

CONCRETE DESIGN SOFTWARE LIBRARY



**DESIGN AND INVESTIGATION OF
REINFORCED CONCRETE CONTINUOUS BEAMS**



PORTLAND CEMENT
ASSOCIATION

5420 Old Orchard Road, Skokie, Illinois 60077-1083
Telephone: (847) 966-4357 (HELP)

605 013 1100

Acknowledgments

This computer program in its entirety (including software design, programming structure, graphics, manual, and on-line help system) was created by the Computer Programs Marketing Department, 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 solutions and design 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.

© 1992 Portland Cement Association. All Rights Reserved.

PCABEAM is a Trademark of the Portland Cement Association

MS-DOS is a trademark of Microsoft, Inc.

Microsoft and Windows are registered trademarks of the Microsoft Corporation.

Appendix A

Conversion Factors—U.S. to SI

To convert from	To	Multiply by
in.	m (0.001 mm)	0.002 540
ft	m	0.304 800
lb	N (0.001 kN)	4.448 222
kips (1000 lbs)	kN	4.448 222
plf (lb/ft)	kN/m	0.014 593
psi (lb/in. ²)	MPa	0.006 895
ksi (kips/in. ²)	MPa	6.894 757
psf (lb/ft ²)	kN/m ²	0.047 880
pcf (lb/ft ³)	kg/m ³	16.018 460
ft-kips	kN-m	1.355 818
kg	N	9.806 650
kips/in.	kN/mm	0.175284

U.S. Bar Sizes

U.S. Bar No.	Diameter (in.)	Cross Sectional Area (in.²)	Closest Metric Bar No.
3	0.375	0.11	N-10
4	0.500	0.20	N-10
5	0.625	0.31	N-15
6	0.750	0.44	N-20
7	0.875	0.60	N-20
8	1.000	0.79	N-25
9	1.128	1.00	N-30
10	1.270	1.27	N-35
11	1.410	1.56	N-35
14	1.693	2.25	N-45
18	2.257	4.00	N-55

SI Bar Sizes

Metric No.	Diameter (mm)	Cross Sectional Area (mm²)	Closest U.S. Bar No.
N-10	11.3	100	4
N-15	16.0	200	5
N-20	19.5	300	6
N-25	25.2	500	8
N-30	29.9	700	9
N-35	35.7	1000	11
N-45	43.7	1500	14
N-55	56.4	2500	18

Table of Contents

Introduction

Program Features	i-1
Terms	i-1
Conventions	i-1
System Requirements	i-2

Chapter 1—Installing the Program

Installing PCABEAM	1-1
Running PCABEAM	1-2

Chapter 2—Method of Solution

The System	2-1
Material Properties	2-3
Geometric Checks	2-5
Code Checks	2-6
Effective Width	2-6
Minimum Thickness	2-6
Loads	2-8
Self Weight	2-8
Superimposed Loads	2-8
Lateral Load Moments	2-8
Load Combinations and Factors	2-8
Loading Patterns	2-10
Design Assumptions	2-12
Analysis	2-15
Moment Redistribution	2-15
Deflections	2-17

<i>Design</i>	2-19
<i>Flexure</i>	2-19
<i>Shear</i>	2-21
<i>Investigation</i>	2-24
<i>Flexure</i>	2-24
<i>Shear</i>	2-24
<i>Reinforcement</i>	2-25
<i>Design Parameters</i>	2-25
<i>Discontinuous Bars</i>	2-26
<i>Continuous Bars</i>	2-26
<i>Stirrups</i>	2-28
<i>Bar Lengths and Splices</i>	2-29
<i>Results</i>	2-31
<i>Analysis results</i>	2-31
<i>Design Results</i>	2-32
<i>Investigation Results</i>	2-33
<i>Deflections</i>	2-34
<i>Material Quantity Estimate</i>	2-34

Chapter 3—Menu Descriptions

<i>The PCABEAM Window</i>	3-1
<i>Working With Menus</i>	3-4
<i>Menu Conventions</i>	3-4
<i>Working With a Dialog Box</i>	3-5
<i>Working With a List Box</i>	3-7
<i>Working With View Windows</i>	3-9
<i>PCABEAM Menu Descriptions</i>	3-11
<i>The File Menu</i>	3-11
<i>The Input Menu</i>	3-13
<i>The Design Menu</i>	3-16
<i>The View Menu</i>	3-17
<i>The Options Menu</i>	3-19
<i>The Window Menu</i>	3-20
<i>The Help Menu</i>	3-21

Chapter 4—Operating the Program

<i>Working With the File Menu Commands</i>	4-1
<i>Creating a New Data File</i>	4-1
<i>Opening an Existing Data File</i>	4-2
<i>Saving the Data</i>	4-3
<i>Printing the Results</i>	4-5
<i>Printing the Screen</i>	4-6
<i>Working With the Input Menu Commands</i>	4-7
<i>Auto Input</i>	4-7
<i>Inputting Labels</i>	4-8
<i>Using the General Information Command</i>	4-9
<i>Inputting Material Properties</i>	4-11
<i>Inputting Beams and Slabs Geometry</i>	4-13
<i>Inputting the Columns Geometry</i>	4-15
<i>Inputting the Reinforcement Design Parameters</i>	4-17
<i>Inputting Top Continuous Bars</i>	4-19
<i>Inputting Top Discontinuous Bars</i>	4-21
<i>Inputting Bottom Continuous Bars</i>	4-23
<i>Inputting Bottom Discontinuous Bars</i>	4-25
<i>Inputting Beam Stirrups</i>	4-27
<i>Inputting Load Factors and Loading Patterns</i>	4-29
<i>Inputting Surface Loads</i>	4-31
<i>Inputting Partial Dead Loads</i>	4-32
<i>Inputting Partial Live Loads</i>	4-34
<i>Inputting Lateral Load Moments</i>	4-36
<i>Inputting Moment Redistribution Factors</i>	4-38
<i>Working With the Design Menu Commands</i>	4-39
<i>Selecting Output Options</i>	4-39
<i>Executing the Solver</i>	4-40
<i>Designing Columns</i>	4-41
<i>Working With the View Menu Commands</i>	4-42
<i>Defining the Zoom Command</i>	4-42
<i>Setting the View Preferences</i>	4-43
<i>Viewing the Results</i>	4-45
<i>Viewing Shear & Moment Diagrams</i>	4-46

