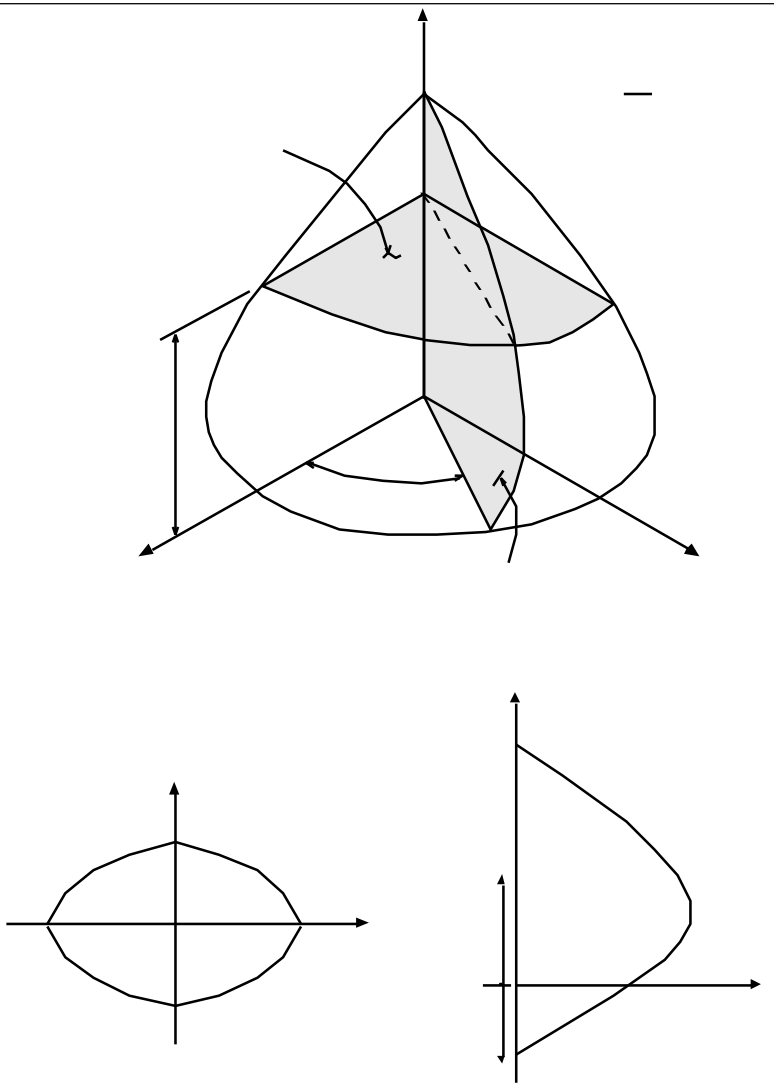


Figure 2-6— Interaction Surface for Combined Axial Load and Biaxial Bending

D- Section Design

1. Based on the specified minimum, maximum and increment specified for the section and the reinforcing bars, the program selects the smallest section with the least amount of reinforcement for which the load-moment capacity exceeds the applied loads. If service loads are input, they are factored using the input load combinations.

2. The program starts the design by trying the smallest section (minimum dimensions) and the least amount of reinforcing bars (smallest bar size and minimum number of bars). The program verifies that the ratio of provided reinforcement is always within the specified minimum and maximum ratios. Furthermore, the bar spacing is always kept greater than or equal to the larger of 1.5 times the bar diameter or 1.5 in. [40 mm] (unless otherwise specified by the user).

3. A section fails the design if, for the design axial load, the ratio of design-to-applied moment is less than 1.0(unless otherwise specified in the Design Criteria box).

4. Once a section passes the design, its capacity is computed and the computations explained in Procedure C above are performed.

E - Moment Magnification (Slenderness Effects) for Braced Columns

1. If properties of framing members are input, PCACOL computes the braced condition effective length factor, k_b, as the smaller of:

   $0.7 + 0.05\ (\psi_A + \psi_B)$
   $0.85 + 0.05\ \psi_{min}$

   Where ψ is the ratio of $\Sigma(EI/L)$ of columns to $\Sigma(EI/L)$ of beams in a plane at one end of the column, ψ_A and ψ_B are the values of ψ at the upper end and the lower end of the column, respectively, and ψ_{min} is the smaller of ψ_A and ψ_B.

   The value of k_b must always be between 0.5 and 1.0. The moment of inertia used in computing ψ is the gross moment of inertia multiplied by the cracked section coefficients (specified in the Slenderness Factors box).

2. The smaller and larger factored column end moments M_1 and M_2, respectively, are computed. M_1 is positive if the column is bent in single curvature and is negative if the column is bent in double curvature. M_2 is always positive.

3. For the ACI code, slenderness effects will be considered if

   $$\frac{k_{b\ u}}{r} \geq 34 - 12\frac{M_1}{M_2}$$

   For the CSA standard,

   $$\frac{k_{b\ u}}{r} \geq \frac{25 - 10M_1 / M_2}{\sqrt{P_f\ /(f_c' A_g)}}$$

   where $_u$ is the unsupported column length and r is the radius of gyration ($r = \sqrt{\frac{I_g}{A}}$).

   The ratio M_1/M_2 is always greater than or equal to -0.5.

4. If the ratio $k_{b\ u} / r$ exceeds 100, slenderness effects using this approximate procedure cannot be applied. A more exact method must be used. In this case, the procedure is aborted and a message is issued.

5. The factor C_m is computed as:

$$C_m = 0.6 + 0.4 \frac{M_1}{M_2} \geq 0.4$$

6. The factor β_d is computed as the ratio of maximum factored axial dead load to the total factored axial load for the load combination under consideration.

7. The quantity $(0.2 E_c I_g + E_s I_{se})$ is determined; where E_c is the modulus elasticity of concrete, E_s is the modulus elasticity of steel, I_g is the gross moment of inertia of the concrete section, and I_{se} is the moment of inertia of reinforcement. The product EI is computed as:

$$EI = \frac{0.2E_c I_g + E_s I_{s\,e}}{1 + \beta_d}$$

8. The critical buckling load, P_c, is computed as:

$$P_c = \frac{\pi^2 EI}{(k_{b\ u})^2}$$

9. The braced (non-sway) moment magnification factor, δ_b, is computed as:

$$\delta_b = \frac{C_m}{1 - \frac{P_u}{\phi_k P_c}} \geq 1.0$$

where the stiffness reduction factor, ϕ_k, is taken as 0.75 (may be modified using the Slenderness Factors box) and P_u is the factored axial load for the load combination under consideration.

10. The moment due to minimum eccentricity, e_{min}, is computed as:

$e_{min} = 0.6 + 0.03h$, where h is in in.
$e_{min} = 15 + 0.03h$, where h is in mm.

where h is the section dimension (diameter for circular sections) in the direction being considered. The moment due to the minimum eccentricity is $M_{min} = P_u\, e_{min}$.

11. The magnified moment, M_c is the larger of $\delta_b M_2$ or $\delta_b M_{min}$.

F- Moment Magnification (Slenderness Effects) for Sway Column

1. If properties of framing members are input, PCACOL computes the sway (unbraced) condition effective length factor, k_s, using the following simplified equations:

   If both ends are restrained and $\psi_m < 2$, $k_s = 0.05(20 - \psi_m)\sqrt{1 + \psi_m} \geq 1.0$

   If both ends are restrained and $\psi_m \geq 2$, $k_s = 0.9\sqrt{1 + \psi_m} \geq 1.0$

   If the upper end is hinged, $k_s = 2 + 0.3\psi_B$

   If the lower end is hinged, $k_s = 2 + 0.3\psi_A$

   If both ends are hinged, $k_s = 30$

   Where ψ is the ratio of $\Sigma(EI/L)$ of columns to $\Sigma(EI/L)$ of beams in a plane at one end of the column, ψ_A and ψ_B are the values of ψ at the upper end and the lower end of the column, respectively, and ψ_m is the average of ψ_A and ψ_B.

   An end is determined to be hinged if its ψ is very large. This happens in the case where $\Sigma(EI/L)$ of beams is too small (or zero) relative to the $\Sigma(EI/L)$ of columns at that end. In this case, the program outputs 999.9 for the value of ψ. The moment of inertia used in computing ψ is the gross moment of inertia multiplied by the cracked section coefficients (specified in the Slenderness Factors box).

2. The smaller and larger factored braced (non-sway) column end moments M_{1b} and M_{2b}, respectively, are computed. These are factored moments due to dead and live loads. M_{1b} is positive if the column is bent in single curvature and is negative if the column is bent in double curvature. M_{2b} is always positive.

3. The smaller and larger factored sway column end moments M_{1s} and M_{2s}, respectively, are computed. These are factored moments due to lateral loads. M_{1s} is positive if the column is bent in single curvature and is negative if the column is bent in double curvature. M_{2s} is always positive.

4. For the ACI code, slenderness effects will be considered if $k_{s\ u} / r \geq 22.0$. For the CSA standard, all sway columns are designed for slenderness effects.

5. If the ratio $k_{s\ u} / r$ exceeds 100, slenderness effects using this approximate procedure cannot be applied. A more exact method must be used. In this case, the procedure is aborted and a message is issued.

6. The quantity ($0.2\ E_c I_g + E_s I_{se}$) is determined; where E_c is the modulus elasticity of concrete, E_s is the modulus elasticity of steel, I_g is the gross moment of inertia of the concrete section, and I_{se} is the moment of inertia of reinforcement. The product *EI* is computed as:

$$EI = 0.2\ E_c I_g + E_s I_{se}$$

7. The critical buckling load, P_c, is computed as:

$$P_c = \frac{\pi^2 EI}{(k_{s\ u})^2}$$

8. The sway moment magnification factor, δ_s, is computed as:

$$\delta_s = \frac{1.0}{1 - \dfrac{\Sigma P_u}{\phi_k \Sigma P_c}} \geq 1.0$$

where the stiffness reduction factor, ϕ_k, is taken as 0.75.

ΣP_u is taken as the factored axial load for the load combination under consideration times the ratio $\Sigma P_u/P_u$.

ΣP_c is taken as the critical buckling load for the load combination under consideration times the ratio $\Sigma P_c/P_c$.

ϕ_k and the ratios $\Sigma P_u/P_u$ and $\Sigma P_c/P_c$ may be modified using the Slenderness Factors box.

9. If $\frac{u}{r} \le \frac{35}{\sqrt{\frac{P_u}{f'_c A_g}}}$, the magnified moment is computed as:

$$M_{1c} = M_{1b} + \delta_s M_{1s}$$

$$M_{2c} = M_{2b} + \delta_s M_{2s}$$

10. If $\frac{u}{r} > \frac{35}{\sqrt{\frac{P_u}{f'_c A_g}}}$, the braced moment magnification factor, δ_b, is computed according to steps 1 through 9 as described in Procedure D above. In step 7, however, β_d is computed for the load combination under consideration. The magnified moment is computed as:

$$M_{1c} = \delta_b (M_{1b} + \delta_s M_{1s})$$

$$M_{2c} = \delta_b (M_{2b} + \delta_s M_{2s})$$

G- Output Notation

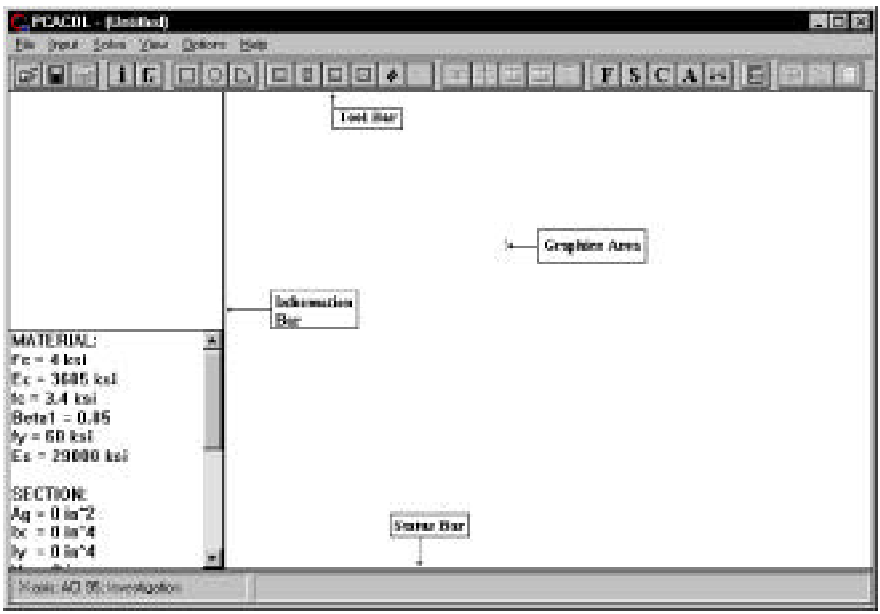
The following table explains some of the notations used in the output of the program:

Notation	ACI code	CSA standard
Pu	Ultimate factored axial load (P_u)	Factored axial load (P_f)
Mu	Ultimate factored moment (M_u)	Factored moment (M_f)
fMn	Moment capacity (ϕM_n)	Factored moment of resistance (M_r)

Chapter 3

The PCACOL Window

Most windows have certain elements in common, such as a title bar and menus. Not all windows, however, have very element. To learn more about how to use window elements, refer to your Microsoft Windows manual. The following is a list of the PCACOL window components.



Control-Menu Box;

The Control-menu box is located in the upper-left corner of the window and includes commands for sizing, moving, enlarging, restoring, and closing the window, as well as switching to other applications. To access the Control-menu box using the mouse, click the left mouse button on the box; using the keyboard, press ALT+ - (hyphen).

Title Bar:

The Title bar displays the name of the application (PCACOL in this case), along with the name of the current data file in use. If the data have not been saved into a file, the word *Untitled* is displayed in the Title bar.

Menu Bar:

The Menu bar is located directly below the Title bar. It lists the available menus. A menu contains a list of commands or actions you can execute.

Tool Bar:

The Tool bar is located directly below the Menu bar. It contains a collection of buttons (or icons) that provide a shortcut to accessing the menu commands. *The Tool bar buttons can only be accessed using the mouse.*

Information Bar:

The Information bar is located on the left side of the PCACOL window. The top part displays the column cross section. The bottom part is a list of selected input data echo.

Graphics Area:

The Graphics area covers most of the window. This is where the interaction diagram is shown.

Status Bar:

The Status bar is located along the bottom of the window. It displays important information such as units, design code, cursor position, and helpful messages.

Using the Menus

To access a menu item using the mouse, point to the menu item and click the left-mouse button. Each menu item can also be selected with the keyboard by simultaneously pressing the ALT key and the underlined letter of the menu you want to open. For example, to open the **File** menu, press ALT + F.

To close a menu without selecting a command, point to any blank area on the screen and click the left-mouse button, or press ESC.

To choose a command from a selected menu, point to that command and click the left-mouse button, or use the UP and DOWN ARROW keys until you reach the command and press ENTER.

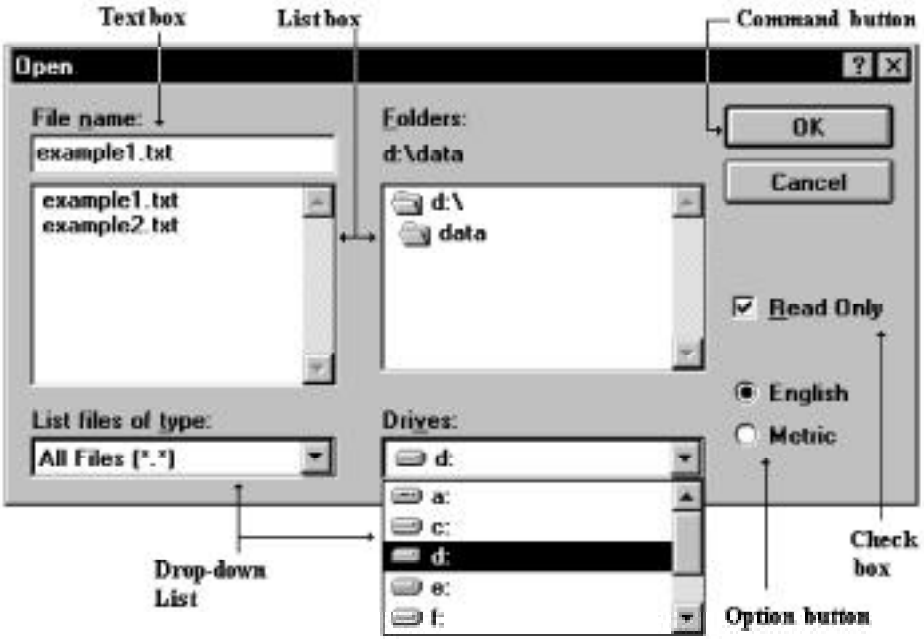
Menu Conventions

The following menu conventions are used in PCACOL.

Menu Convention	Meaning
Dimmed (or not visible) command.	The command is not available at the current time.
An ellipsis (…) following a command.	When the command is selected, a dialog box will appear, requesting data input or listing options to select from..
A check mark (✔) next to a command.	The command is in effect. When the check mark is removed (by selecting the command), the command is no longer in effect.

Using a Dialog Box

A dialog box is displayed to request information or to supply you with information. An ellipsis (…) after a menu command indicates a dialog box will appear when that command is elected. For example, if you choose the **Open** command from the **File** menu, the OPEN dialog box appears. In this dialog box, specify the name of the file you want to open.



Command Button:

The Command button (or Push button) is labeled to do exactly what it says. The OK and CANCEL buttons are common command buttons. A command button followed by an ellipsis (…) opens another dialog box. The currently selected button has a darker border than other buttons. You can choose the selected button by pressing ENTER.

Option Button:

The Option button (or Radio button) is one of a group of related options in which you can select only one. When an option is selected, the selected option button contains a black dot. To select an option with the mouse, click the mouse button on the option you want, or using the keyboard, press TAB to move to the group of options you want and use the arrow keys to select the option button.

Check Box:

The Check box is an option that can be turned on or off. Unlike option buttons, you can turn on as many check boxes as you want, even if they are in a group. When a check box is turned on, an "X" appears in the box. If you are using a mouse, click a blank check box to select it or click a selected check box to clear it. If you are using a keyboard, press TAB to move to the check box. Press the SPACEBAR to select the box. Press the SPACEBAR again to clear the selection.

Text Box:

The Text box is where data or information are entered. When you move into an empty text box, an *insertion point* (flashing vertical bar) appears. The text you type starts at the insertion point. If the box already contains text, it is selected and any text typed replaces it. You can also delete the existing text by pressing DEL or BACKSPACE.

List Box:

A List box displays a list of choices. If there are more choices than can fit in the box, scroll bars are provided so you can move quickly through the list. To select an item from a list box, click the scroll arrows until the item you want to select appears in the list box, then click on that item. If you are using the keyboard, use the arrow keys to scroll to the item you want.

Drop-Down List:

A Drop-down list appears initially as a rectangular box with the current selection marked. To see a list of available choices, click the arrow at the right of the box or press ALT + DOWN ARROW to open the box. You can now select another item by scrolling as you do with a list box.

Working with a List Box

A list box displays a list of choices or items. If there are more choices than can fit in the box, scroll bars are provided. The scroll bar is usually located to the right (vertical scroll bar) or at the bottom (horizontal scroll bar) of a list box.

To select a single item from a list box:

- Click the scroll arrows until the item you want to select appears in the list box. Click the item.
- Using the keyboard, use the UP or DOWN arrow key to scroll to the item you want.

To select multiple sequential items in a list:

- Click the first item you want to select and then drag the cursor to the last item you want to select. Alternatively, click on the last item you want to select. Both items, and all items in between, are selected. To cancel the selections, release the SHIFT key and click any item.
- Using the keyboard, use the UP ARROW or DOWN ARROW key to move the cursor to the first item you want to select. Press and hold down the SHIFT key. Continue to press the arrow key repeatedly until all the items you want are selected. To cancel the selections, release the SHIFT key and press one of the arrow keys..

To select multiple nonsequential items in a list:

- Press and hold down the CTRL key, and click each item you want to select. To cancel a selection, press and hold down the CTRL key, and click the item again. There are no keyboard equivalents for this procedure.

File Menu

The **File** menu is used for working with data files, printing results, and exiting the program.

New
Clears any input data and returns the data to the default values so that a new data file may be input.

Open
Opens an existing PCACOL data file.

Save
Saves the changes you have made to the current data file under that same filename.

Save As
Names or renames a data file.

Import
Reads geometry, reinforcement or loads data from an ASCII (Text) file.

Revert
Discards any changes to the data and returns to the last version of saved data. This option will only be available if the file has been modified since the last file save.

Print
Prints the results file and displayed curve and cross-section.

Exit
Quits the PCACOL program. If you have made any changes to your data and have not saved them, PCACOL will prompt you whether you want to save or abandon any changes you've made before you exit.

Input Menu

The **Input** menu includes commands used to input the data needed to define a problem.

General Information

Input labels to identify the input and select from several options that control the run.

Material Properties

Input the material properties for concrete and reinforcement.

Section

Rectangular: Input the dimensions of a rectangular section.
Circular: Input the dimension of a circular section.
Irregular: Execute IRRCOL to define an irregularly shaped section. Requires IRRCOL. See Chapter 7.

Reinforcement

All Sides Equal: Input a reinforcement pattern in which all the bars are of one size, and the number of bars is the same on all four sides of a rectangular layout or are equally spaced for a circular layout. *Available for rectangular and circular sections.*
Equal Spacing: Input a reinforcement pattern in which all bars are of one size and are uniformly spaced on all four sides of a rectangular layout. *Available for rectangular sections only.*
Sides Different: Input a reinforcement pattern in which each one of the four sides of a rectangular section has a certain number of bars and a certain bar size. *Available for rectangular sections only.*
Irregular Pattern: Input a reinforcement pattern in such a way that bars of any size can be placed anywhere within the column section.
Confinement: Input confinement-related data such as the capacity reduction factors and the size of ties associated with the longitudinal bar size.
Design Criteria: Input parameters that govern the design of the section such as: minimum and maximum reinforcement ratio, and minimum clear spacing.

Slenderness

Design Column: Input data needed for slenderness related to the column being considered.

Column Above/Below: Input data needed for slenderness related to the columns above and below the design column.

X-Beams: Input data needed for slenderness related to beams perpendicular to the x-axis framing into the design column.

Y-Beams: Input data needed for slenderness related to beams perpendicular to the y-axis framing into the design column.

Factors: Input factors that affect slenderness calculations.

Loads

Factored: Input factored axial loads and moments. *This command is available for non-slender columns only.*

Service: Input service-level (dead, live, and lateral) axial loads and moments about the active axis. Moments are input at the top and at the bottom of the column. *This command must be used for slender columns*.

Control Points: If chosen, the program will compute key points on the interaction diagram. In addition, the splice regions are shown on the interaction diagram. *This option is available for short (non-slender) columns under the investigation option only.*

Axial Loads: Input a group of axial loads with an initial value, a final value, and an increment. *This command is available for uniaxial short (non-slender) columns only.*

Load Combinations: Input the load factors and combinations used to combine the service loads.

Flip/Rotate Section

Flip an unsymmetrical section about an axis or rotate an irregular section.

Solve Menu

The **Solve** menu contains commands used to perform the design or investigation, trace through the design process, and send the output to a text file.

Execute
Executes the solver.

Design Trace
Toggles trace-through-a-design On and Off.

Results File
Toggles output-to-results-file On and Off.

View Menu

The **View** menu commands used to customize the display screen to suit your viewing needs, view the interaction diagram at a given angle, view the contour for a given axial load, superimpose an interaction diagram or a contour from another run, and view the results file.

Redraw
Redraws the displayed diagram or contour and clear any superimposed curves.

Information Bar
Shows or hides the Information bar.

Tool Bar
Shows or hides the Tool bar.

Status bar
Shows or hides the Status bar.

Grid
Shows or hides the grid.

Contour
View a contour of the failure surface sliced at a constant axial load.

Interaction
View an interaction diagram sliced at a constant (Mx, My) angle.

Superimpose
Superimpose a diagram from a previously saved run over the current diagram.

Results
View the results file after a successful run has been performed with the **Solve | Results File** option ON.

Help Menu

Contents and Index

Lists all the available help topics about the program.

Using Help

Provides help on using the Microsoft Windows help system.

About

Displays copyright information and the version number of the program.

Chapter 4

Operating the Program

This chapter is divided into sections that follow the order in which commands and options appear in the program.

Creating a New Data File

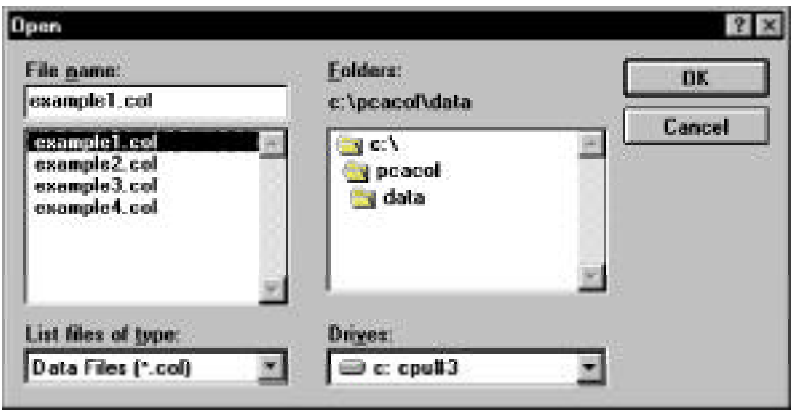
When PCACOL is first loaded, the program is ready to obtain input for a new file. The data will not have a filename associated with it, therefore, *Untitled* appears in the Title bar.

- Select **File | New**. This clears the screen and returns to the default settings of the program. If existing data have been changed prior to selecting the **New** command, PCACOL asks if you want to save the data. At any time after starting a new file, use the **File | Save As** command to give the file a name.

Opening an Existing Data File

You can open PCACOL data files that were previously saved to disk. PCACOL reads files that have been saved in binary format (with COL extension).

- Select **File | Open**. A dialog box appears.



- All files with the COL extension contained in the current drive and directory are displayed in the File Name list box.
- To open a file that exists in another drive or directory, select the drive or directory you want from the Drives drop-down list or the Directories list box respectively.
- From the File Name list box, select the file to be opened, or simply type its name in the text box.
- Choose the OK button.

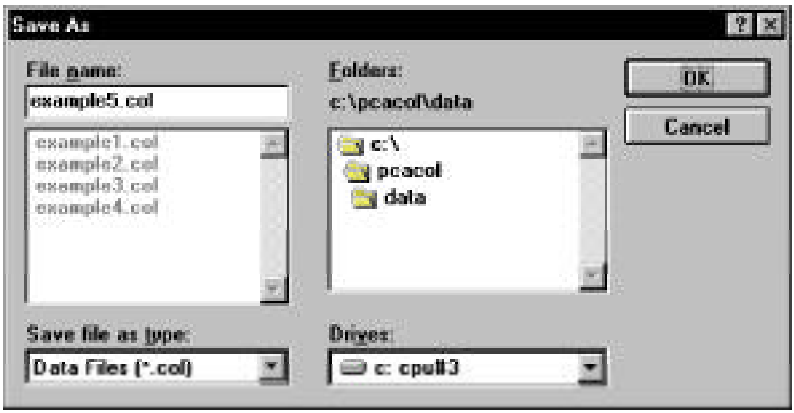
Saving the Data

To save the data with the same filename:

At anytime while editing a data file that has previously been saved under a filename, choose **File | Save** to save the changes under the same file name. Selecting the **File | Save** command before giving the data file a name displays the Save As dialog box.

To give the data a new or different filename:

- Select **File | Save As**. A dialog box appears.



- All files with COL extension contained in the current drive and directory are displayed in the File Name list box.
- To save the file to a drive or directory other than the default, select a different drive from the Drives drop-down list or a different directory from the Directories list box.
- In the File Name text box, type a new filename. If no file extension is specified, .COL will be appended to the specified filename.
- Choose the OK button.

Importing Data

Geometry, reinforcement or load data may be imported from a text file. The import file must be saved in pure ASCII (text) format.

- Select **File | Import**. A dialog box appears.

- Select the type of data you want to import: Service Loads, Factored Loads, Reinforcement or Geometry.
- Choose the OK button. The Open Import file box is displayed.
- All files with the TXT extension contained in the current drive and directory are displayed in the File Name list box.
- To open a file that exists in another drive or directory, select the drive or directory you want from the Drives drop-down list or the Directories list box respectively.
- From the File Name list box, select the file to be opened, or simply type its name in the text box.
- Choose the OK button.

Printing the Results

Once a successful run has been performed you may print the displayed diagram. Also, if the **Solve | Results File** option was selected, you may print the results file. PCACOL sends the printout to the active printer.

- Select **File | Print**. A dialog box appears.

- Check the applicable option(s): Screen and/or Results File.
- Choose the OK button. A message box appears informing you that the file is being printed. Choose Cancel to terminate the print job.

General Information

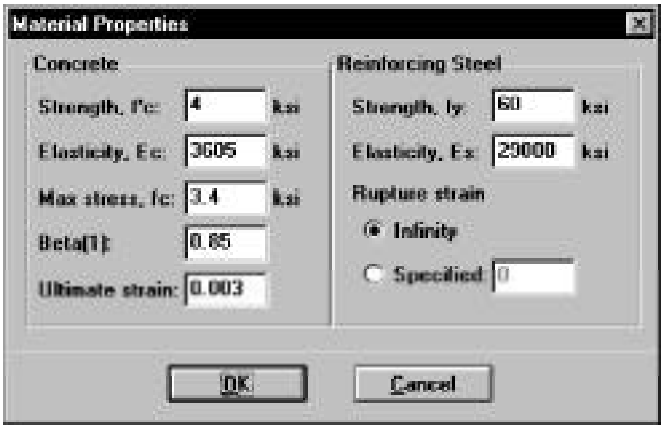
The **General Information** command allows you to define or change options that will affect the rest of the input.

- Select **Input | General Information**. A dialog box appears.

- Enter the project, column, and engineer labels.
- Select the units of measure, English or Metric.
- Select the axis about which the column capacity is to be computed: X-axis, Y-axis, or Biaxial.
- Select the design code to conform to: ACI 318-95 or CSA A23.3-94.
- Choose a run option: Investigation or Design.
- Specify whether slenderness effects are to be considered or not.
- Choose the OK button.

Material Properties

- Select **Input | Material Properties**. A dialog box appears.

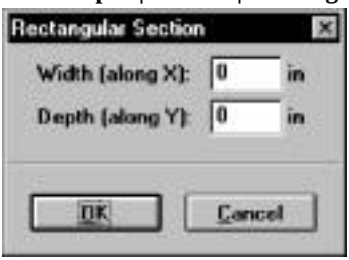


- Enter the concrete compressive strength, f'c.
  The modulus of elasticity, Ec; the constant stress intensity at the maximum usable strain used to calculate the force on concrete, fc; the factor used to determine the depth of the stress block, Beta(1); and the maximum usable strain at the extreme concrete compression fiber, are automatically computed and displayed. Modify any of the values if different that computed.
- For the reinforcing steel, enter the yield strength, fy, the modulus of elasticity, Es, and the rupture strain.
- Choose the OK button.

Section | Rectangular

Inputting for the Investigation Option:

- Select **Input | Section | Rectangular**. A dialog box appears.



- Enter the width and depth of the rectangular section. Note that width is measured along the x-axis and depth is measured along the y-axis.
- Choose the OK button.

Inputting for the Design Option:

- Select **Input | Section | Rectangular**. A dialog box appears.

- Enter the starting and ending values for each dimension (width and depth) along with an increment for each dimension. Note that the starting and ending values may be the same (in which case the increment must be zero).
- Choose the OK button.

Section | Circular

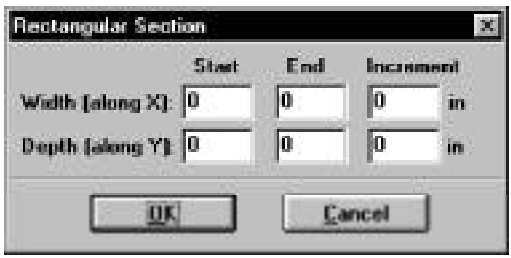
Inputting for the Investigation Option:

- Select **Input | Section | Circular**. A dialog box appears.

- Enter the diameter of the column section.
- Choose the OK button.

Inputting for the Design Option:

- Select **Input | Section | Circular**. A dialog box appears.



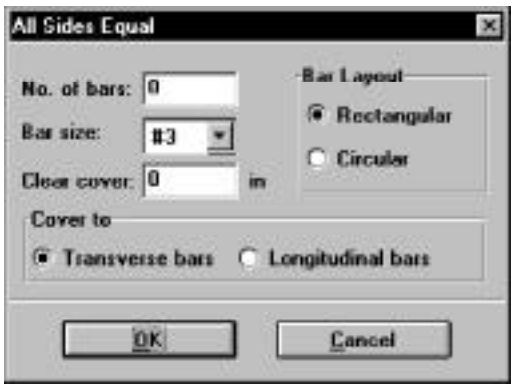
- Enter the starting and ending values the diameter along with an increment. Note that the starting and ending values may be the same (in which case the increment must be zero).
- Choose the OK button.

Reinforcement | All Sides Equal

The **Input | Reinforcement | All Sides Equal** command allows you to enter the total number of bars in the column section. The program attempts to divide the bars equally between the four sides of a rectangular layout then equally space them on each side. Corner bars are associated with the top and bottom sides, and if the total number of bars is not a multiple of four, the top and bottom sides get the extra bars. For a circular layout, the program attempts to equally space the bars.

Inputting for the Investigation Option:

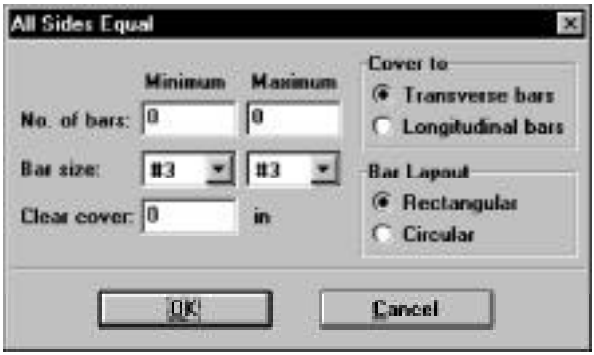
- Select **Input | Reinforcement | All Sides Equal**. A dialog box appears.



- Enter the total number of bars in the section (*must be at least 4 and must be a multiple of 2 for a rectangular layout*).
- Select the bar size.
- Enter the clear cover and specify whether it is to the transverse (ties) or to the longitudinal bars.
- Select a bar layout, Circular or Rectangular. The default is Circular for a circular section and Rectangular for a rectangular section.
- Choose the OK button.

Inputting for the Design Option:

- Select **Input | Reinforcement | All Sides Equal**. A dialog box appears.



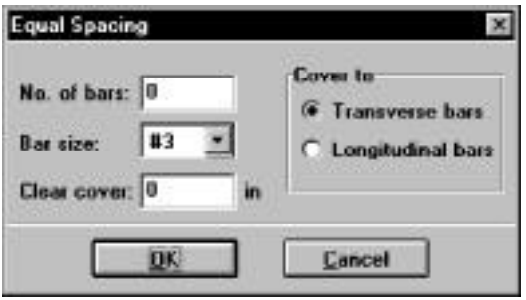
- Enter the minimum and maximum total number of bars in the section (*must be at least 4 and must be a multiple of 4 for a rectangular layout*).
- Select the minimum and maximum bar sizes along with the clear cover.
- Enter the clear cover and specify whether it is to the transverse (ties) or to the longitudinal bars.
- Select a bar layout, Circular or Rectangular. The default is Circular for a circular section and Rectangular for a rectangular section.

Reinforcement | Equal Spacing

The **Input | Reinforcement | Equal Spacing** command allows you to enter the total number of bars in the column section. The program attempts to place the bars equally spaced on all four sides of a rectangular layout. Corner bars are associated with the top and bottom sides and if the total number of bars is not a multiple of 4, the top and bottom sides will get the extra bars.

Inputting for the Investigation Option:

- Select **Input | Reinforcement | Equal Spacing**. A dialog box appears.



- Enter the total number of bars in the section (*must be at least 4 and a multiple of 2*).
- Select the bar size.
- Enter the clear cover and specify whether it is to the transverse (ties) or to the longitudinal bars.
- Choose the OK button.