<i>Viewing Shear & Moment Envelope and Capacity Diagrams</i>	4-47
<i>Viewing Deflection Diagrams</i>	4-48
<i>Working With the Options Menu Commands</i>	4-49
<i>Changing the View Color</i>	4-49
<i>Changing the Default Units and Code</i>	4-51
<i>Changing the Default Data Directory</i>	4-52

Chapter 5—Examples

<i>Example 1</i>	5-1
<i>Example 2</i>	5-28

<i>Appendix A</i>	A-1
<i>Conversion Factors—U.S. to SI</i>	A-1
<i>U.S. Bar Sizes</i>	A-2
<i>SI Bar Sizes</i>	A-3

Introduction

Program Features

PCABEAM is a program that was developed to give engineers the capacity to *design* or *investigate* reinforced concrete continuous beam structures.

Terms

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

DOS refers to the personal computer operating system from either IBM PC-DOS or Microsoft MS-DOS version 3.3 or higher.

WINDOWS refers to the Microsoft Windows environment version 3.0 or higher.

[] indicates equivalent Canadian Code requirements.

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.

Marquee select means to depress the mouse button and continue to hold it while moving the mouse. As you drag the mouse, a rectangle (known as a marquee) follows the cursor. Release the mouse button and the area inside the marquee is selected.

Conventions

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

Style	Meaning
<i>Italic</i>	Indicates a glossary item, or emphasizes a given word.

SMALL CAPS	Indicates the name of an object such as a dialog box or a dialog-box component. For example, the OPEN dialog box or the CANCEL button.
ALL CAPS	Indicates the name of a path, a directory, or a file. For example, C:\PCABEAM\PCABEAM1.EXE.
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, type "c:*.*.bms".
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 PCABEAM, a personal computer with the following hardware and software configuration is needed:

- MS-DOS operating system version 3.3 or higher.
- Microsoft Windows version 3.x (running in Standard or Enhanced mode).

Minimum requirements:

- 2.0MB of RAM memory.
- One floppy disk drive.
- A hard disk with at least 2.0MB of free space.
- Any monitor supported by Windows V3.x.

Recommended options:

- A printer supported by Windows V3.x.
- A mouse or other pointing device supported by Windows V3.x.
- A math co-processor.

Program Capacity

- 14 spans (13 supports)
- 100 partial dead loads
- 100 partial live loads
- 6 top bar layers (Investigation option)
- 6 bottom bar layers (Investigation option)
- 3 top bar layers (Design option)
- 3 bottom bar layers (Design option)
- 5 continuous bar sets per layer per span (top or bottom)
- 5 discontinuous bar sets per layer per span (top or bottom)

Chapter 1

Installing the Program

Installing PCABEAM

1. Execute Windows and insert the PCABEAM diskette into the proper floppy disk drive.
2. From the Program Manager **File** menu, select **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. The setup program prompts you for the directory where PCABEAM is to be installed. The default directory is `C:\PCABEAM`. To specify an alternate directory, type in the directory complete with a drive letter. If the directory does not exist, the setup program creates it. Press **ENTER** or choose **INSTALL** to proceed.
5. The setup program then initiates a search for a previous version of PCABEAM on the same drive where PCABEAM is to be installed. If an older version of PCABEAM is found, the program asks if you want to overwrite it. Select **YES** or **NO** accordingly.
6. The **DEFAULT OPTIONS** dialog box appears next. This allows you to customize PCABEAM by setting the units of measure (**US** or **SI**) and the code (**ACI** or **CSA**). These defaults are saved in the `PCAPROGS.INI` file, located in the **WINDOWS** directory.
7. Next, you are prompted for the default data directory. This is where PCABEAM looks for data when you elect to open a data file. This information is saved in the `PCAPROGS.INI` file, located in the **WINDOWS** directory.
8. The **CUSTOMER INFORMATION** dialog box appears next asking for the firm name, city, and state. Be sure to press **TAB** to move to the **CITY** and **STATE** text boxes. Press

ENTER or choose OK when done. A verification dialog box appears. You can accept the information or choose CHANGE to edit it.

9. The setup program begins to copy files from the installation drive to the destination drive. Once the file transfer is done, setup adds the "PCABEAM" icon to the Program Manager. A "PCA Programs" group box is created (if one does not exist), and the "PCABEAM" icon is added.
10. After the setup program displays a successful installation message, the last dialog box displayed asks whether you would like to view the PCABEAM README file. This file contains information supplemental to the manual.

Running PCABEAM

Once PCABEAM is installed, it is ready to run.

To execute PCABEAM from within Windows:

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

To execute PCABEAM from the DOS prompt:

- Change to the directory where the program is installed, type `win pcabeam` and press ENTER. This will execute Windows and load the PCABEAM program as the application program.

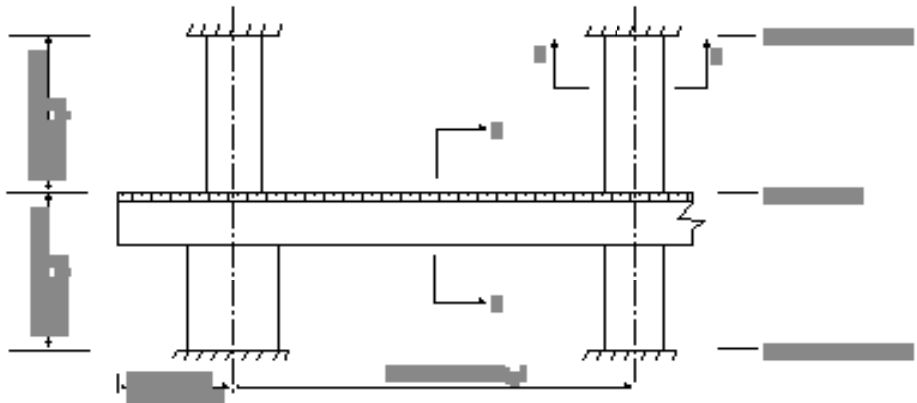
Chapter 2

Method of Solution

The System

The continuous beam (or one-way slab) system modeled using PCABEAM, may consist of up to 14 spans including two cantilevers, one at each end. The first span is always the left cantilever and the last span is the right cantilever. Span length is measured from centerline of support to centerline of support (or from edge to support centerline for cantilevers). Span length, span width, and beam cross section, may vary from span to span. Besides, the slab may have unequal widths on each side of the column centerline (in the transverse direction).