Inputting for the Design Option:

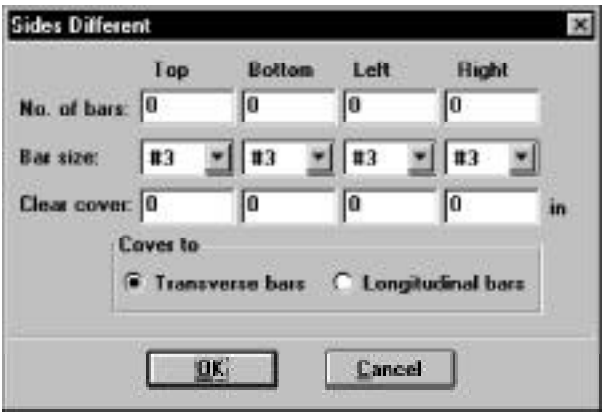
- Select **Input | Reinforcement | Equal Spacing**. A dialog box appears.

- Enter the minimum and maximum total number of bars in the section (*must be a multiple of 4*).
- Select the minimum and maximum bar sizes.
- Enter the clear cover and specify whether it is to the transverse (ties) or to the longitudinal bars.
- Choose the OK button.

Reinforcement | Sides Different

Inputting for the Investigation Option:

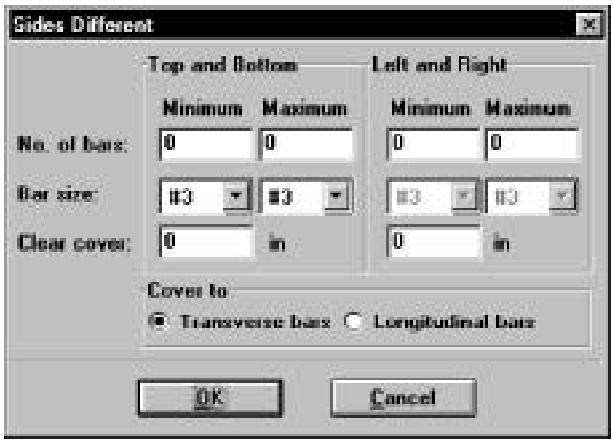
- Select **Input | Reinforcement | Sides Different**. A dialog box appears.



- Enter the number of bars to be placed on each of the top, bottom, left and right sides of the rectangular section. *Note that corner bars are associated with the top and bottom sides.*
- Select the bar size and enter the clear cover for each of the top, bottom, left and right sides of the section.
- Specify whether the clear cover entered is to the transverse (ties) or to the longitudinal bars.
- Choose the OK button.

Inputting for the Design Option:

- Select **Input | Reinforcement | Sides Different**. A dialog box appears.



- In the Top and Bottom group, enter the minimum and maximum total number of bars to be placed on the top and bottom sides, along with the bar size and the clear cover. *The number of bars must be greater than 4 and a multiple of 2. Corner bars are associated with the top and bottom sides.*
- Select the minimum and maximum bar size and enter the clear cover.
- Repeat the steps above for the Left and Right sides.
- Specify whether the clear cover entered is to the transverse (ties) or to the longitudinal bars.
- Choose the OK button.

Reinforcement | Irregular

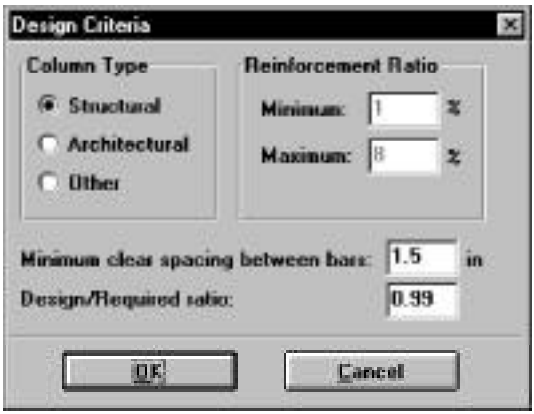
The **Input | Reinforcement | Irregular Pattern** command allows you to specify any number of bars (up to 500) to be placed anywhere within the column section. Each bar may have any cross-sectional area. *This command is available with the Investigation option only.*

- Select **Input | Reinforcement | Irregular Pattern**. If you have the IRRCOL module and the section is irregular, the IRRCOL window, where the section reinforcement may be graphically input, is displayed. Otherwise, a dialog box appears.

- Enter the bar area, along with its x- and y-coordinates.
- Choose the Insert button. The data are added to the list box.
- Repeat the steps above for each entry.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button. To change the area for several bars at once, select those bars from the list box, enter the new area in the Area text box, and choose Modify.
- To delete a bar entry, select it from the list box and choose the Delete button.
- Choose the OK button.

Reinforcement | Design Criteria

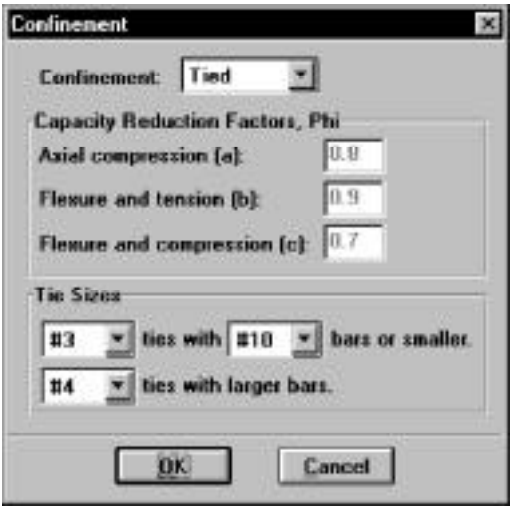
- Select **Input | Reinforcement | Design Criteria**. A dialog box appears. *This command is available with the Design option only.*



- Specify whether the column is to be considered Structural (default), Architectural, or Other. Notice that the minimum and maximum reinforcement ratios change depending on the selection.
- To modify the limiting reinforcement ratios, choose Other. Enter the minimum and maximum reinforcement ratios in the corresponding text boxes.
- Enter the minimum clear bar spacing and the design-to-required ratio.
- Choose the OK button.

Reinforcement | Confinement

- Select **Input | Reinforcement | Confinement**. A dialog box appears.



- Select the type of bar confinement: Tied (default), Spiral, or Other. The capacity reduction factors change depending on the selection.
- To modify the capacity reduction factors, choose Other for confinement type.

  For the ACI code:

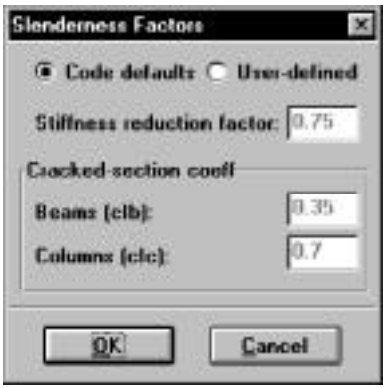
  Phi(a) is the reduction factor for pure axial compression
  Phi(b) is the reduction factor flexure and axial tension
  Phi(c) is the reduction factor for combined flexure and axial compression

  For the CSA code:

  Phi(a) is the reduction factor for pure axial compression
  Phi(s) is the material resistance factor for steel
  Phi(c) is the material resistance factor for concrete
- Specify the tie size associated with the particular longitudinal bar sizes. The code requirements are used as defaults.
- Choose the OK button.

Slenderness | Factors

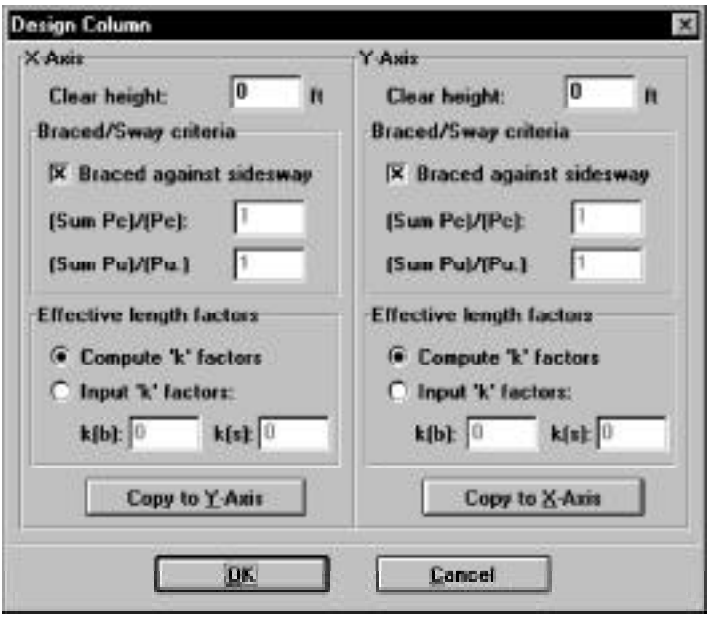
- Select **Input | Slenderness | Factors**. A dialog box appears. *Note: this command is available for slender columns only.*



- The default values, per the design code, for the stiffness reduction factor (used to compute the moment magnification factors) and the cracked section coefficients for beams and columns are displayed. To modify these values, select the User-defined option and enter your values.
- Choose the OK button.

Slenderness | Design Column

- **Input | Slenderness | Design Column**. A dialog box appears. The x-axis or y-axis data groups may be disabled depending on the active run axis. *Note: this command is available for slender columns only.*

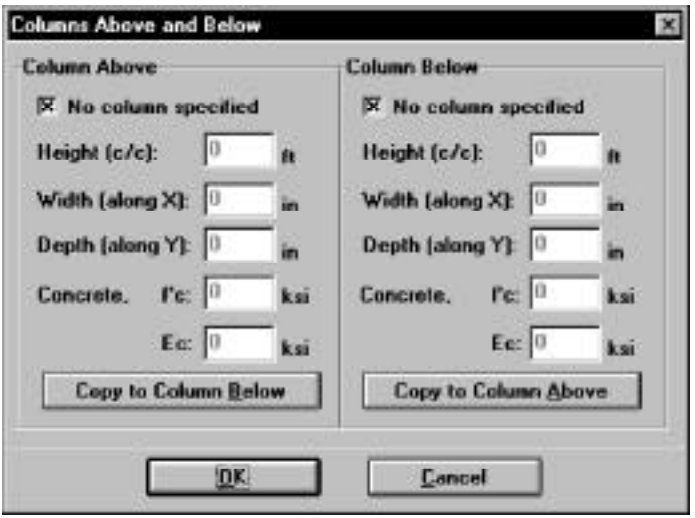


- Enter the clear column height in the corresponding text box.
- Specify whether the column is braced against sidesway or not by selecting or clearing the Braced Against Sidesway option.
- For an unbraced column, enter the ratio of the sum of the critical load of all story columns to the critical load of the column under consideration, (Sum Pc)/Pc. Also, enter the ratio of the sum of the factored axial load of all story columns to the factored axial load of the column under consideration, (Sum Pu)/Pu.

- In the Effective Length Factors group, specify whether the k factor(s) are to be user-input or computed by the program, by selecting the corresponding option.
- If you chose Input k Factors, enter the effective length factors for braced and sway conditions, Kb and Ks, respectively.
- Repeat the steps above for each active axis.
- Choose the OK button.

Slenderness | Columns Above and Below

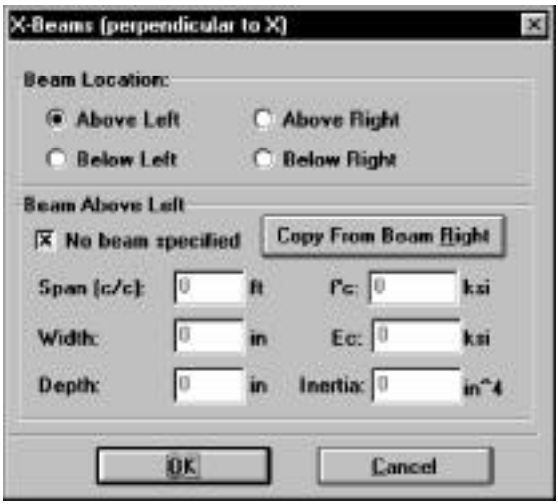
- **Input | Slenderness | Columns Above/Below**. A dialog box appears. *Note: this command is available for slender columns only for which the Compute k Factors option is selected in the Design Column dialog box.*



- In the Column Above group, specify whether or not a column above the design column exists, by selecting or clearing the No column specified option.
- If a column above exists; enter the center-to-center height, the width (along the x-axis), the depth (along the y-axis), the concrete compressive strength, and the concrete modulus of elasticity of the column above in the corresponding text boxes. For circular sections, enter zero for the Depth and input the section diameter in the Width box. If zero is entered for both Width and Depth, the dimensions of the design column will be automatically used.
- Repeat the same for the column below or, if the column above and the column below are the same, choose the Copy to Column Below button.
- Choose the OK button.

Slenderness | X-Beams

- **Input | Slenderness | X-Beams**. A dialog box appears. Beams defined using this command span perpendicular to the x-axis. *Note: this command is available for slender columns only for which the Compute k Factors option is selected in the Design Column dialog box and the run axis is biaxial or uniaxial x-axis.*



- From the Beam Location group, choose the appropriate option. The data group below updates depending on the selection.
- For the selected beam, specify whether or not a beam exists, by selecting or clearing the No Beam Specified option.
- If a beam exists; enter the center-to-center span length, width, depth, concrete compressive strength, concrete modulus of elasticity, and moment of inertia (if different than computed) of the beam.
- Repeat the steps above for the beams above right, below left, and below right.
- Choose the OK button.

Slenderness | Y-Beams

- Select **Input | Slenderness | Y-Beams**. A dialog box appears. Beams defined using this command span perpendicular to the y-axis. *Note: this command is available for slender columns only for which the Compute k Factors option is selected in the Design Column dialog box and the run axis is biaxial or uniaxial y-axis.*

- From the Beam Location group, choose the appropriate option. The data group below updates depending on the selection.
- For the selected beam, specify whether or not a beam exists, by selecting or clearing the No Beam Specified option.
- If a beam exists; enter the center-to-center span length, width, depth, concrete compressive strength, concrete modulus of elasticity, and moment of inertia (if different than computed) of the beam.
- Repeat the steps above for the beams above right, below left, and below right.
- Choose the OK button.

Loads | Factored

Whether designing or investigating a short (non-slender) column, factored loads consisting of applied axial loads and moments about the active axis may be input.

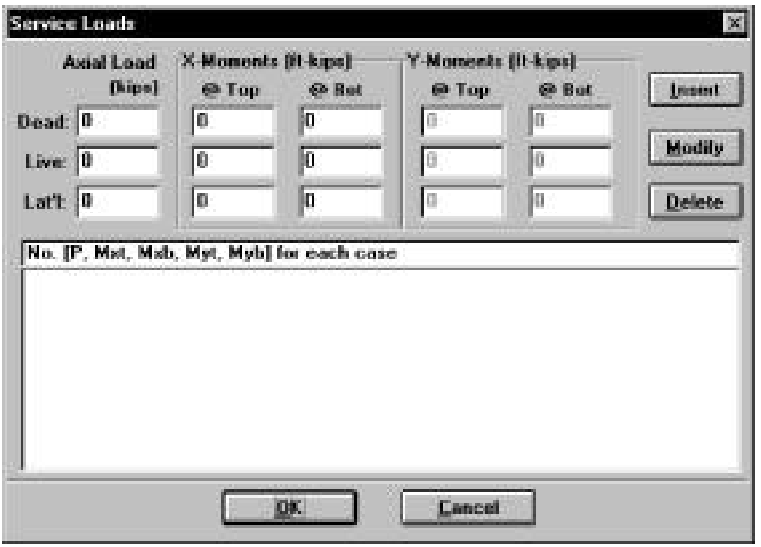
- Select **Input | Loads | Factored**. A dialog box appears.

- Enter the factored axial load in the Load text box.
- Depending on the active axis, enter the factored moment about the x-axis, the factored moment about the y-axis, or both.
- Choose the Insert button. The data are added to the list box.
- Repeat the steps above for each factored load entry.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button.
- To delete an entry, choose it from the list box and choose the Delete button.
- Choose the OK button.

Loads | Service

When designing or investigating a short or slender column, service level loads consisting of dead, live, and lateral axial loads and moments at both column ends about the active axis may be input. These loads are then factored and combined based on the load combinations.

- Select **Input | Loads | Service**. A dialog box appears.



- For each of the load cases: dead, live and lateral, enter the axial load and the applicable moment (about x or y) at the column top and bottom.
- Choose the Insert button. The data are added to the list box.
- Repeat the steps above for each entry.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button.
- To delete an entry, choose it from the list box and choose the Delete button.
- Choose the OK button.

Load | Control Points

A non-slender column may be investigated without inputting any loads. The capacity of the section is computed along with other important information. *This command is not available for slender columns or under the Design option.*

- Select **Input | Loads | Control Points**.

  In the output file, key points on the interaction diagram are listed. For each point, the axial load and bending moment capacities, along with the corresponding neutral axis depth, are listed.

  For a biaxial run, these points are listed for bending about positive and negative X and Y axes.

  For a uniaxial runs, the bar-splice regions are indicated on the interaction diagram.

Loads | Axial Loads

When investigating a short column (uniaxial runs only), grouped factored applied axial loads may be input. The corresponding moment capacity for each load is computed and output. *This command is not available under the Design option, for biaxial runs, or if the column is slender.*

- Select **Input | Loads | Axial Loads**. A dialog box appears.

- To input a range of axial loads, enter the initial, final and increment values in the corresponding text boxes. To input a discrete (single) axial load, enter its value in the Initial text box, and enter zero in the Final and Increment text boxes.
- Choose the Insert button. The data are added to the list box.
- Repeat the steps above for each axial load entry you may have.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button.
- To delete an entry, choose it from the list box and choose the Delete button.
- Choose the OK button.

Loads | Load Combinations

PCACOL allows you to specify up to eight load combinations. The service load input under the load cases (dead, live and lateral) are combined based on these combinations. A minimum of one combination must be input.

- Select **Input | Loads | Load Combinations**. A dialog box appears. For a new file, the default combinations per the selected code are shown in the list box.

- To add a new combination, enter the load factors or multipliers for each load case, Dead, Live and Lateral.
- Choose the Insert button. The data are added to the list box.
- Repeat the steps above for each load combination. Up to eight combinations may be defined.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button.
- To delete an entry, choose it from the list box and choose the Delete button.
- Choose OK when done.

Flipping / Rotating the Section

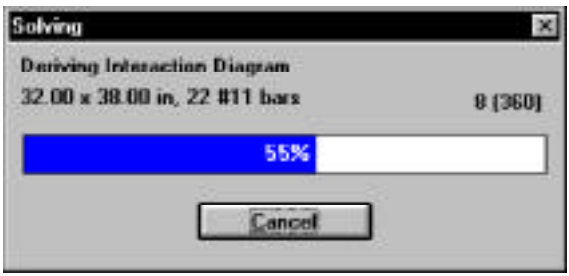
This command is available for Irregular sections and for sections with Sides Different or Irregular reinforcement.

- Select **Input | Flip/Rotate Section**. A dialog box appears.

- Select one of the options: Flip section about x-axis, Flip section about y-axis, or Rotate section.The last option is available for irregular sections only.
- If Rotate section is selected, enter the angle of rotation (clockwise is positive).
- Choose the OK button.

Executing a Run

- Select **Solve | Results File**. This is an option that may be turned ON or OFF by choosing it again. With this option ON, an output file will be created while the program is solving. The output file is a text file that contains an echo of the input along with slenderness calculation output, load combinations, and the critical points on the interaction diagram.
- Select **Solve | Design Trace (design option only)**. This is an option that may by turned ON or OFF be choosing it again. With this option ON, if designing a column section, the program will pause at each cross section and reinforcement layout it tries, and display the cross section with its bars in the Information bar.
- Select **Solve | Execute**. If more data are needed to solve the problem, an error box appears and you will be prompted for the required data. If all the needed data are input, the solver portion of PCACOL will start executing. The Solving box appears with a progress indicator. Click Cancel to terminate the solution process.



After a successful run, the interaction diagram or the contour plot is displayed in the Graphics area of the window.

Viewing the Results

Once a successful run has been performed with the **Execute | Results File** option ON, you may view the results file.

- Select **View | Results**. PCACOL executes the PCAVIEW program, an applet that enables viewing of text-based files.
- Use the scroll bars or the cursor keys to navigate through the file.
- Select **Exit** to quit PCAVIEW and return to PCACOL.

Changing the Startup Defaults

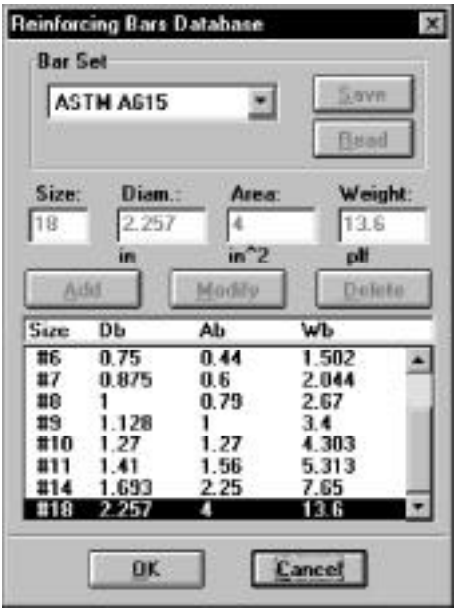
You can change some default settings that PCACOL uses when it is started or when the **File | New** command is executed.

- Select **Options | Startup Defaults**. A dialog box appears.

- Enter the full path of the directory to which PCACOL defaults when opening a data file.
- Select the system of units, design code, and rebar database to be used as defaults whenever a new file is started.
- Enter the name of the program to be used by PCACOL when viewing results. By default, PCACOL uses PCAVIEW.
- Enter the number of lines per page to be used in formatting the results file.
- Choose the Save button to save these options in the PCAPROGS.INI file.

Changing the Rebar Database

- Select **Options | Rebar Database**. A dialog box appears.



To select a different set:

- From the Bar Set drop-down list, select the set to be used.
- Choose the OK button.

To create or modify a user-defined set:

- From the Bar Set drop-down list, select User-defined. The Add, Delete and Modify buttons are enabled.
- To edit an entry, select it from the list box, modify its values, and choose the Modify button.
- To delete an entry, select it from the list box and choose the Delete button.
- To add an entry, enter the bar size, diameter, area and unit weight in the corresponding text box, and choose the Add button. The data are added to the list box.
- A user-defined set may be saved to disk for use in other files. To do so, choose the Save button after inputting the bars. To retrieve a saved set, choose the Read button. Rebar database is saved to the PCACOL.BAR file.
- Choose the OK button.

Superimposing Diagrams

After executing a run and selecting the **File | Save** command, the program saves the diagram data in a separate file that has the same filename as the input file, with an IAD extension. This file may be later read by the program and two diagrams (of different files) may be superimposed for the purpose of comparison. Note that the run option (uniaxial or biaxial) of the two files must be the same

- Select **View | Superimpose**. The open dialog box appears.
- All files with the IAD extension contained in the current drive and directory are displayed in the File Name list box. To open a file that exists in another drive or directory, select the drive or directory you want from the Drives drop-down list or the Directories list box respectively.
- From the File Name list box, select the file to be opened, or simply type its name in the text box.
- Choose the OK button. If the data is successfully read, the diagram from the IAD file (shown dotted) is superimposed over the currently displayed diagram.
- To clear any superimposed diagrams, select the **View | Redraw** command.

Viewing the Diagram

For a uniaxial run, the program computes and displays the P-M interaction diagram. For a biaxial run, the program computes the three-dimensional failure surface (P, Mx, My). The failure surface can be viewed in one of two ways:

a) P-M interaction diagram (vertical slice at an angle measured from Mx)
b) Mx-My contour curve (horizontal slice at a constant axial load).

Whenever the mouse cursor moves over the Graph area, the axial load, moment, and eccentricity are displayed in the Status bar.

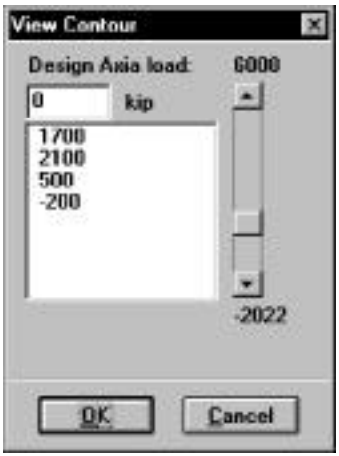
Viewing the Interaction Diagram

- Select **View | Interaction**. A dialog box appears.

- Specify whether you want to enter an angle or the values of the moments that produce a view angle by selecting either the Angle or Moments option, respectively.
- If the Angle option was selected, select an angle from the list, use the scroll bar to pick an angle, or simply enter the value in the text box.
- If the Moments option was selected, enter the values of Mx and My which produce a view angle. The value of that view angle is computed and shown in the Angle text box.
- Choose the OK button.

Viewing the Contour

- Select **View | Contour**. A dialog box appears.



- Select an axial load from the list, use the scroll bar to pick a load, or simply enter the value in the text box.
- Choose the OK button.

Chapter 5

Examples

Example 1 - Design of a Short Column

Determine the size and reinforcement of a column section to carry the following factored loads:

Pu = 1,700 kips, Mux = 1,900 ft-kips, Muy = 1,200 ft-kips

The column section is rectangular, limited in width to 32 in but the depth could be as large as 48 in. The reinforcing bars are equally distributed on all four sides and the reinforcement ratio should be limited to 3%. Use 6-ksi concrete and 60-ksi reinforcing steel.

1. From the **File** menu, choose **New**. Any input data is cleared and the default values are restored.

2. From the **Input** menu, choose **General Information**.
   - Input the project header.
   - Select English units and ACI 319-95 code.
   - Select Biaxial for run axis, Design for run option and No for slenderness.
   - Choose Ok.

3. From the **Input** menu, choose **Material Properties**.
   - Input `6` for the concrete strength. Other properties are computed and will be accepted.
   - Choose Ok.

4. From the **Input** menu, choose **Section | Rectangular**.
   - For the section Width, input `32`, `32` and `0` under Start, End and Increment.
   - For the section Depth, input `36`, `48` and `2` under Start, End and Increment.
   - Choose Ok.

5. From the **Input** menu, choose **Reinforcement | Equal Spacing**.
   - Input `20`-#9 bars for minimum and `40`-#11 bars for maximum.
   - Input `1.5` in for the cover and select Transverse bars.
   - Choose Ok.

6. From the **Input** menu, choose **Reinforcement | Design Criteria**.
   - Under Column Type, select Other.
   - Input `3` for Maximum Reinforcement Ratio.
   - Choose Ok.

7. From the **Input** menu, choose **Loads | Factored**.
   - Input `1700`, `1900` and `1200` for the Load, X-Moment and Y-Moment, respectively.
   - Choose Insert to add the entry to the list box.
   - Choose Ok.

8. From the Solve menu, choose **Execute**.
   - The solver of the program is started and, upon completion, displays the selected section that passes the design (32x38 with 22-#11 bars).
   - The Mx-My contour at zero-axial load is displayed in the Graphics area.

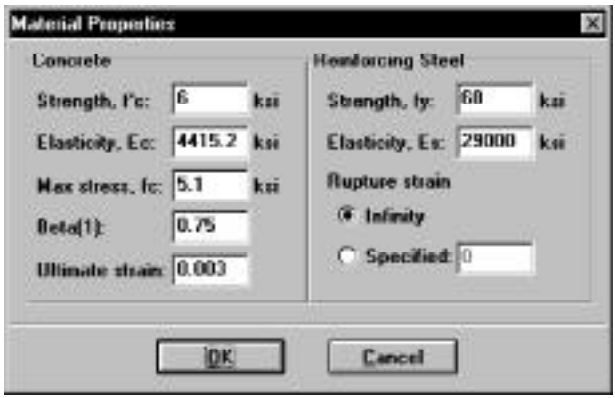
9. From the **View** menu, choose **Interaction**.
   - The Angle list displays the Mx-My angle for the input moment(s). Choose 32 from the list.
   - Choose Ok. The PM interaction diagram (at 32-deg Mx-My angle) is displayed in the Graphics area. The load point is indicated on the diagram.

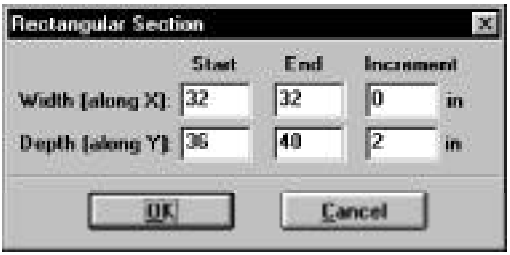
10. From the **View** menu, choose **Contour**.
    - The Design Axial Load list displays the value(s) of any input load(s). Choose 1700 from the list.
    - Choose Ok. The Mx-My contour at P=1700 kips is displayed in the Graphics area. The load point is indicated on the diagram.

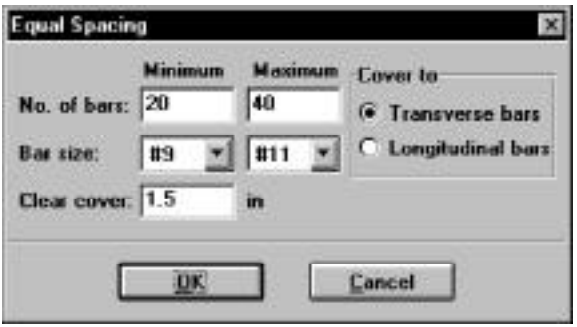
11. From the **View** menu, choose **Results**.
    - Page through the results file.
    - Choose **Exit** to quit the PCAVIEW program and get back to PCACOL.

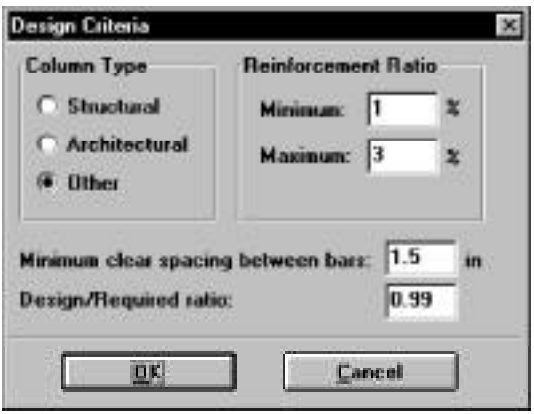
12. From the **File** menu, choose **Print**.
    - Insure that both Screen and Results File are checked. Choose Ok.

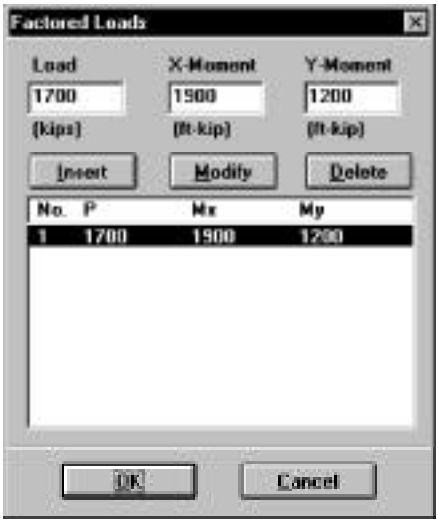
General Information
Labels
Project:
PCACOL Manual, Biaxial
Column:
Ex_1
Engineer:
PCA
Units
English
Metric
Design Code
ACI 318-95
CSA A23.3-94
Run Axis
About X-Axis
About Y-Axis
Biaxial
Run Option
Investigation
Design
Consider slenderness?
Yes
No
OK
Cancel


Material Properties
Concrete
Strength, f'c:
6
ksi
Elasticity, Ec:
4415.2
ksi
Max stress, fc:
5.1
ksi
Beta(1):
0.75
Ultimate strain:
0.003
Reinforcing Steel
Strength, fy:
60
ksi
Elasticity, Es:
29000
ksi
Rupture strain
Infinity
Specified:
0
OK
Cancel


Rectangular Section
Start
End
Increment
Width [along X]:
32
32
0
in
Depth [along Y]:
36
40
2
in
OK
Cancel

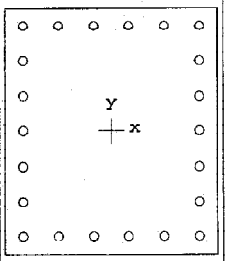

Equal Spacing
Minimum
Maximum
Cover to
No. of bars:
20
40
Transverse bars
Bar size:
#9
#11
Longitudinal bars
Clear cover:
1.5
in
OK
Cancel


Design Criteria
Column Type
Structural
Architectural
Other
Reinforcement Ratio
Minimum:
1
%
Maximum:
3
%
Minimum clear spacing between bars:
1.5
in
Design/Required ratio:
0.99
OK
Cancel


Factored Loads
Load
X-Moment
Y-Moment
1700
1500
1200
(kips)
(ft-kip)
(ft-kip)
Insert
Modify
Delete
No. P Mx My
1 1700 1900 1200
OK
Cancel

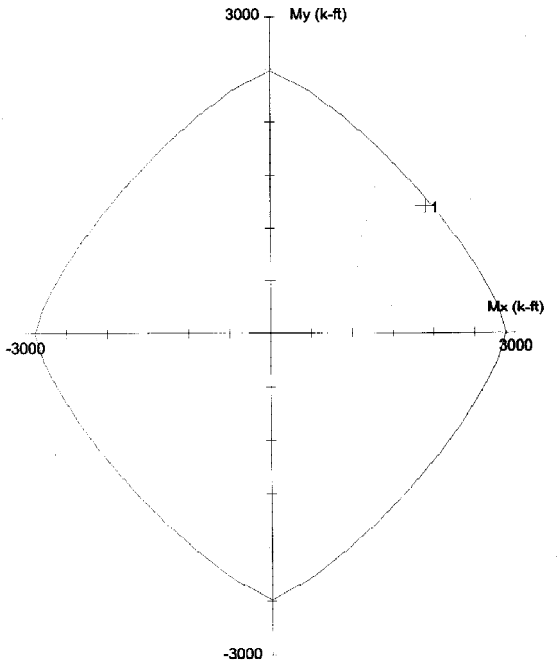
View Contour
Design Axis load:
5600
1700
kip
1700
-1853
OK
Cancel

View Interaction
Angle (Mx, My):
0
32
degrees
32
359
Moments:
Mx
My
0
0
k-ft
OK
Cancel



32 x 38 in

Code: ACI 318-95
Units: English
Run axis: Biaxial
Run option: Design
Slenderness: Not considered
Column type: User-defined
Bars: ASTM English
Date: 10/29/98
Time: 09:39:36

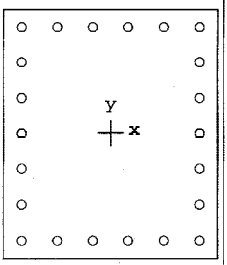


P = 1700 kip

PCACOL V 3.00 (PCA 1998) - Licensed to: Portland Cement Assoc, Skokie, IL

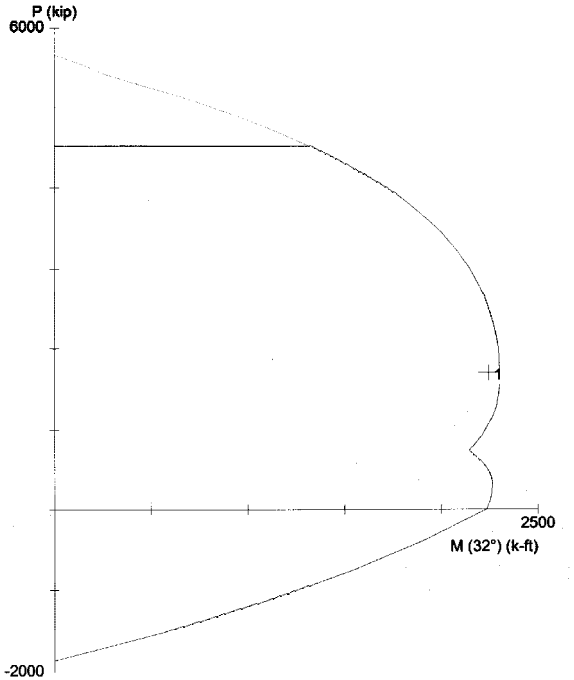
File: D:\DATA\PCACOL\EXAMPLE1.COL
Project: PCACOL Manual, Biaxial
Column: Ex_1
Engineer: PCA

f'c = 6 ksi	fy = 60 ksi	Ag = 1216 in^2	22 #11 bars
Ec = 4415 ksi	Es = 29000 ksi	As = 34.32 in^2	Rho = 2.82%
fc = 5.1 ksi	e_rup = Infinity	Xo = 0.00 in	Ix = 146325 in^4
e_u = 0.003 in/in		Yo = 0.00 in	Iy = 103765 in^4
Beta1 = 0.75		Clear spacing = 3.91 in	Clear cover = 2.00 in
Confinement: Tied	phi(a) = 0.8, phi(b) = 0.9, phi(c) = 0.7		