Supports are assumed to be columns. Columns specified above and below the design floor may have different heights and cross sections. The column section may be round or rectangular. If no columns are specified above and below the design floor, a pin support is assumed.



*May vary from span to span

**May vary from support to support

Figure 2-1—System Geometry

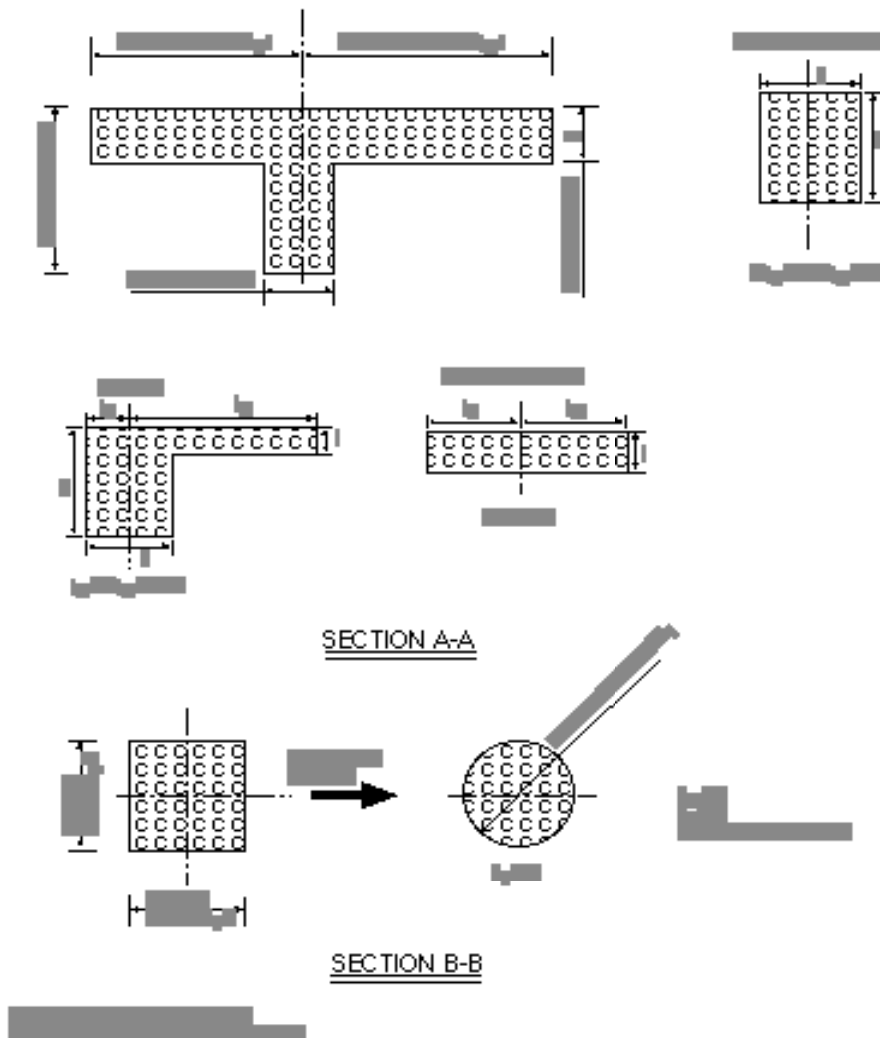


Figure 2-2—Beam, Slab, and Column Section Types

Material Properties

- Beams and columns may have different concrete properties. By entering the concrete unit weight and compressive strength, default values for the other concrete properties are determined. The concrete unit weight is used to determine the concrete type and to calculate member self weight.
- The concrete type is determined in accordance with Table 2.1.

Table 2.1—Default Concrete Types

Density (w)		Type
pcf	kg/m ³	
w 130	w 2000	Normal
105 < w < 130	1700 < w < 2000	Sand-Lightweight
w 105	w 1700	All-Lightweight

- The modulus of elasticity of concrete is computed as:

$$E_c = 33 w^{1.5} \sqrt{f_c} \quad [E_c = 0.043 w^{1.5} \sqrt{f_c}] \quad \text{Eq. 2-1}$$

where w is the unit weight of concrete and f_c is concrete compressive strength.

The square root of f_c is limited to 100 psi for the computation of the concrete shear strength, V_c , the reinforcing bar development lengths, and modulus of rupture.

- For the ACI Code, the default value of the average split tensile strength of concrete, f_{ct} , is computed based on the concrete type. *Note that The Canadian Standard does not use f_{ct} .* For normal weight concrete, the default value of f_{ct} is:

$$f_{ct} = 6.7 \sqrt{f_c} \quad \text{Eq. 2-2}$$

In no case will $f_{ct}/6.7$ exceed $\sqrt{f_c}$. f_{ct} will be modified according to the concrete type based on the default values shown in Table 2.2a. No interpolation is performed for partial sand replacement.

- The default value of the concrete modulus of rupture, f_r , is based on the average split tensile strength. f_r is then used to determine the cracking moment when computing the effective moment of inertia for deflection computations. The default value of the modulus of rupture, f_r , is:

$$f_r = 1.12 f_{ct} \quad 7.5 \sqrt{f_c} \quad [f_r = 0.6 \sqrt{f_c}] \quad \text{Eq. 2-3}$$

Table 2.2a—Default Average Split Tensile Strength

Type	f_{ct}
Normal	$6.7 \sqrt{f_c}$
Sand-Lightweight	$(0.85)6.7 \sqrt{f_c}$
All-Lightweight	$(0.75)6.7 \sqrt{f_c}$

is defined in Table 2.2b

Table 2.2b

Type	
Normal	1.0
Sand-Lightweight	0.85
All-Lightweight	0.75

No limit is imposed on f_r . Entering a large value for f_r will produce deflections based on gross section properties (i.e., uncracked section).

- The default values for the longitudinal reinforcement yield strength, f_y , and transverse reinforcement yield strength, f_{yv} , are set equal to 60 ksi [400 MPa].

Geometric Checks

PCABEAM performs geometric checks as you input the data to avoid performing an analysis with an inconsistent system. The dimensions of the beams, slabs, and columns are checked and modified if necessary to produce a legitimate system.

- If a non-zero cantilever length is set to less than one-half the exterior column depth, c_1 , the cantilever length will be increased to half of the exterior column depth.
- If the slab width is less than one-half the width of the column below the design floor, c_2 , the slab width will be increased.
- For an L-beam section, the left or right slab width must be equal to one-half the beam width.
- If the height of the column below is greater than 0, it must be greater than the beam depth (or the slab thickness for one-way slab systems).
- There is no limit imposed on the span length-to-beam depth ratio. In other words, PCABEAM does not check if the beam is a deep beam and consequently, no deep-beam design provisions are implemented in the program.

Code Checks

Prior to executing the analysis, PCABEAM checks the input data against Code requirements.

Effective Width¹

For beams with slab on both sides (T-beams), the effective flange width will not exceed one-quarter the span length², and the maximum flange overhang will not exceed eight times the slab thickness [twelve times the slab thickness for CSA Standard] or the input slab overhang on that side.

For beams with slab on one side only (L-beams), the maximum flange overhang will not exceed one-twelfth the span length², six times the slab thickness, or the input slab overhang on the overhang side.

For one-way slabs, the effective width is taken as the lesser of three times the slab thickness plus the column width, or the total slab width.

The effective width is used in computing the stiffness of the section. However, the entire slab width is considered when computing the self weight and the superimposed surface dead and live loads.

Minimum Thickness

The minimum thickness of continuous beam systems and one-way slabs, made of normal weight concrete and using Grade 60 [400] reinforcing bars, is shown in Table 2.3³. Note: l is the center-to-center span length*.

For the ACI Code, for concrete density in the range of 90-120 pcf, the above minimum thicknesses are increased by $(1.65-0.005w)$ but not less than 1.09, where w is the concrete density in pcf. For f_y other than 60 ksi, the minimum thickness is increased by $(0.4 + f_y/100)$.

¹ ACI 318-89, 8.10; CAN3-A23.3-M84, 8.10

² except for cantilevers

³ ACI 318-89, 9.5.2.1; CAN3-A23.3-M84, 9.5.5.3

*For cantilevers, l is the clear projection of the cantilever.

Table 2.3—Minimum Thickness of Continuous Beam and One-Way Slab

Minimum Thickness				
Floor System	Simply Supported	One end Continuous	Both ends Continuous	Cantilever
Continuous Beams	/16	/18.5	/21	/8
One-Way Slabs	/20	/24	/28	/10

For the CSA Standard, for low to semi-low density concrete, the above minimum thicknesses are increased by $(1.65 - 0.0003w)$ but not less than 1.00, where w is the concrete density in kg/m^3 . For f_y other than 400 MPa, the minimum thickness is increased by $(0.4 + f_y/667)$.

The minimum code required depth or thickness is printed with the output.

Loads

All applied loads are input as factored loads and are assumed to act at the longitudinal centerline of the beam. No limitations are imposed on the magnitude of the input loads or on the ratio of the dead-to-live loads. Upward (negative) vertical loads are *not* allowed.

Self Weight

The self weight of the floor system is computed internally by PCABEAM based on the input concrete unit weight and the defined geometry. Only the concrete weight of the slabs and beams is considered in the self weight computations. The weight of the reinforcing steel is ignored. Self weight is included under the Dead load case.

Superimposed Loads

Surface loads are considered acting over the entire width of the slab, if any. Torsional effects on the beam are *not* included in the analysis in case of unequal slab width on each side of the beam.

Partial loads may be input as point (concentrated), uniform (line), and trapezoidal loads. Loads acting over less than 1/20 th of the span length are averaged over 1/20 th the span length. Partial loads are not allowed within the support (column) region.

Lateral Load Moments

Lateral loads may be input as joint moments resulting from a lateral load analysis. The design is then performed under gravity and lateral load results combined. However, the effects of cracking and reinforcement on member stiffnesses must be accounted for in the lateral load analysis.

Load Combinations and Factors

PCABEAM considers the three basic load combinations per the Code. However, it allows you to change the magnification factors for the three load cases. The default values of the load factors depend on the Code selected with the **General Information** command.

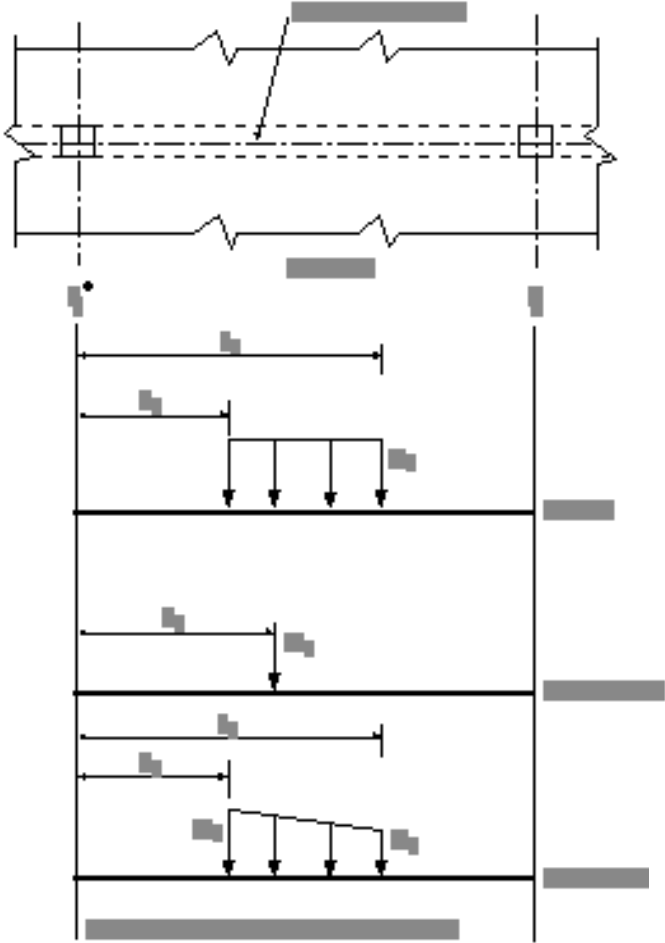


Figure 2-3—Partial Loads

For the ACI Code, the default load factors are:

$$U1 = 1.4\text{Dead} + 1.7\text{Live}$$

$$U2 = 0.75 (1.4\text{Dead} + 1.7\text{Live} + 1.7\text{Lateral})$$

$$U3 = 0.9\text{Dead} + 1.3\text{Lateral}$$

For the CSA Standard, the default load factors are:

$$U1 = 1.25\text{Dead} + 1.5\text{Live}$$

$$U2 = 1.25\text{Dead} + 0.7 (1.5\text{Live} + 1.5\text{Lateral})$$

$$U3 = 0.85\text{Dead} + 1.5\text{Lateral}$$

Note that the Dead and Live load factors of the first load case, U1, are linked to the Dead and Live load factors of the second load case, U2. Therefore, different factors for the Dead and Live loads cannot be input for the two cases.

Loading Patterns

The analysis of the floor system requires the consideration of pattern loading. With PCABEAM, up to eight loading patterns are considered. The Code⁴ allows the use of 75% of the live load for pattern loading. This percentage, however, may be modified.

For gravity loads only, PCABEAM selects the design moments and shears based on the following loading patterns (see Figure 2-4):

Pattern 1: Combination U1, full dead load + 75% live load on two adjacent spans.

Pattern 2: Combination U1, full dead load + 75% live load on alternate odd spans.

Pattern 3: Combination U1, full dead load + 75% live load on alternate even spans.

Pattern 4: Combination U1, full dead load + full live load on all spans.

For combined gravity and lateral loads, the following additional loading patterns are considered:

Pattern 5: Combination U3, full dead load + lateral load (left-to-right).

Pattern 6: Combination U3, full dead load + lateral load (right-to-left).

Pattern 7: Combination U2, full dead load + lateral load (left-to-right).

Pattern 8: Combination U2, full dead load + lateral load (right-to-left).

⁴ ACI 318-89, 13.7.6.3; CAN3-A23.3-M84, 13.7.6.3

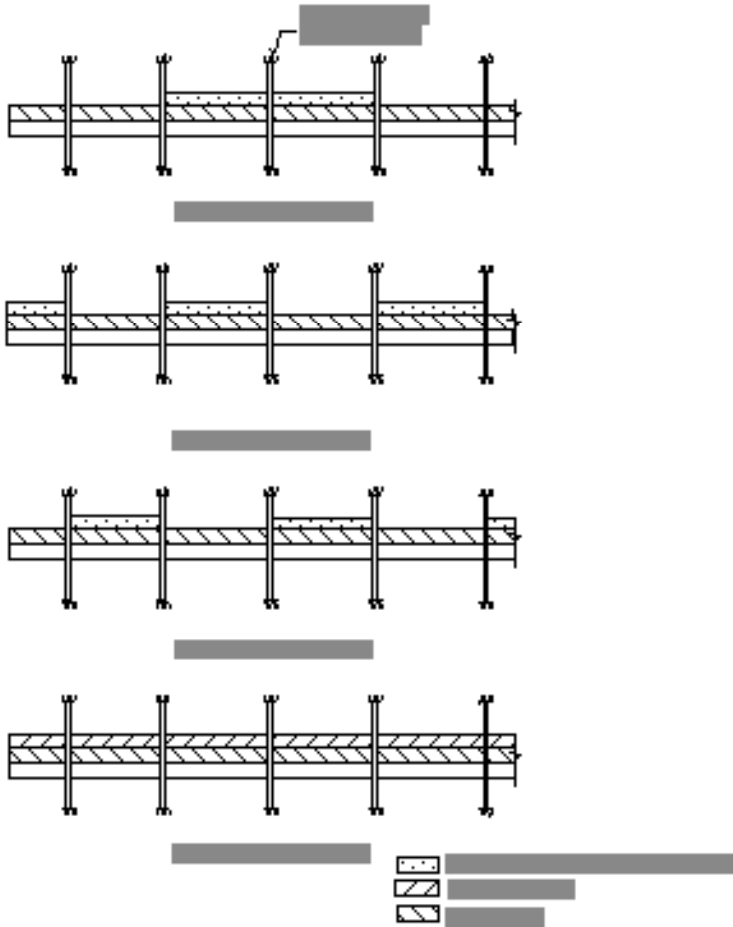


Figure 2-4—Basic Live Load Patterns

Design Assumptions

The design assumptions and limits used, conform to the provisions of the design specifications based on the accepted Strength Design or Limit States theories for reinforced concrete design. The following is a brief summary of the method of solution.

1. All conditions of strength satisfy the applicable conditions of equilibrium and compatibility of strains.
2. Strain in the concrete and reinforcement is directly proportional to the distance from the neutral axis.
3. The maximum usable strain at the extreme concrete compression fiber is assumed to 0.003.
4. The equivalent rectangular concrete stress block is used, for which the block depth is:

$$a = \beta_1 c$$

where:

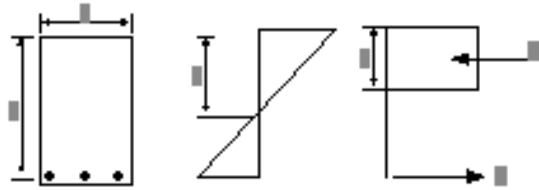
c = distance from extreme compression fiber to neutral axis.