32 x 38 in

Code: ACI 318-95
Units: English
Run axis: Biaxial
Run option: Design
Slenderness: Not considered
Column type: User-defined
Bars: ASTM English
Date: 10/29/98
Time: 09:39:45



PCACOL V 3.00 (PCA 1998) - Licensed to: Portland Cement Assoc, Skokie, IL

File: D:\DATA\PCACOL\EXAMPLE1.COL
Project: PCACOL Manual, Biaxial
Column: Ex_1 Engineer: PCA

f'c = 6 ksi	fy = 60 ksi	Ag = 1216 in^2	22 #11 bars
Ec = 4415 ksi	Es = 29000 ksi	As = 34.32 in^2	Rho = 2.82%
fc = 5.1 ksi	e_rup = Infinity	Xo = 0.00 in	Ix = 146325 in^4
e_u = 0.003 in/in		Yo = 0.00 in	Iy = 103765 in^4
Beta1 = 0.75		Clear spacing = 3.91 in	Clear cover = 2.00 in
Confinement: Tied	phi(a) = 0.8, phi(b) = 0.9, phi(c) = 0.7		

```
10/29/98 PCACOL V3.00 - PORTLAND CEMENT ASSOCIATION -                    Page   1
09:39:16 Licensed to: Portland Cement Assoc, Skokie, IL                   EXAMPLE1

 General Information:
 ====================
    File Name: D:\DATA\PCACOL\EXAMPLE1.COL
    Project:  PCACOL Manual, Biaxial
    Column:   Ex_1                          Engineer: PCA
    Code:     ACI 318-95                    Units: English

    Run Option: Design                      Slenderness: Not considered
    Run Axis:   Biaxial                     Column Type: User-defined

 Material Properties:
 ====================
    f'c   = 6 ksi                           fy   = 60 ksi
    Ec    = 4415.21 ksi                     Es   = 29000 ksi
    fc    = 5.1 ksi                         Rupture strain = Infinity
    Ultimate strain = 0.003 in/in
    Beta1 = 0.75

 Section:
 ========
    Rectangular: Width = 32 in              Depth = 38 in

    Gross section area, Ag =  1216 in^2
    Ix =  146325 in^4                       Iy =  103765 in^4
    Xo =  0 in                              Yo =  0 in

 Reinforcement:
 ==============
    Rebar Database: ASTM English
    Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)
    ---- --------- -----------   ---- --------- -----------   ---- --------- -----------
    #  3     0.38        0.11    #  4     0.50        0.20    #  5     0.63        0.31
    #  6     0.75        0.44    #  7     0.88        0.60    #  8     1.00        0.79
    #  9     1.13        1.00    # 10     1.27        1.27    # 11     1.41        1.56
    # 14     1.69        2.25    # 18     2.26        4.00

    Confinement: Tied; #3 ties with #10 bars,  #4 with larger bars.
    phi(a) = 0.8,  phi(b) = 0.9,  phi(c) = 0.7

    Layout: Rectangular
    Pattern: Equal Bar Spacing  (Cover to transverse reinforcement)
    Total steel area, As = 34.32 in^2 at 2.82%
    22 #11   Cover = 1.5 in

 Factored Loads and Moments with Corresponding Capacities:
 =========================================================
                Pu          Mux          Muy        fMnx         fMny
    No.        kip         k-ft         k-ft        k-ft         k-ft    fMn/Mu
    --- ---------- ---------- ---------- ---------- ---------- --------
      1     1700.0       1900.0       1200.0       1945.9       1232.4     1.025

    *** Program completed as requested! ***
```

Example 2 - Capacity of a Braced Column

Check the capacity of an interior column, located on the third floor of a braced frame. The frame has 5 bays (24 ft on centers) and 6 floors (14 ft floor-to-floor height).

All interior columns are 18x18 and all exterior columns are 16x16. All interior columns of the third floor have 8-#10 bars with 1.5 in cover. All beams are 48x12 and support a 6-in slab. The moment of inertia of the T-section is 13,823 in^4. Use 4-ksi concrete and 60-ksi steel for all beams and columns.

The service load values for the column under consideration are as follows:

	Dead	Live
P	230	173
Top Mx	2	108
Bottom Mx	2	-100

The column is to be checked for the following load combination:

U1 = 1.4 D + 1.7 L

1. From the **File** menu, choose **New**. Any input data is cleared and the default values are restored.

2. From the **Input** menu, choose **General Information**.
   - Input the project header.
   - Select English units and ACI 319-95 code.
   - Select About X for run axis, Investigation for run option and Yes for slenderness.
   - Choose Ok.

3. From the **Input** menu, choose **Material Properties**.
   - Input `4` for the concrete strength. Other properties are computed and will be accepted.
   - Choose Ok.

4. From the **Input** menu, choose **Section | Rectangular**.
   - Input `18` and `18` for the section width and depth.
   - Choose Ok.

5. From the **Input** menu, choose **Reinforcement | Equal Spacing**.
   - Input `8`-#10 bars and `1.5` in for the cover and select Transverse bars.
   - Choose Ok.

6. From the **Input** menu, choose **Slenderness | Design Column**.
   - Input `13` for the column clear height.
   - Check the Braced against sidesway option and select Compute ‘k’ factors.
   - Choose Ok.

7. From the **Input** menu, choose **Slenderness | Columns Above/Below**.
   - Clear the No column specified option.
   - Input `14` for the column height (center-to-center) and leave the other data as is.
   - Choose Copy to Column Below.
   - Choose Ok.

8. From the **Input** menu, choose **Slenderness | X-Beams**.
   - Choose Above Left.
   - Clear the No beam specified option.
   - Input `24` for the span (center-to-center).
   - Input `48` and `12` for the width and depth, respectively.
   - Input `13823` for the moment of inertia.
   - Leave the other data as is.
   - Choose Above Right and click on Copy From Beam Left.
   - Choose Below Left and click on Copy From Beam Above.
   - Choose Below Right and click on Copy From Beam Above.
   - Choose Ok.

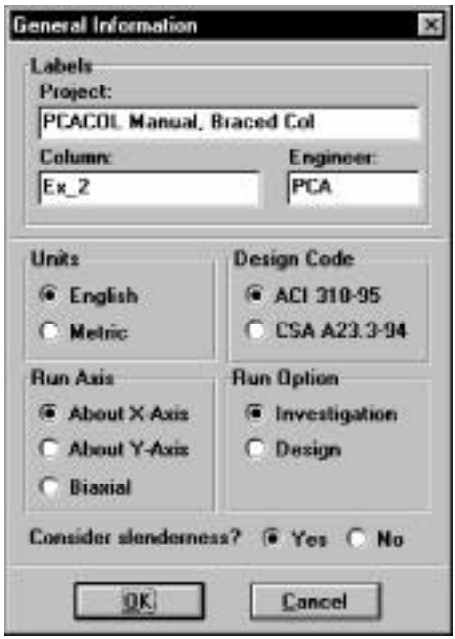
9. From the **Input** menu, choose **Loads | Service**.
   - Input `230`, `2.0` and `2.0` for the dead Load, X-Moment and Y-Moment, respectively.
   - Input `173`, `108` and `-100` for the live Load, X-Moment and Y-Moment, respectively.
   - Choose Insert to add the entry to the list box.
   - Choose Ok.

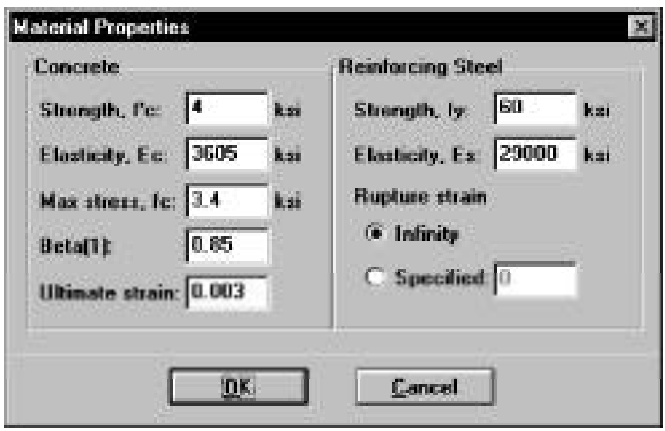
10. From the **Input** menu, choose **Loads | Load Combinations**.
    - If the list displays four combinations, select the last three and choose Delete. and eave the other data as is. only need to consider the first combination (1.4 Dead + 1.7 Live).
    - Choose Ok.

11. From the **Solve** menu, choose **Execute**.
    - The solver of the program is started and, upon completion, displays the interaction diagram of the section with the load point plotted within the diagram.

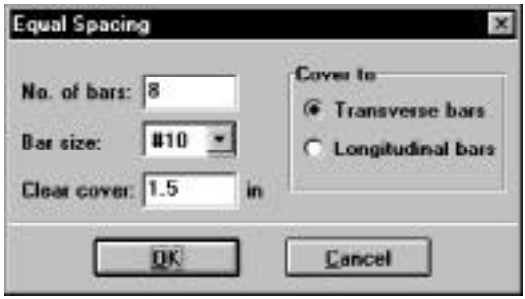
12. From the **View** menu, choose **Results**.
    - Page through the results file.
    - Choose Exit to quit the PCAVIEW program and get back to PCACOL.

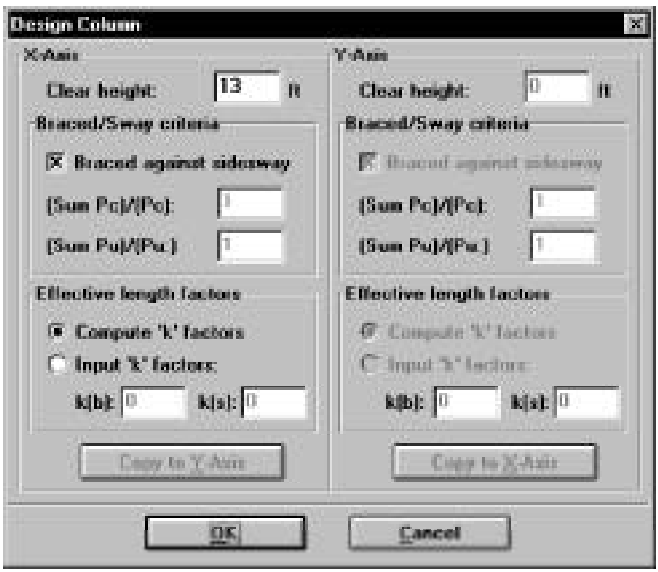
13. From the **File** menu, choose **Print**.
    - Insure that both Screen and Results File are checked. Choose Ok

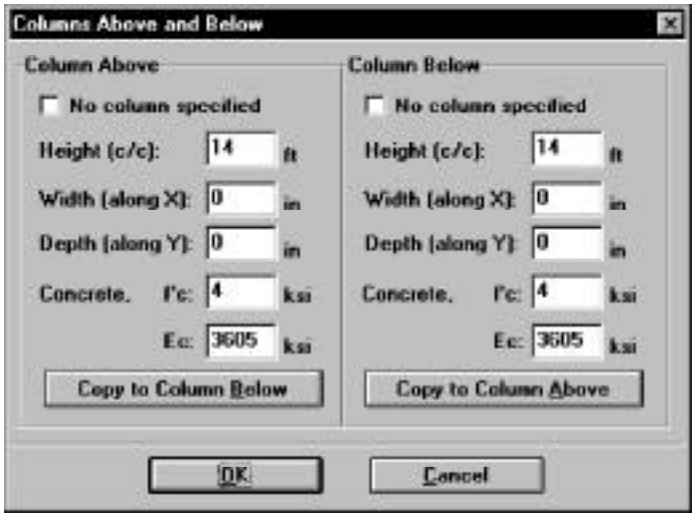

General Information
Labels
Project:
PCACOL Manual, Braced Col
Column:
Ex_2
Engineer:
PCA
Units
English
Metric
Design Code
ACI 318-95
CSA A23.3-94
Run Axis
About X Axis
About Y-Axis
Biaxial
Run Option
Investigation
Design
Consider slenderness? Yes No
OK
Cancel


Material Properties
Concrete
Strength, f'c: 4 ksi
Elasticity, Ec: 3605 ksi
Max stress, fc: 3.4 ksi
Beta[1]: 0.85
Ultimate strain: 0.003
Reinforcing Steel
Strength, fy: 60 ksi
Elasticity, Es: 29000 ksi
Rupture strain
Infinity
Specified 0
OK
Cancel

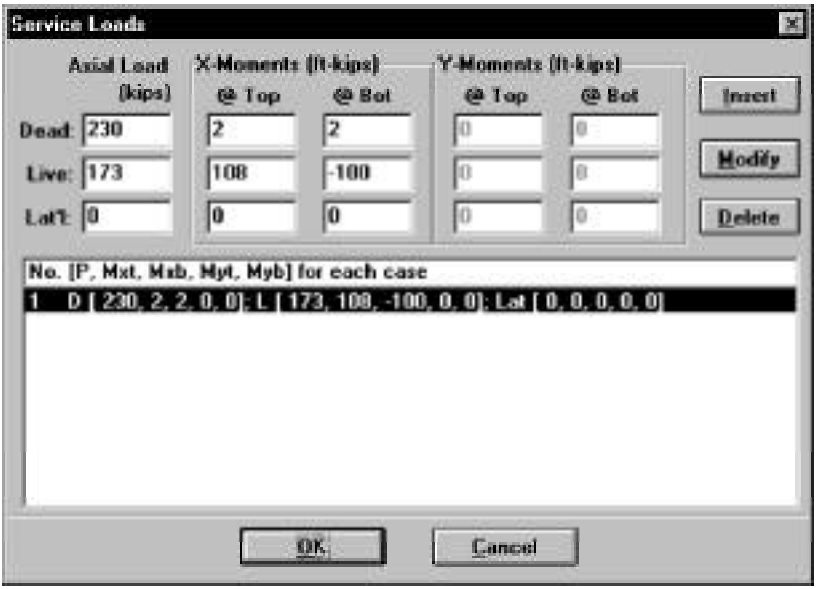
Rectangular Section
Width (along X): 18 in
Depth (along Y): 18 in
OK
Cancel


Equal Spacing
No. of bars: 8
Bar size: #10
Clear cover: 1.5 in
Cover to
Transverse bars
Longitudinal bars
OK
Cancel

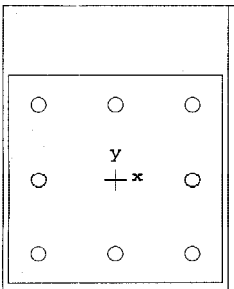

Design Column
X-Axis
Clear height: 13 ft
Braced/Sway criteria
Braced against sidesway
(Sum Pc)/(Pc):
(Sum Pu)/(Pu):
Effective length factors
Compute 'k' factors
Input 'k' factors:
k(b):
k(s):
Copy to Y-Axis
Y-Axis
Clear height: ft
Braced/Sway criteria
Braced against sidesway
(Sum Pc)/(Pc):
(Sum Pu)/(Pu):
Effective length factors
Compute 'k' factors
Input 'k' factors:
k(b):
k(s):
Copy to X-Axis
OK
Cancel


Columns Above and Below
Column Above
No column specified
Height (c/c): 14 ft
Width (along X): 0 in
Depth (along Y): 0 in
Concrete. f'c: 4 ksi
Ec: 3605 ksi
Copy to Column Below
Column Below
No column specified
Height (c/c): 14 ft
Width (along X): 0 in
Depth (along Y): 0 in
Concrete. f'c: 4 ksi
Ec: 3605 ksi
Copy to Column Above
OK
Cancel

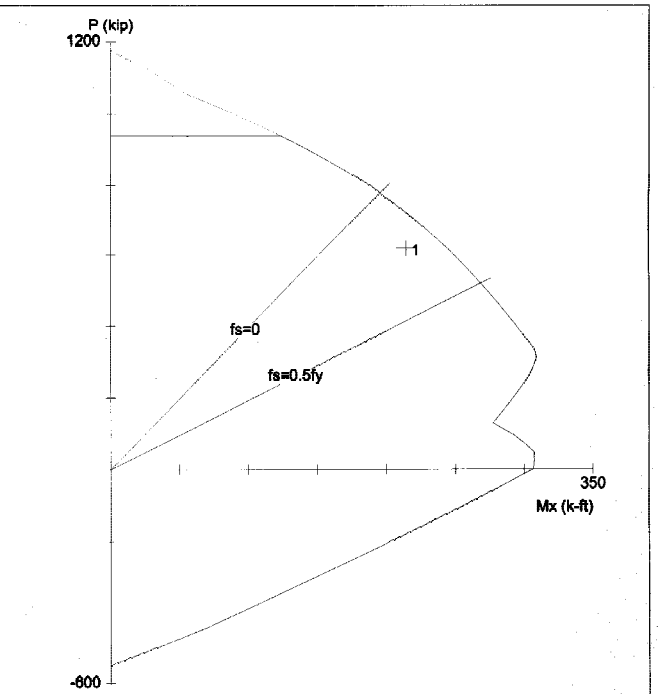
X-Beams (perpendicular to X)
Beam Location:
Above Left
Above Right
Below Left
Below Right
Beam Above Left
No beam specified
Copy From Beam Right
Span (c/c): 24 ft
f'c: 4 ksi
Width: 48 in
Ec: 3605 ksi
Depth: 12 in
Inertia: 13823 in^4
OK
Cancel


Service Loads
Axial Load (kips)
X-Moments (ft-kips)
@ Top
@ Bot
Y-Moments (ft-kips)
@ Top
@ Bot
Insert
Modify
Delete
Dead: 230
2
2
0
0
Live: 173
108
-100
0
0
Lat'l: 0
0
0
0
0
No. [P, Mxt, Mxb, Myt, Myb] for each case
1 D [230, 2, 2, 0, 0]; L [173, 108, -100, 0, 0]; Lat [0, 0, 0, 0, 0]
OK
Cancel

Load Combinations
1.4
Dead +
1.7
Live +
0
Lat.
Add
Modify
Delete
Combo
Dead
Live
Lateral
U1
1.4
1.7
0
OK
Cancel



18 x 18 in



Code: ACI 318-95
Units: English
Run axis: About X-axis
Run option: Investigation
Slenderness: Considered
Column type: User-defined
Bars: ASTM English
Date: 10/29/98
Time: 09:40:08