For the ACI Code:

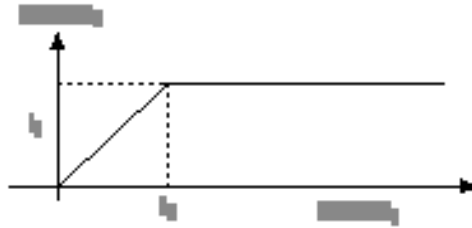
$$\begin{aligned} \beta_1 &= 0.85 && \text{for } f_c \leq 4 \text{ ksi} \\ &= 0.65 && \text{for } f_c \geq 8 \text{ ksi} \\ &= 1.05 - 0.05 f_c && \text{for } 4 \text{ ksi} < f_c < 8 \text{ ksi} \end{aligned}$$

For the CSA Code:

$$\begin{aligned} \beta_1 &= 0.85 && \text{for } f_c \leq 30 \text{ MPa} \\ &= 0.65 && \text{for } f_c \geq 55 \text{ MPa} \\ &= 1.08 - 0.008 f_c && \text{for } 30 \text{ MPa} < f_c < 55 \text{ MPa} \end{aligned}$$



5. The elastic-perfectly 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 design yield strength, the reinforcement stress remains constant and equal to f_y . The modulus of elasticity of steel, E_s , is taken as 29,000 ksi [200,000 MPa].



6. Based on the above assumptions, for a rectangular section with one layer of reinforcement, the required area of steel is:

$$A_s = bd$$

$$= g(1 - \sqrt{1 - m})$$

$$m = \frac{2M_u}{0.85f_c b d^2 c_f}$$

$$g = 0.85 \frac{c_f}{s_f y}$$

c_f , s_f , and y are reduction factors defined in Table 2.4.

Table 2.4—Reduction Factors

Code	c	s	f
ACI	1.0	1.0	0.9
CSA	0.60	0.85	1.0

7. If the neutral axis of the section is greater than the slab thickness, a T-section analysis is performed based on the same principles and assumptions mentioned above.
8. For the investigation option, the same principles and assumptions of stress-strain compatibility are followed to compute the moment capacity of the section. Note that for computing the section moment capacity, both top and bottom steel are taken into consideration.
9. Balanced reinforcement ratio, ρ_b , is computed based on the assumption that the first layer of tension reinforcement reaches the strain corresponding to its specified yield strength, f_y , at the same time as the concrete in compression reaches its assumed ultimate strain of 0.003.
10. PCABEAM assumes that all beam and slab sections are symmetrical about the plane of analysis. The plane of analysis is the vertical plane at the support centerlines.

Analysis

PCABEAM uses the direct stiffness method for analysis. The stiffness matrix of the structure is assembled and the simultaneous equations are solved for equilibrium considering only the rotational degrees of freedom at the joints. PCABEAM assumes that the supports are fixed against translational, vertical, and horizontal displacements. The program then solves for the joint rotations and obtains the member end forces for each loading pattern.

Column ends (top end of the columns above and bottom end of the columns below) are assumed to be fixed. The gross column section multiplied by its input stiffness reduction factor is considered in the analysis. To account for the cracked column section, you may modify the stiffness reduction factor (SRF) of a column (default is 100%). If a stiffness reduction factor is greater than 100%, then large flexural column stiffness will be assigned fixing the support not to rotate. When a stiffness reduction factor is equal to 0, then a pin support is considered. Note that when $SRF = 0$ or greater than 100%, or when no columns exist, joint moments due to lateral forces will be equal to zero. Note, however, that the same reduction factor applies for both columns above and below, at a support. If no columns are specified, a pin support is assumed.

In computing intermediate results along a member, PCABEAM divides each span into 20 equal segments and computes the shear, moments, and deflections at the center of each segment.

Moment Redistribution

Current design codes such as ACI 318-89 and CAN3-A23.3-M84 prescribe required strength forces as those obtained from structural elastic analysis multiplied by appropriate load factors. However, in order to take into account the effects of flexural cracking and yielding that generally occur in concrete elements, they allow a limited amount of moment redistribution. This amount of moment redistribution depends on the rotational ductility provided at critical sections where plastic hinges may form. PCABEAM follows a moment redistribution technique suggested by T. Paulay⁵. This technique is to be used mainly when lateral forces control the beam design producing

5 Thomas Paulay, "Moment Redistribution in continuous beams of Earthquake Resistant Multistory Reinforced Concrete Frames," *Bulletin of the New Zealand National Society of Earthquake Engineering*, Vol. 9, No. 4, New Zealand, 1976.

uneven demands for tension and compression reinforcement at member sections. The principle of equilibrium is satisfied and the prescribed lateral strength capacity is maintained. Therefore, PCABEAM allows for moment redistribution only for negative moments at supports according to the following code equations⁶.

For the ACI Code:

$$x = 20 \left(1 - \frac{\rho'}{\rho_b} \right) \text{ percent} \quad \text{Eq. 2-4}$$

$$\rho' \leq 0.5 \rho_b \quad \text{Eq. 2-5}$$

where:

- x = percentage of moment redistribution
- ρ = tension reinforcement ratio
- ρ' = compression reinforcement ratio
- ρ_b = balanced reinforcement ratio

For the CSA Code:

$$x = 30 - 50 \frac{c}{d} \quad 20\% \quad \text{Eq. 2-6}$$

where:

- c = distance to neutral axis at ultimate level
- d = effective depth

While any value of moment redistribution percentage less than 20 percent may be input, the program computes the maximum values allowed by the Code and prints a message if they are exceeded.

Under the design option, PCABEAM revises the amount of compression reinforcement needed to satisfy the specified moment redistribution percentage as follows:

⁶ ACI 318-89, 8.4; CAN3-A23.3-M84, 8.4

For the ACI Code:

$$- \quad 1 - \frac{x}{20} \quad b \quad 0 \quad \text{Eq. 2-7}$$

For the CSA Code:

$$\frac{c}{d} \quad 0.6 - \frac{x}{50} \quad \text{Eq. 2-8}$$

Deflections

Deflections are based on the deflection due to simply supported condition less the deflection due to end support rotations. Deflections are computed for dead loads and for total (dead + live) loads. Total load deflections are computed for pattern 4 only (full dead and live load on all spans). Concentrated loads occurring at other locations will give rise to larger deflections. These must be checked manually. The effects of support settlement or shear deformations are not considered.

Note that these deflections are elastic (instantaneous) deflections and do not include creep and shrinkage deflections. For a crude approximation, additional long-term deflections may be computed as three times the elastic deflections.

The effect of cracking in slabs and beams is a reduction in the flexural stiffness and, consequently, an increase in deflections. At the critical sections along any span, the effective moment of inertia used to compute deflections as required by Code⁷ is:

$$I_e = I_g \frac{M_{cr}^3}{M_{max}^3} + I_{cr} \left(1 - \frac{M_{cr}^3}{M_{max}^3} \right) \quad \text{Eq. 2-9}$$

where:

I_g = moment of inertia of the gross transformed uncracked concrete section.