PCACOL V 3.00 (PCA 1998) - Licensed to: Portland Cement Assoc, Skokie, IL

File: D:\DATA\PCACOL\EXAMPLE2.COL
Project: PCACOL Manual, Braced Col

Column: Ex_2		Engineer: PCA	
f'c = 4 ksi	fy = 60 ksi	Ag = 324 in^2	8 #10 bars
Ec = 3605 ksi	Es = 29000 ksi	As = 10.16 in^2	Rho = 3.14%
fc = 3.4 ksi	e_rup = Infinity	Xo = 0.00 in	Ix = 8748 in^4
e_u = 0.003 in/in		Yo = 0.00 in	Iy = 8748 in^4
Beta1 = 0.85		Clear spacing = 5.22 in	Clear cover = 1.87 in
Confinement: Tied	phi(a) = 0.8, phi(b) = 0.9, phi(c) = 0.7		

kx(braced) = 0.91698, kx(sway) = N/A

```
10/29/98 PCACOL V3.00 - PORTLAND CEMENT ASSOCIATION -                    Page   1
09:40:02 Licensed to: Portland Cement Assoc, Skokie, IL                  EXAMPLE2

  General Information:
  ====================
     File Name: D:\DATA\PCACOL\EXAMPLE2.COL
     Project:  PCACOL Manual, Braced Col
     Column:   Ex_2                          Engineer: PCA
     Code:     ACI 318-95                    Units: English

     Run Option: Investigation               Slenderness: Considered
     Run Axis:   X-axis                      Column Type: Structural

  Material Properties:
  ====================
     f'c   = 4 ksi                           fy   = 60 ksi
     Ec    = 3605 ksi                        Es   = 29000 ksi
     fc    = 3.4 ksi                         Rupture strain = Infinity
     Ultimate strain = 0.003 in/in
     Beta1 = 0.85

  Section:
  ========
     Rectangular: Width = 18 in              Depth = 18 in

     Gross section area, Ag =  324 in^2
     Ix =  8748 in^4                         Iy =  8748 in^4
     Xo =  0 in                              Yo =  0 in

  Reinforcement:
  ==============
     Rebar Database: ASTM English
     Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)
     ---- --------- -----------   ---- --------- -----------   ---- --------- -----------
     #  3      0.38        0.11   #  4      0.50        0.20   #  5      0.63        0.31
     #  6      0.75        0.44   #  7      0.88        0.60   #  8      1.00        0.79
     #  9      1.13        1.00   # 10      1.27        1.27   # 11      1.41        1.56
     # 14      1.69        2.25   # 18      2.26        4.00

     Confinement: Tied; #3 ties with #10 bars,  #4 with larger bars.
     phi(a) = 0.8,  phi(b) = 0.9,  phi(c) = 0.7

     Layout: Rectangular
     Pattern: Equal Bar Spacing  (Cover to transverse reinforcement)
     Total steel area, As = 10.16 in^2 at 3.14%
     8 #10   Cover = 1.5 in

  Slenderness:
  ------------
     Sway Criteria:
     --------------
     X-axis: Braced column.

                    Height       Width       Depth           I        f'c          Ec
     Column Axis        ft          in          in        in^4        ksi         ksi
     ------ ---- --------- ---------- ---------- ----------- -------- -----------
     Design  X          13          18          18        8748        4        3605
     Above   X          14           0           0        8748        4        3605
     Below   X          14           0           0        8748        4        3605
```

X-Beams Location	Length ft	Width in	Depth in	I in^4	f'c ksi	Ec ksi
Above Left	24	48	12	13823	4	3605
Above Right	24	48	12	13823	4	3605
Below Left	24	48	12	13823	4	3605
Below Right	24	48	12	13823	4	3605

Effective Length Factors:

Axis	Psi(top)	Psi(bot)	k(Braced)	k(Sway)	klu/r
X	2.170	2.170	0.917	(N/A)	27.53

Moment Magnification Factors:

Stiffness reduction factor, phi(K) = 0.75
Cracked-section coefficients: cI(beams) = 0.35; cI(columns) = 0.7

0.2*Ec*Ig + Es*Ise (X-axis) = 1.56e+007 kip-in^2

X-axis	Braced				Sway	
Ld/Comb	Pc(kip)	Betad	Cm	Delta	Pc(kip)	Delta
1 U1	4946	0.523	0.959	1.150	--- N/A ---	

Load Combinations:

U1 = 1.400*Dead + 1.700*Live + 0.000*Lateral

Service Loads:

No.	Load Case	Axial Load kip	Mx @ Top k-ft	Mx @ Bot k-ft	My @ Top k-ft	My @ Bot k-ft
1	Dead	230.0	2.0	2.0	0.0	0.0
	Live	173.0	108.0	-100.0	0.0	0.0
	Lat1	0.0	0.0	0.0	0.0	0.0

Factored Loads and Moments with Corresponding Capacities:

No.	Load Combo	Pu kip	Mux k-ft	fMnx k-ft	fMn/Mu
1	1 U1	616.1	214.3	244.2	1.139

*** Program completed as requested! ***

Example 3 - Capacity of a Sway Column

Check the capacity of an interior column, located on the third floor of a sway (unbraced)frame. The frame has 5 bays (24 ft on centers) and 6 floors (14 ft floor to-floor height).

All interior columns are 18x18 and all exterior columns are 16x16. All interior columns of the third floor have 8-#10 bars with 1.5 in cover. All exterior columns of the third floor have 8-#8 bars. All beams are 48x12 and support a 6-in slab. The moment of inertia of the T-section is 13,823 in^4. Use 4-ksi concrete and 60-ksi steel for all beams and columns.

In addition to the gravity load analysis, a lateral load analysis due to wind was performed. The service load values for the column under consideration are as follows:

	Dead	Live	Wind
P	230	173	6
Top Mx	2	108	84
Bottom Mx	2	-100	-70

The column is to be checked for the following load combinations:

U1 = 1.4 D + 1.7 L
U2 = 0.75 (1.4 D + 1.7 L + 1.7 W)
U3 = 0.9 D + 1.3 W

Based on preliminary computations, Pc for interior columns is 2,466 kips and Pc for exterior columns is 1,120 kips. With 2 exterior columns and 4 interior columns, SPc = 2x1,120 + 4 x 2,466 = 12,104 kips. For the column under consideration, the ratio SPc/Pc = 12,104/2466 = 4.91.
Theoretically, the ratio ΣΣPu/Pu would be different for each combination. Since the program allows for one ratio only, the most conservative one (the ratio due to the second load combination) will be used. For combination U2, the total factored gravity loads on the floor is ΣPu = 2,320 kips and the factored load on the column is Pu = 470 kips. For the column under consideration, the ratio ΣPu/Pu = 2,320/470 = 4.94.

1. From the **File** menu, choose **New**. Any input data is cleared and the default values are restored.

2. From the **Input** menu, choose **General Information**.
   - Input the project header.
   - Select English units and ACI 319-95 code.
   - Select About X for run axis, Investigation for run option and Yes for slenderness.
   - Choose Ok.

3. From the **Input** menu, choose **Material Properties**.
   - Input `4` for the concrete strength. Other properties are computed and will be accepted.
   - Choose Ok.

4. From the **Input** menu, choose **Section | Rectangular**.
   - Input `18` and `18` for the section width and depth.
   - Choose Ok.

5. From the **Input** menu, choose **Reinforcement | Equal Spacing**.
   - Input `8`-#10 bars and `1.5` in for the cover and select Transverse bars.
   - Choose Ok.

6. From the **Input** menu, choose **Slenderness | Design Column**.
   - Input `13` for the column clear height.
   - Clear the Braced against sidesway option and input `4.91` and `4.94` for the Sum Pc/Pc and Sum Pu/Pu, respectively.
   - Select Compute ‘k’ factors
   - Choose Ok.

7. From the **Input** menu, choose **Slenderness | Columns Above/Below**.
   - Clear the No column specified option.
   - Input `14` for the column height (center-to-center) and leave the other data as is.
   - Choose Copy to Column Below.
   - Choose Ok.

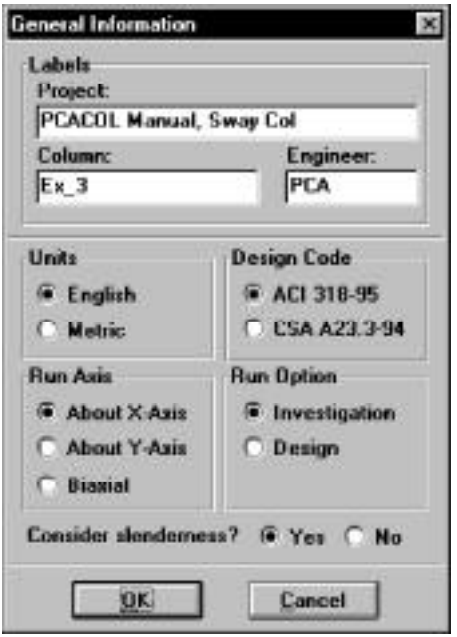
8. From the **Input** menu, choose **Slenderness | X-Beams**.
   - Choose Above Left.
   - Clear the No beam specified option.
   - Input `24` for the span (center-to-center).
   - Input `48` and `12` for the width and depth, respectively.
   - Input `13823` for the moment of inertia.
   - Leave the other data as is.
   - Choose Above Right and click on Copy From Beam Left.
   - Choose Below Left and click on Copy From Beam Above.
   - Choose Below Right and click on Copy From Beam Above.
   - Choose Ok.
9. From the **Input** menu, choose **Loads | Service**.
   - Input `230`, `2.0` and `2.0` for the dead Load, X-Moment and Y-Moment, respectively.
   - Input `173`, `108` and `-100` for the live Load, X-Moment and Y-Moment, respectively.
   - Input `6`, `84` and `-70` for the lateral Load, X-Moment and Y-Moment, respectively.
   - Choose Insert to add the entry to the list box.
   - Choose Ok.

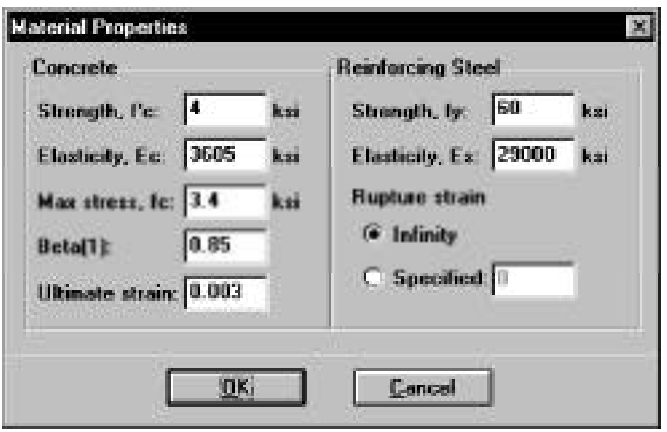
10. From the **Input** menu, choose **Loads | Load Combinations**.
    - If the list displays four combinations, select the third combination and choose Delete.
    - Choose Ok.

11. From the **Solve** menu, choose **Execute**.
    - The solver of the program is started and, upon completion, displays the interaction diagram of the section with the load points plotted within the diagram. Note that point 2 falls just outside the interaction diagram indicating inadequate capacity of this section.

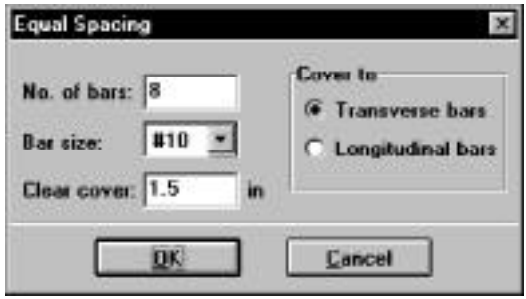
12. From the **View** menu, choose **Results**.
    - Page through the results file.
    - Choose **Exit** to quit the PCAVIEW program and get back to PCACOL.

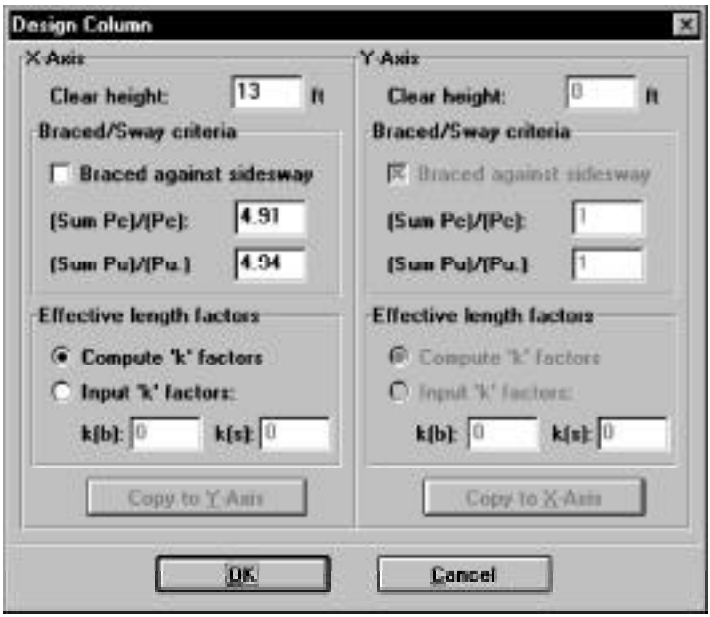
13. From the **File** menu, choose **Print**.
    - Insure that both Screen and Results File are checked. Choose Ok.

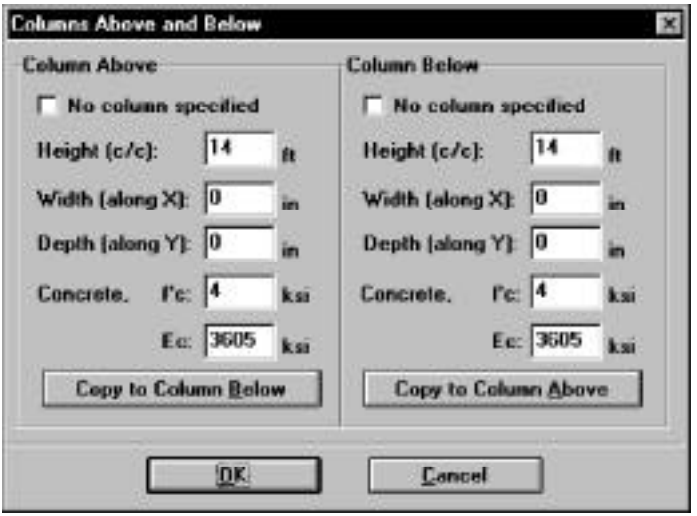

General Information
Labels
Project:
PCACOL Manual, Sway Col
Column:
Ex_3
Engineer:
PCA
Units
English
Metric
Design Code
ACI 318-95
CSA A23.3-94
Run Axis
About X-Axis
About Y-Axis
Biaxial
Run Option
Investigation
Design
Consider slenderness?
Yes
No
OK
Cancel


Material Properties
Concrete
Strength, f'c:
4
ksi
Elasticity, Ec:
3605
ksi
Max stress, fc:
3.4
ksi
Beta[1]:
0.85
Ultimate strain:
0.003
Reinforcing Steel
Strength, fy:
60
ksi
Elasticity, Es:
29000
ksi
Rupture strain
Infinity
Specified:
0
OK
Cancel

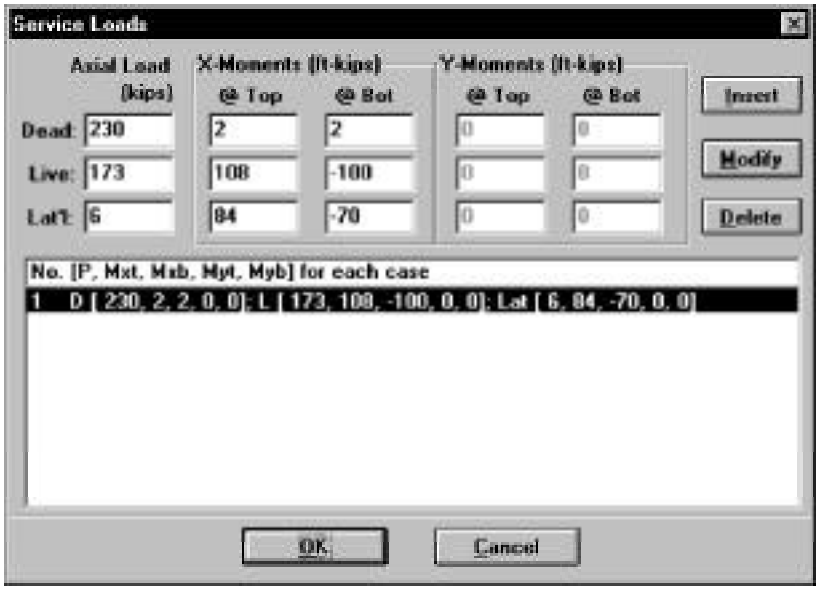
Rectangular Section
Width (along X): 18 in
Depth (along Y): 18 in
OK
Cancel


Equal Spacing
No. of bars: 8
Bar size: #10
Clear cover: 1.5 in
Cover to
Transverse bars
Longitudinal bars
OK
Cancel

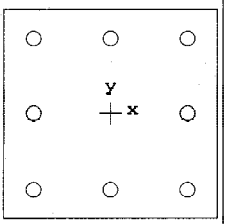

Design Column
X Axis
Clear height: 13 ft
Braced/Sway criteria
Braced against sidesway
(Sum Pc)/(Pc): 4.91
(Sum Pu)/(Pu.) 4.94
Effective length factors
Compute 'k' factors
Input 'k' factors:
k(b): 0 k(s): 0
Copy to Y Axis
Y Axis
Clear height: 0 ft
Braced/Sway criteria
Braced against sidesway
(Sum Pc)/(Pc): 1
(Sum Pu)/(Pu.) 1
Effective length factors
Compute 'k' factors
Input 'k' factors:
k(b): 0 k(s): 0
Copy to X Axis
OK
Cancel


Columns Above and Below
Column Above
No column specified
Height (c/c): 14 ft
Width (along X): 0 in
Depth (along Y): 0 in
Concrete. f'c: 4 ksi
Ec: 3605 ksi
Copy to Column Below
Column Below
No column specified
Height (c/c): 14 ft
Width (along X): 0 in
Depth (along Y): 0 in
Concrete. f'c: 4 ksi
Ec: 3605 ksi
Copy to Column Above
OK
Cancel