I_{cr} = moment of inertia of the cracked transformed concrete section.

⁷ ACI 318-89, 9.5.2.3; CAN3-A23.3-M84, 9.5.2.3

M_{cr} = cracking bending moment

M_{max} = maximum bending moment at the load stage for which deflection is computed

The effective moment of inertia is considered to be between the following limits:

$$I_{cr} \leq I_e \leq I_g$$

Under the Design option, if PCABEAM cannot compute the required reinforcement for a span for any reason, I_{cr} is approximated as $0.4I_g$.

To apply the Code specified effective moment of inertia, an equivalent span moment of inertia, I_{span} , is used⁸.

For an interior span, the equivalent span moment of inertia is:

$$I_{span} = 0.7I_e^+ + 0.3 \frac{I_{e,l}^- + I_{e,r}^-}{2} \quad \text{Eq. 2-10}$$

For a cantilever, the equivalent span moment of inertia is:

$$I_{span} = 0.15I_{e,sup}^- + 0.85I_{e,end}^- \quad \text{Eq. 2-11}$$

where:

I_e^+ = effective moment of inertia for the positive moment region

$I_{e,l}^-$ = effective moment of inertia for the negative moment region at left support

$I_{e,r}^-$ = effective moment of inertia for the negative moment region at right support

$I_{e,sup}^-$ = effective moment of inertia for the negative moment region at cantilever support

$I_{e,end}^-$ = effective moment of inertia for the negative moment region at cantilever end

⁸ ACI Committee 435, *State-of-the-art Report, Deflection of Two-Way Floor Systems, Special Publication SP43-3*, American Concrete Institute, Detroit, Michigan, 1974.

Design

Once the analysis is finished and the moment and shear envelopes are obtained, PCABEAM will attempt to design the system (under the Design option).

Flexure

PCABEAM computes the design moment envelopes producing tension at the top and the bottom of the critical sections. Critical sections are at the faces of each support and at the location of maximum positive moment. If negative moment is encountered near midspan, another critical section is defined at that location.

Minimum reinforcement

The minimum reinforcement ratio for beams will not be less than⁹:

$$\rho_{\min} = 200/f_y \quad [\rho_{\min} = 1.4/f_y] \quad \text{Eq. 2-12}$$

For the ACI Code, the minimum reinforcement ratio for slabs (based on the slab gross section) is 0.0018 when the reinforcement yield strength, f_y , is 60 ksi, and $0.0018 \times 60/f_y$ when f_y exceeds 60 ksi. In no case will this ratio be less than 0.0014 ¹⁰. For the CSA Standard, the reinforcement ratio is 0.002 for all steel grades¹¹.

Maximum reinforcement

For the ACI Code, the reinforcement ratio, $\rho = A_s/bd$, is kept below 75 percent of the balanced reinforcement ratio, ρ_b ¹².

For the CSA Standard, the area of tension reinforcement is such that the neutral axis-to-depth ratio is¹³:

$$c/d \leq \frac{600}{600 + f_y} \quad \text{Eq. 2-13}$$

⁹ ACI 318-89, 10.5.1; CAN3-A23.3-M84, 10.5.1

¹⁰ ACI 318-89, 7.12

¹¹ CAN3-A23.3-M84, 7.8.1

¹² ACI 318-89, 10.3.3

¹³ CAN3-A23.3-M84, 10.3.3

Integrity reinforcement¹⁴

For exterior beams, at least one-sixth the tension reinforcement required for the negative moment at the support will be made continuous. Also for exterior beams, at least one-quarter of the positive moment reinforcement required at midspan will be made continuous.

For interior beams with open stirrups, at least one-quarter of the positive moment reinforcement required at midspan will be made continuous.

Required Reinforcement

Based on the Design Assumptions, the required reinforcement at the top and bottom of each critical section is computed, compared with the minimum reinforcement and the required reinforcement for integrity, and the larger amount is provided.

For T- or L-sections, the top reinforcement is computed based on a rectangular section as described above. For bottom reinforcement, a rectangular section is first assumed and the stress block depth is computed. If this falls below the flange depth, a T-section analysis is performed.

For a rectangular beam section, if the quadratic equation to compute the required area of steel cannot be solved, the program will try to design the section as doubly-reinforced section.

Reinforcement Selection

PCABEAM tries to fit the top or bottom bars in one layer. If not possible, the program tries to fit the bars in two layers for top and bottom bars of a rectangular beam section and for bottom bars of an L-beam or a T-beam section.

The specified minimum spacing is compared to the code prescribed minimum spacing of reinforcing bars of 1 in. [25 mm] or one bar diameter¹⁵ and the larger value is selected.

The maximum spacing satisfying flexural cracking requirements (z factor) ACI 318-89, 10.6.4 and R10.6.4 [CAN3-A23.3-M84, 10.6.4] considering one bar in one layer is compared to the specified maximum spacing and the lesser value is selected.

¹⁴ (ACI Code only) ACI 318-89, 7.13.2

¹⁵ ACI 318-89, 7.6.1; CAN3-A23.3-M84, 7.4.1.1

Besides, for slabs only, the Code prescribed maximum spacing of 18 in. [500 mm] or twice the slab thickness for slabs¹⁶ is compared to the specified maximum spacing and the lesser value is selected.

An iterative procedure is performed to determine the required number of bars and bar size. Based on the required area of steel, A_{sreq} , and the minimum and maximum specified bar sizes, the program computes a range of bar numbers for each bar size. The program then picks the combination that is closest to (but larger than) the required area of steel. The bar spacing of the bar selection is then compared to the minimum and maximum spacings. If the spacing criteria is not met, the combination is ruled out and another one is selected. This iterative process is repeated until all combinations are exhausted, after which the program tries placing the bars in two layers (if applicable).

If bars need to be placed in two layers, the program does not recalculate the required area of steel based on a new effective depth but rather increases the area of steel by the ratio of d_1/d_2 where d_1 is the effective depth based on one layer and d_2 is the effective depth based on two layers. If bars cannot be fit into the section, a message is printed with the output.