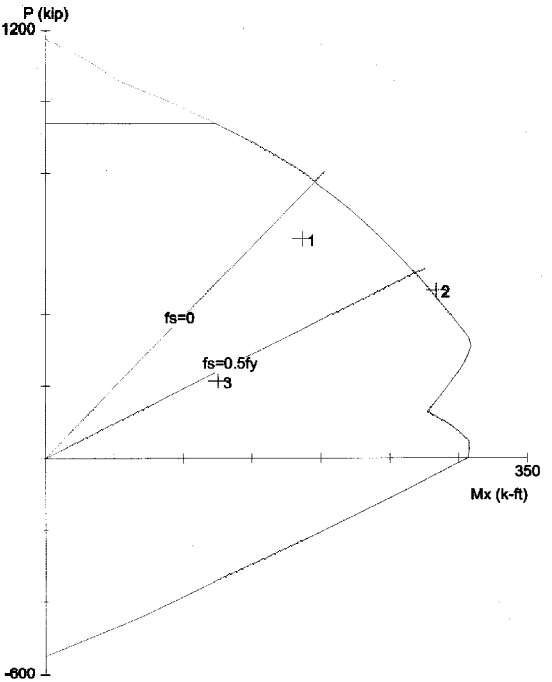
X-Beams (perpendicular to X)
Beam Location:
Above Left
Above Right
Below Left
Below Right
Beam Above Left
No beam specified
Copy From Beam Right
Span (c/c): 24 ft
f'c: 4 ksi
Width: 48 in
Ec: 3605 ksi
Depth: 12 in
Inertia: 13823 in^4
OK
Cancel


Service Loads
Axial Load (kips)
X-Moments (ft-kips)
@ Top
@ Bot
Y-Moments (ft-kips)
@ Top
@ Bot
Insert
Modify
Delete
Dead: 230
2
2
0
0
Live: 173
108
-108
0
0
Lat'l: 6
84
-70
0
0
No. [P, Mxt, Mxb, Myt, Myb] for each case
1 D [230, 2, 2, 0, 0]; L [173, 108, -100, 0, 0]; Lat [6, 84, -70, 0, 0]
OK
Cancel

Load Combinations
0.9
Dead +
0
Live +
1.3
Lat.
Add
Modify
Delete
Combo Dead Live Lateral
U1 1.4 1.7 0
U2 1.05 1.275 1.275
U3 0.9 0 1.3
OK
Cancel



18 x 18 in



Code: ACI 318-95
Units: English
Run axis: About X-axis
Run option: Investigation
Slenderness: Considered
Column type: User-defined
Bars: ASTM English
Date: 10/29/98
Time: 09:41:54

PCACOL V 3.00 (PCA 1998) - Licensed to: Portland Cement Assoc, Skokie, IL

File: D:\DATA\PCACOL\EXAMPLE3.COL
Project: PCACOL Manual, Sway Col

Column: Ex_3		Engineer: PCA	
f'c = 4 ksi	fy = 60 ksi	Ag = 324 in^2	8 #10 bars
Ec = 3605 ksi	Es = 29000 ksi	As = 10.16 in^2	Rho = 3.14%
fc = 3.4 ksi	e_rup = Inflnlty	Xo = 0.00 in	Ix = 8748 in^4
e_u = 0.003 in/in		Yo = 0.00 in	Iy = 8748 in^4
Beta1 = 0.85		Clear spacing = 5.22 in	Clear cover = 1.87 in
Confinement: Tied	phi(a) = 0.8, phi(b) = 0.9, phi(c) = 0.7		

kx(braced) = 0.91698, kx(sway) = 1.60235

```
10/29/98 PCACOL v3.00 - PORTLAND CEMENT ASSOCIATION -                    Page   1
09:40:21 Licensed to: Portland Cement Assoc, Skokie, IL                  EXAMPLE3

  General Information:
  ====================
     File Name: D:\DATA\PCACOL\EXAMPLE3.COL
     Project:  PCACOL Manual, Sway Col
     Column:   Ex_3                          Engineer: PCA
     Code:     ACI 318-95                    Units: English

     Run Option: Investigation               Slenderness: Considered
     Run Axis:   X-axis                      Column Type: Structural

  Material Properties:
  --------------------
     f'c   = 4 ksi                           fy   = 60 ksi
     Ec    = 3605 ksi                        Es   = 29000 ksi
     fc    = 3.4 ksi                         Rupture strain = Infinity
     Ultimate strain = 0.003 in/in
     Beta1 = 0.85

  Section:
  --------
     Rectangular: Width = 18 in              Depth = 18 in

     Gross section area, Ag =  324 in^2
     Ix =  8748 in^4                         Iy =  8748 in^4
     Xo =  0 in                              Yo =  0 in

  Reinforcement:
  ==============
     Rebar Database: ASTM English
     Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)
     ---- --------- -----------   ---- --------- -----------   ---- --------- -----------
     #  3      0.38        0.11   #  4      0.50        0.20   #  5      0.63        0.31
     #  6      0.75        0.44   #  7      0.88        0.60   #  8      1.00        0.79
     #  9      1.13        1.00   # 10      1.27        1.27   # 11      1.41        1.56
     # 14      1.69        2.25   # 18      2.26        4.00

     Confinement: Tied: #3 ties with #10 bars,  #4 with larger bars.
     phi(a) = 0.8,  phi(b) = 0.9,  phi(c) = 0.7

     Layout: Rectangular
     Pattern: Equal Bar Spacing  (Cover to transverse reinforcement)
     Total steel area, As = 10.16 in^2 at 3.14%
     8 #10   Cover = 1.5 in

  Slenderness:
  ============
     Sway Criteria:
     --------------
     X-axis: Unbraced column. Sum of Pc = 4.91*Pc   Sum of Pu = 4.94*Pu

                    Height       Width       Depth           I       f'c          Ec
     Column Axis        ft          in          in         in^4       ksi         ksi
     ------ ---- --------- ---------- ---------- ------------ -------- ------------
     Design  X          13          18          18         8748        4         3605
     Above   X          14           0           0         8748        4         3605
     Below   X          14           0           0         8748        4         3605
```

```
10/29/98 PCACOL V3.00 - PORTLAND CEMENT ASSOCIATION -                    Page   2
09:40:21 Licensed to: Portland Cement Assoc, Skokie, IL                  EXAMPLE3
```

X-Beams Location	Length ft	Width in	Depth in	I in^4	f'c ksi	Ec ksi
Above Left	24	48	12	13823	4	3605
Above Right	24	48	12	13823	4	3605
Below Left	24	48	12	13823	4	3605
Below Right	24	48	12	13823	4	3605

Effective Length Factors:

Axis	Psi(top)	Psi(bot)	k(Braced)	k(Sway)	klu/r
X	2.170	2.170	0.917	1.602	48.11

Moment Magnification Factors:

Stiffness reduction factor, phi(K) = 0.75
Cracked-section coefficients: cI(beams) = 0.35; cI(columns) = 0.7

0.2*Ec*Ig + Es*Ise (X-axis) = 1.56e+007 kip-in^2

X-axis Ld/Comb	Braced Pc(kip)	Betad	Cm	Delta	Sway Pc(kip)	Sway Delta
1 U1	4946	0.523	0.959	1.150	2466	1.504
U2	4468	0.686	0.959	1.115	2466	1.343
U3	3014	1.499	0.400	1.000	2466	1.132

Load Combinations:

U1 = 1.400*Dead + 1.700*Live + 0.000*Lateral
U2 = 1.050*Dead + 1.275*Live + 1.275*Lateral
U3 = 0.900*Dead + 0.000*Live + 1.300*Lateral

Service Loads:

No.	Load Case	Axial Load kip	Mx @ Top k-ft	Mx @ Bot k-ft	My @ Top k-ft	My @ Bot k-ft
1	Dead	230.0	2.0	2.0	0.0	0.0
	Live	173.0	108.0	-100.0	0.0	0.0
	Latl	6.0	84.0	-70.0	0.0	0.0

Factored Loads and Moments with Corresponding Capacities:

No.	Load Combo	Pu kip	Mux k-ft	fMnx k-ft	fMn/Mu
1	1 U1	616.1	186.4	244.2	1.310
2	U2	469.7	283.7	279.2	0.984
3	U3	214.8	125.4	295.0	2.352

*** Program completed as requested! ***

Chapter 6

The IRRCOL Module

IRRCOL is an add-on module to the PCACOL program. It enables the user to investigate irregularly shaped reinforced concrete sections subject to a combination of loadings. IRRCOL considers a polygonal section reinforced with bars located at any location. The section may contain one opening. In addition, IRRCOL allows easy modification of a circular section by converting it into an approximate polygon.

IRRCOL is accessed from within PCACOL. Once PCACOL is started, you can activate IRRCOL by selecting the **Section | Irregular** command from the **Input** menu. This starts IRRCOL, minimizes and locks the PCACOL window and maximizes the IRRCOL window. The section geometry may then be defined by drawing it on the screen or by inputting the coordinates of the polygon nodes. Once the section definition is complete, return to PCACOL by selecting **Save and Exit** from the **Main** menu.

Once the irregular section is defined, its reinforcement is input using IRRCOL as well. This is done by selecting the **Reinforcement | Irregular** command from the **Input** menu of PCACOL. The section reinforcement may also be defined graphically.

Main Menu

Drawing Area
Set the limits for the drawing area and control the grid and snap settings.

Save and Exit
Accept the input and/or modifications made in IRRCOL and return to PCACOL.

Exit without Saving
Discard the input and/or modifications made in IRRCOL and return to PCACOL.

Edit Menu

Section
Toggles the input/editing session to the section.

Opening
Toggles the input/editing session to the section opening.

Reinforcement
Toggles the input/editing session to the section reinforcement.

Add
Switches to the Add mode to graphically insert section or opening nodes or reinforcing bars.

Delete
Switches to the Delete mode to graphically delete section or opening nodes or reinforcing bars.

Move
Switches to the Move mode to graphically move section or opening nodes or reinforcing bars.

Modify
Allows manual editing of the section or opening node coordinates or the reinforcing bar area and coordinates.

View Menu

Zoom
Magnifies or reduces the graphical image.

Pan
Translates or moves the graphical image.

Redraw
Redraws the displayed section and resets the Zoom and Pan actions.

Grid
Shows or hides the grid.

Snap
Toggles the Snap On and Off. With the Snap On, the cursor snaps (sticks) to exact locations.

Labels
Shows or hides the internal numbering of section (or opening) nodes and bars.

Status bar
Shows or hides the Status bar.

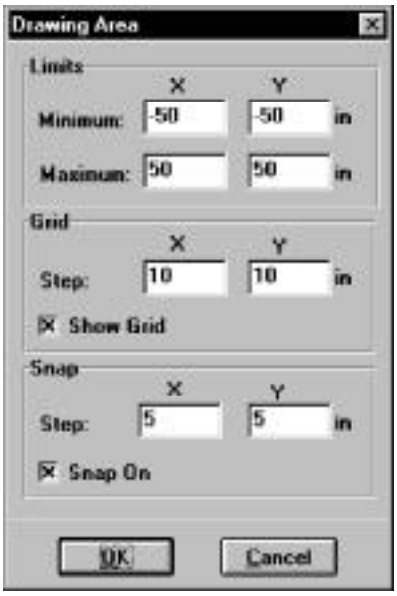
Help Menu

Index
Lists all the available help topics about the program.

About
Displays copyright information and the version number of the program.

Defining the Drawing Area

- Select **Main | Drawing Area**. A dialog box appears. This step is not needed the first time IRRCOL is started using PCACOL's **Section | Irregular** command. The box is automatically displayed.



- Input the limits (minimum and maximum) of the area within which the section is to be drawn in both directions.
- Input the interval (step) for the grid system. The grid is used as a guideline when drawing.
- Select or clear the Show Grid option to show or hide the grid.
- Input the interval (step) for the snap. The snap settings may be different than the grid settings and may be different in each direction (x and y).
- Select or clear the Snap On option to activate or disable the Snap.
- Choose the OK button.

Graphical Editing of the Section or Opening

In IRRCOL, the section (or opening) must be a closed polygon. It is defined by inputting the nodes at each corner or vertex of the polygon. No two segments (sides) may overlap or cross and no two nodes may coincide. An opening must lie completely within the section.
The coordinates that correspond to the cursor location are always displayed in the Status bar. To force the cursor to "stick" to exact locations, turn the Snap option On. The program will snap to locations whose coordinates are multiples of the snap steps.

To add section or opening nodes:

- Switch the input session to Section (or Opening) by selecting **Edit | Section** (or **Edit | Opening**).
- Switch the input mode to Add by selecting **Edit | Add**.
- If you are editing a section (or opening), you must first click on an existing node. Once this is done, you can insert subsequent nodes.
- Adding a node is done by moving the cursor to a desired location within the Graphics area and clicking the left-mouse button.
- After inserting a node, a line extending from the last added node follows the cursor. Click at another location to insert subsequent nodes. A line connecting the two nodes is drawn.
- To end the drawing, click the right-mouse button or press the ESC key. The first and last nodes are connected to close the section. Also note that while in the Add mode, moving the cursor outside the Graphics area ends the drawing.

To delete section or opening nodes:

- Switch the input session to Section (or Opening) by selecting **Edit | Section** (or **Edit | Opening**).
- Switch the input mode to Delete by selecting **Edit | Delete**.
- Marquee select the node (or nodes) to be deleted. This is done by moving the cursor to one location, pressing the left-mouse button and while holding the button down, moving to another location. While moving, a dotted rectangle (called the marquee) is drawn. Releasing the left-mouse button end the marquee. All nodes that are enclosed with the marquee are deleted and the section (or opening) is redrawn.To cancel a marquee action while holding down the left-mouse button, press the ESC key or click the right-mouse button. Also, leaving the Graphics area cancels the marquee operation.

To move section or opening nodes:

- Switch the input session to Section (or Opening) by selecting **Edit | Section** (or **Edit | Opening**).
- Switch the input mode to Move by selecting **Edit | Move**.
- Place the cursor over the node to be moved then press and hold the left-mouse button. As long as the button is held down, moving the cursor moves the selected node to the cursor location. Once the node is at the new desired location, release the mouse button. To cancel a move action while holding down the left-mouse button, press the ESC key or click the right-mouse button. Also, leaving the Graphics area cancels the move operation.

Graphical Editing of the Reinforcement

In IRRCOL, the reinforcing bars may be located anywhere. The only restriction imposed in the program is that no two bars may overlap. The coordinates that correspond to the cursor location are always displayed in the Status bar. To force the cursor to "stick" to exact locations, turn the Snap option On. The program will snap to locations whose coordinates are multiples of the snap steps. While in the Reinforcement session, you cannot switch to the Section or Opening sessions without returning to PCACOL. Therefore, the **Edit | Reinforcement** command will have a check mark to its left. You use the command to specify the bar size or area and the number of bars to be added.

To add one single bar at a time:

- Switch the input mode to Add by selecting **Edit | Add**.
- Select **Reinforcement** from the **Edit** menu. A dialog box appears.

- In the Number of bars box, input 1.
- Choose the Size option and select the size of the bar or choose the Area option and input the bar area in the edit box.
- Choose the OK button.
- In the Graphics area, point to the desired location and click. A bar is inserted.
- Repeat the step above to insert more bars.

To generate multiple bars:

- Switch the input mode to Add by selecting **Edit | Add**. The command must have a check mark left of it.
- Select **Reinforcement** from the **Edit** menu. A dialog box appears.

- Input the number of bars to be generated (should be greater than 1).
- Choose the Size option and select the size of the bar or choose the Area option and input the bar area in the edit box.
- Choose the OK button.
- In the Graphics area, click at a location (let us call it point A), move the cursor to another location and click (let us call it Point B). The specified number of equally-spaced bars are added, along the line connecting the points A and B. For subsequent generations, point B becomes A and you only need to click at the second point (B) to generate more bars.
- To end the multiple bar generation or move to another location (without adding bars), click the right-mouse button or press the ESC key. Also note that while in the multiple bar generation of the Add mode, moving the cursor outside the Graphics area aborts the generation.

To delete bars:

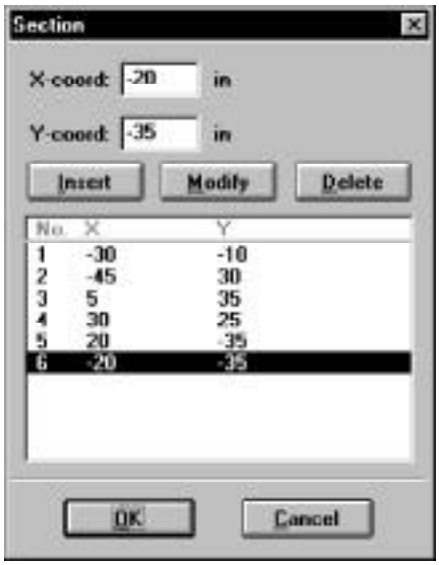
- Switch the input mode to Delete by selecting **Edit | Delete**.
- Marquee select the bar(s) to be deleted. This is done by moving the cursor to one location, pressing the left-mouse button and while holding the button down, moving to another location. While moving, a dotted rectangle (called the marquee) is drawn. Releasing the left-mouse button end the marquee. All bars that are enclosed with the marquee are deleted. To cancel a marquee action while holding down the left-mouse button, press the ESC key or click the right-mouse button. Also, leaving the Graphics area cancels the marquee operation.

To move a bar:

- Switch the input mode to Move by selecting **Edit | Move**.
- Place the cursor over the bar to be moved then press and hold the left-mouse button. As long as the button is held down, moving the cursor moves the selected bar with it. Once the bar is at the new desired location, release the mouse button. To cancel a move action while holding down the left-mouse button, press the ESC key or click the right-mouse button. Also, leaving the Graphics area cancels the move operation.

Manual Editing of the Section or Opening

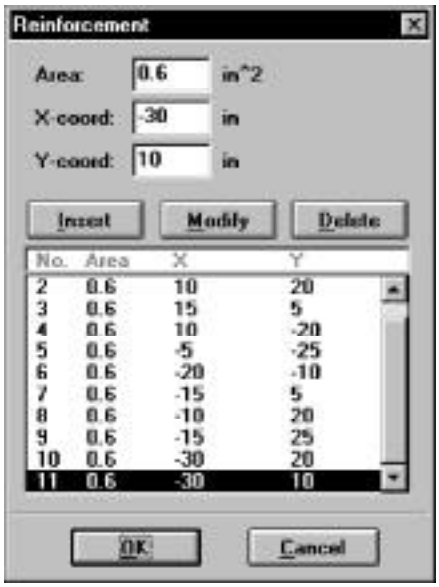
- Switch the input session to Section (or Opening) by selecting **Edit | Section** (or **Edit | Opening**).
- Select **Edit | Modify**. A dialog box appears. If you have already defined a section (or opening), the coordinates of its nodes are shown in the list.



- To add a node, enter its x and y coordinates and choose the Insert button. The entry is inserted in the list box following the selected entry.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button.
- To delete an entry, choose it from the list box and choose the Delete button.
- Choose the OK button.

Manual Editing of the Reinforcement

- Select **Edit | Modify**. A dialog box appears.



- To add a bar, enter its area, x-coordinate and y-coordinate and choose the Insert button. The entry is inserted in the list box following the selected entry.
- To edit an entry, select it from the list box, modify its values in the text boxes, and choose the Modify button. You can change the area of a group of bars by selecting them from the list, enter the new area and choose Modify.
- To delete an entry, choose it from the list box and choose the Delete button.
- Choose the OK button.

Zooming and Panning the Image

Zooming is the enlargement or reduction of the graphical image. Panning is the translation or movement of the image.

Zooming:

- Select **View | Zoom**.
- To enlarge the view (zoom-in): using the left-mouse button, click anywhere within the Graphics area. Click again for further magnification.
- To reduce the view (zoom-out): using the right-mouse button, click anywhere within the Graphics area. Click again for further reduction.
- To reset the Zoom and return to the default, select **View | Redraw**.

Panning:

- Select **View | Pan**.
- Place the cursor over the Graphics area. Press the left-mouse button and, while still holding the button, move the cursor in the direction you wish to pan. A line showing the amount and direction of panning is drawn.
- Release the mouse button. The view is panned and the image is redrawn.
- To reset the Pan and return to the default, select **View | Redraw**.

Keyboard shortcuts:

The following keyboard assignments are always active:

Key combination	Action
Ctrl + Page Up	Zoom-in
Ctrl + Page Down	Zoom-out
Ctrl + Cursor key	Pan
Enter	Reset zoom/pan (Redraw)

Example 4 - Capacity of an Irregular Section

This example uses the IRRCOL add-on module.

Obtain the interaction diagram of a T-shaped section which contains a rectangular opening. The capacity is to be computed about the x-axis. The section is reinforced with 19-#8 bars, spaced 6 in. apart. The distance between the center of each edge bar to the face of the section is 3 in. Use 6-ksi concrete and 60-ksi reinforcing steel.

Note: when instructed to move to a location (15, 25), move the cursor until the cells in the Status bar show "x=15" and "y=25".

1. From the **File** menu, choose **New**. Any input data is cleared and the default values are restored.

2. From the **Input** menu, choose **General Information**.
   - Select English units and ACI 319-95 code.
   - Select About X for run axis, Investigation for run option and No for slenderness.
   - Choose Ok.

3. From the **Input** menu, choose **Material Properties**.
   - Input 6 for the concrete strength. Other properties are computed and will be accepted.
   - Choose Ok.

4. From the **Input** menu, choose **Section | Irregular**. The IRRCOL module is loaded, its window occupies the desktop, and the Drawing Area box appears.
   - Input −3 and −3 for the minimum X and Y, respectively.
   - Input 45 and 45 for the maximum X and Y, respectively.

- Input 3 and 3 for X and Y grid step, respectively and check the Show Grid option.
- Input 3 and 3 for X and Y snap step, respectively and check the Snap On option.
- Choose Ok.

5. The default input session is Section and the default input mode is Add (the **Edit | Section** and **Edit | Add** options are checked).
   - In the Graphics area, move the cursor to the location (6, 0) and click the left mouse button. A node is inserted.
   - Move to (24, 0) and click the left mouse button.
   - Move to (24, 24) and click the left mouse button.
   - Move to (30, 24) and click the left mouse button.
   - Move to (30, 36) and click the left mouse button.
   - Move to (0, 36) and click the left mouse button.
   - Move to (0, 24) and click the left mouse button.
   - Move to (6, 24) and click the left mouse button.
   - Click the right-mouse button to end drawing the section.
6. To define the opening nodes, switch the input session to Opening by selecting **Edit | Opening**. The input mode should still be Add (the **Edit | Opening** and **Edit | Add** options are now checked).
   - In the Graphics area, move the cursor to the location (12, 6) and click the left mouse button. A node is inserted.
   - Move to (18, 6) and click the left mouse button.
   - Move to (18, 24) and click the left mouse button.
   - Move to (12, 24) and click the left mouse button.
   - Click the right-mouse button to end drawing the opening.

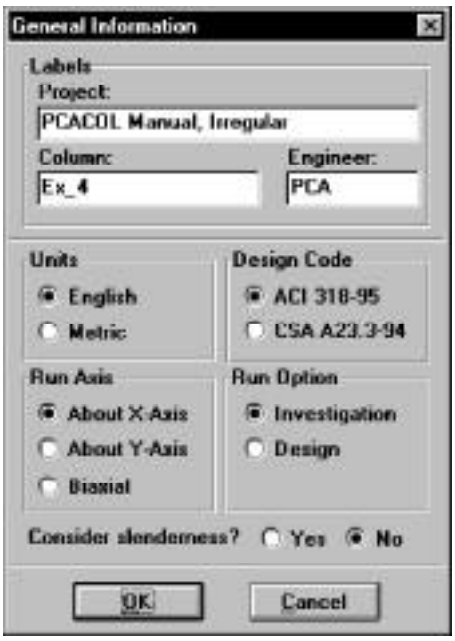
7. From the **Main** menu, choose **Save and Exit**. You are returned to PCACOL and the section is shown in the Information bar.

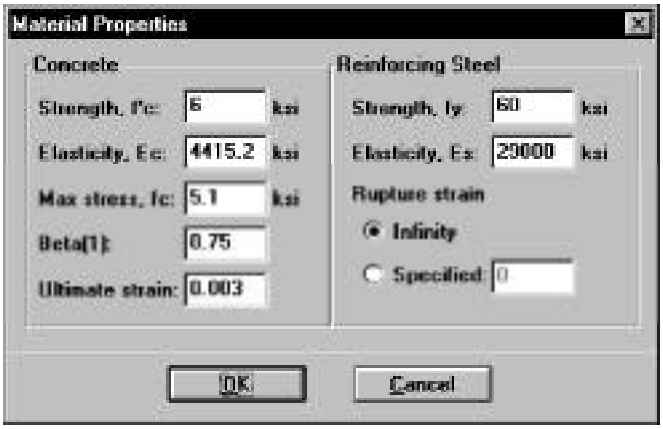
8. From the **Input** menu, choose **Reinforcement | Irregular**. The IRRCOL module is loaded, its window occupies the desktop.

9. The default input session is Reinforcement and the default input mode is Add (the **Edit** | **Reinforcement** and **Edit** | **Add** options are checked).

10. Define some bars using multiple bar generation:
    - From the **Edit** menu, choose **Reinforcement**. Input 5 for the Number of bars and select #8 from the Size list. Choose Ok.
    - Move the cursor to the location (3, 33) and click the left mouse button.
    - Move to (27, 33) and click the left mouse button. Five equally-spaced bars are generated between the two locations.
    - Click the right-mouse button to end.
    - Move the cursor to the location (9, 27) and click the left mouse button.
    - Move to (9, 3) and click the left mouse button. Five equally-spaced bars are generated between the two locations.
    - Click the right-mouse button to end.
    - Move the cursor to the location (21, 27) and click the left mouse button.
    - Move to (21, 3) and click the left mouse button. Five equally-spaced bars are generated between the two locations.
    - Click the right-mouse button to end.

11. Define the remaining bars using single bar placement:
    - From the **Edit** menu, choose **Reinforcement**. Input `1` for the Number of bars and select #8 from the Size list. Choose Ok.
    - Move the cursor to the location (3, 27) and click the left mouse button. A bar is inserted.
    - Move to (15, 27) and click the left mouse button. Another bar is inserted.
    - Move to (27, 27) and click the left mouse button. Another bar is inserted.
    - Move to (15, 3) and click the left mouse button. Another bar is inserted.
12. From the **Main** menu, choose **Save and Exit**. You are returned to PCACOL and the section is shown in the Information bar.
13. From the **Input** menu, choose **Loads | Control Points**.
14. From the **Solve** menu, choose **Execute**.
    - The solver of the program is started and, upon completion, displays the interaction diagram of the section.
15. From the **View** menu, choose **Results**.
    - Page through the results file.
    - Choose **Exit** to quit the PCAVIEW program and get back to PCACOL.
16. From the **File** menu, choose **Print**
    - Insure that both Screen and Results File are checked. Choose Ok.

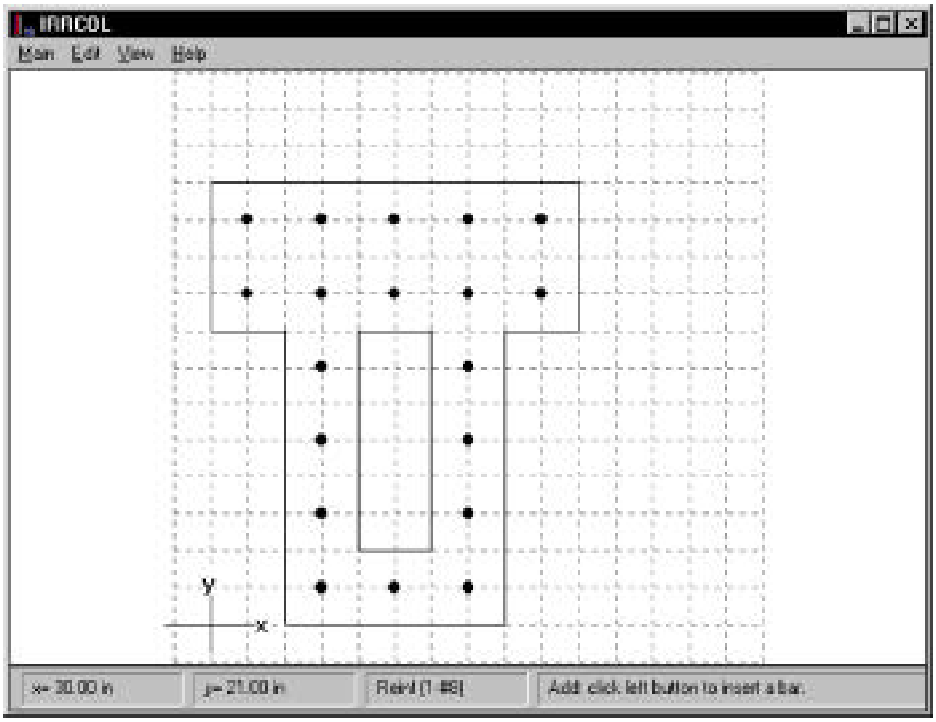

General Information
Labels
Project:
PCACOL Manual, Irregular
Column:
Ex_4
Engineer:
PCA
Units
English
Metric
Design Code
ACI 318-95
CSA A23.3-94
Run Axis
About X-Axis
About Y-Axis
Biaxial
Run Option
Investigation
Design
Consider slenderness? Yes No
OK
Cancel

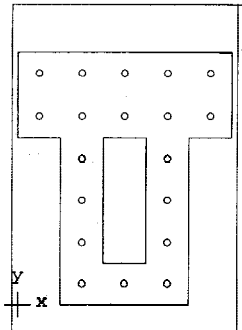

Material Properties
Concrete
Strength, fc: 6 ksi
Elasticity, Ec: 4415.2 ksi
Max stress, fc: 5.1 ksi
Beta(1): 0.75
Ultimate strain: 0.003
Reinforcing Steel
Strength, fy: 60 ksi
Elasticity, Es: 29000 ksi
Rupture strain
Infinity
Specified: 0
OK
Cancel

Drawing Area
Limits
X
Y
Minimum:
-3
-3
in
Maximum:
45
45
in
Grid
X
Y
Step:
3
3
in
Show Grid
Snap
X
Y
Step:
3
3
in
Snap On
OK
Cancel

Reinforcement
Number of bars:
5
Bar size:
#8
Bar area:
0.79
in
OK
Cancel

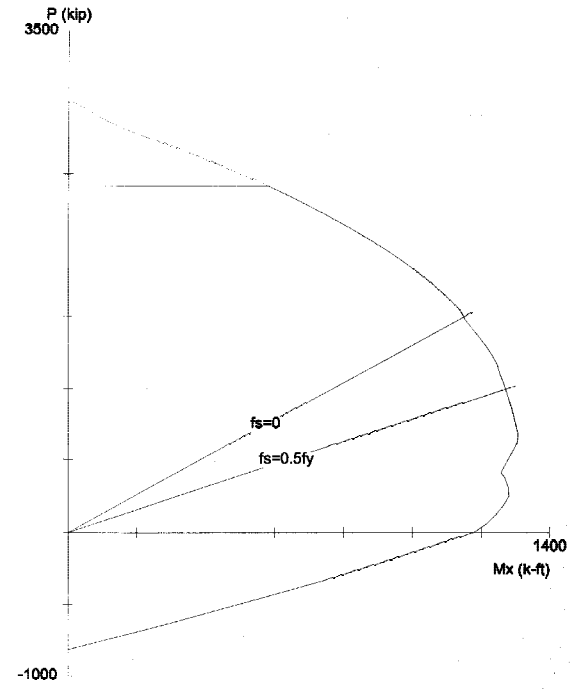
Reinforcement
Number of bars:
1
Bar size:
#8
Bar area:
0.79
in
OK
Cancel


IRRCOL
Main Edit View Help
y
x
x= 30.00 in
y= 21.00 in
Point [1-85]
Add: click left button to insert a bar.



30 x 36 in

Code: ACI 318-95
Units: English
Run axis: About X-axis
Run option: Investigation
Slenderness: Not considered
Column type: Structural
Bars: ASTM English
Date: 10/29/98
Time: 09:42:14



PCACOL V 3.00 (PCA 1998) - Licensed to: Portland Cement Assoc, Skokie, IL

File: D:\DATA\PCACOL\EXAMPLE4.COL
Project: PCACOL Manual, Irregular

Column: Ex_4		Engineer: MIK	
f'c = 6 ksi	fy = 60 ksi	Ag = 684 in^2	19 #8 bars
Ec = 4415 ksi	Es = 29000 ksi	As = 15.01 in^2	Rho = 2.19%
fc = 5.1 ksi	e_rup = Infinity	Xo = 15.00 in	Ix = 82404 in^4
e_u = 0.003 in/in		Yo = 21.00 in	Iy = 38340 in^4
Beta1 = 0.75		Clear spacing = 5.00 in	Clear cover = N/A
Confinement: Tied	phi(a) = 0.8, phi(b) = 0.9, phi(c) = 0.7		

```
10/29/98 PCACOL V3.00 - PORTLAND CEMENT ASSOCIATION -                    Page   1
09:42:10 Licensed to: Portland Cement Assoc, Skokie, IL                  EXAMPLE4

  General Information:
  ====================
     File Name: D:\DATA\PCACOL\EXAMPLE4.COL
     Project:  PCACOL Manual, Irregular
     Column:   Ex_4                          Engineer: MIK
     Code:     ACI 318-95                    Units: English

     Run Option: Investigation               Slenderness: Not considered
     Run Axis:   X-axis                      Column Type: Structural

  Material Properties:
  ====================
     f'c   = 6 ksi                           fy   = 60 ksi
     Ec    = 4415.21 ksi                     Es   = 29000 ksi
     fc    = 5.1 ksi                         Rupture strain = Infinity
     Ultimate strain = 0.003 in/in
     Betal = 0.75

  Section:
  ========
     Exterior Points
       No.     X (in)     Y (in)     No.     X (in)     Y (in)     No.     X (in)     Y (in)
     ----- ---------- ---------- ----- ---------- ---------- ----- ---------- ----------
         1        6.0        0.0     2       24.0        0.0     3       24.0       24.0
         4       30.0       24.0     5       30.0       36.0     6        0.0       36.0
         7        0.0       24.0     8        6.0       24.0
     Interior Points
       No.     X (in)     Y (in)     No.     X (in)     Y (in)     No.     X (in)     Y (in)
     ----- ---------- ---------- ----- ---------- ---------- ----- ---------- ----------
         1       12.0        6.0     2       18.0        6.0     3       18.0       24.0
         4       12.0       24.0

     Gross section area, Ag =  684 in^2
     Ix =  82404 in^4                        Iy =  38340 in^4
     Xo =  15 in                             Yo =  21 in

  Reinforcement:
  ==============
     Rebar Database: ASTM English
     Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)   Size Diam (in) Area (in^2)
     ---- --------- -----------   ---- --------- -----------   ---- --------- -----------
     #  3      0.38        0.11   #  4      0.50        0.20   #  5      0.63        0.31
     #  6      0.75        0.44   #  7      0.88        0.60   #  8      1.00        0.79
     #  9      1.13        1.00   # 10      1.27        1.27   # 11      1.41        1.56
     # 14      1.69        2.25   # 18      2.26        4.00

     Confinement: Tied; #3 ties with #10 bars,  #4 with larger bars.
     phi(a) = 0.8,  phi(b) = 0.9,  phi(c) = 0.7

     Pattern: Irregular

     Total steel area, As = 15.01 in^2 at 2.19%

     Area in^2   X (in)   Y (in)  Area in^2   X (in)   Y (in)  Area in^2   X (in)   Y (in)
     --------- -------- --------  --------- -------- --------  --------- -------- --------
          0.79      3.0     33.0       0.79      9.0     33.0       0.79     15.0     33.0
          0.79     21.0     33.0       0.79     27.0     33.0       0.79      9.0     27.0
          0.79      9.0     21.0       0.79      9.0     15.0       0.79      9.0      9.0
          0.79      9.0      3.0       0.79     21.0     27.0       0.79     21.0     21.0
          0.79     21.0     15.0       0.79     21.0      9.0       0.79     21.0      3.0
          0.79      3.0     27.0       0.79     15.0     27.0       0.79     27.0     27.0
          0.79     15.0      3.0
```

10/29/98 PCACOL V3.60 - PORTLAND CEMENT ASSOCIATION - Page 2
09:42:10 Licensed to: Portland Cement Assoc, Skokie, IL. EXAMPLE4

Control Points:

Bending about	Axial Load P kip	X-Moment k-ft	Y-Moment k-ft	N.A. depth in
X @ Pure compression	3018.7	0	0	106.33
@ Max compression	2413.0	581	-0	43.06
@ fs = 0.0	1094.6	1147	0	30.00
@ fs = 0.5*fy	998.2	1171	0	24.54
@ Balanced point	644.9	1311	0	19.53
@ Pure bending	0.0	1178	-0	8.18
@ Pure tension	-810.5	0	0	0.00

*** Program completed as requested! ***