Shear

PCABEAM computes the design shear envelopes (when moment redistribution is considered, the most critical shear envelope before and after moment redistribution is used for design) at the critical sections and along the span. Critical sections are taken at a distance “d” (effective depth) away from the face of the support for interior spans. If a concentrated load is encountered within a distance “d” away from the support, the critical section is taken at the face of the support. For cantilevers, the critical section is at the face of the support.

Every span is divided into a number of segments. Each segment is then designed for the maximum factored shear, V_u , within that segment.

Concrete Shear Strength

The concrete shear strength of a section, V_c , is¹⁷:

$$V_c = 2\sqrt{f_c} b_w d \quad [V_c = 2 \sqrt{f_c} b_w d] \quad \text{Eq. 2-14}$$

¹⁶ ACI 318-89, 7.6.5 and 13.4.2; CAN3-A23.3-M84, 7.4.1.2 and 13.4.2

¹⁷ ACI 318-89, 11.3.1.1; CAN3-A23.3-M84, 11.2.5.2

where b_w is the beam web width and d is the effective beam depth.

Minimum Shear Reinforcement

When $V_u > V_c/2$, the beam must be provided with at least a minimum shear reinforcement of¹⁸:

$$A_v = 50b_w s / f_{yv} \quad [A_v = 0.35b_w s / f_{yv}] \quad \text{Eq. 2-15}$$

If $V_u \leq V_c/2$, no shear reinforcement is required.

Maximum Shear Reinforcement

If V_u exceeds $5V_c$, the beam section dimensions must be increased or higher concrete strength must be provided¹⁹.

Maximum Stirrup Spacing

For the ACI Code, when $V_s \leq 2V_c$, the stirrup spacing must not exceed 24 in. nor $d/2$. When $V_s > 2V_c$, the maximum spacing is reduced by half²⁰ to 12 in. or $d/4$.

For the CSA Standard, when $V_s \leq 2V_c$, the stirrup spacing must not exceed 600 mm nor $d/2$. When $V_s > 2V_c$, the maximum spacing is reduced by half²¹ to 300 mm or $d/4$.

Required Shear Reinforcement

When $V_u > V_c$, the required shear reinforcement is²²:

$$A_v = \frac{sV_s}{f_{yv}d} \quad \text{Eq. 2-16}$$

where A_v is the total area of a stirrup, s is the required stirrup spacing, b_w is the beam web width, and $V_s = V_u - V_c$.

18 ACI 318-89, 11.5.5; CAN3-A23.3-M84, 11.2.5

19 ACI 318-89, 11.5.6.8; CAN3-A23.3-M84, 11.3.6.6

20 ACI 318-89, 11.5.4

21 CAN3-A23.3-M84, 11.3.8

22 ACI 318-89, 11.5.6.1; CAN3-A23.3-M84, 11.3.6.1

Stirrup Selection

Beam stirrups are selected starting with the maximum spacing input, two legs of stirrups, and the smallest stirrup size. If this does not satisfy the required A_v/s ratio, the spacing is reduced until it reaches the minimum specified, then number of legs is increased (to a maximum of four legs), and finally the bar size is increased until it reaches the maximum specified.

Investigation

Flexure

Flexural capacity is computed based on the input longitudinal top and bottom bars. Bars, top or bottom, may be discontinuous or continuous throughout the span. A span may contain up to five sets of continuous bars and up to five sets of discontinuous bars, each top and bottom. Note that, with the investigation option, PCABEAM does not check the Code requirements regarding bar spacing, minimum reinforcement, maximum reinforcement, or development length and lap splices.

Based on the input top and bottom steel, PCABEAM computes the section capacity at the critical sections (defined in *Flexure* under *Design on page 2-19*). If at a section, both top and bottom bars are provided, the section capacity takes into account the tension and compression steel.

Shear

Transverse reinforcement along a span is input in sets of stirrups. A stirrup set is a group of uniformly distributed stirrups that are of the same size and have the same number of legs. The location of the first stirrup from the face of the left support must be input for each span. With the Investigation option, PCABEAM does not check any Code requirements regarding stirrup spacing, whether the minimum area of shear reinforcement is provided, or whether the maximum area of shear reinforcement is exceeded.

Reinforcement

Design Parameters

PCABEAM allows you to set the criteria that will govern the selection of reinforcing bars and stirrups by inputting reinforcement design parameters. You may specify the minimum and maximum number and size for top and bottom bars in a section in addition to the vertical cover, side cover, minimum clear spacing between bars, and maximum center-to-center spacing. Besides, you also specify the minimum and maximum stirrup size and spacing and whether stirrups are closed or open.

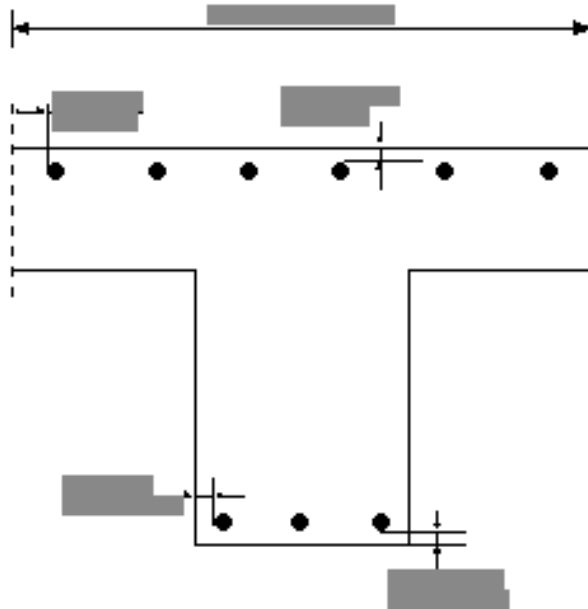


Figure 2-5—Section Design Parameters

Discontinuous Bars

A discontinuous top bar set consists of a number of bars on the left end of a span and a number of bars on the right end of the span. Bars on each end may be of different number and different size and do not lap each other.

A discontinuous bottom bar set consists of a number of bars in the center portion of the span, a number of bars on the left end of a span, and a number of bars on the right end of the span. The center bars and/or the bars on each end may be of different number and/or size and do not lap each other.

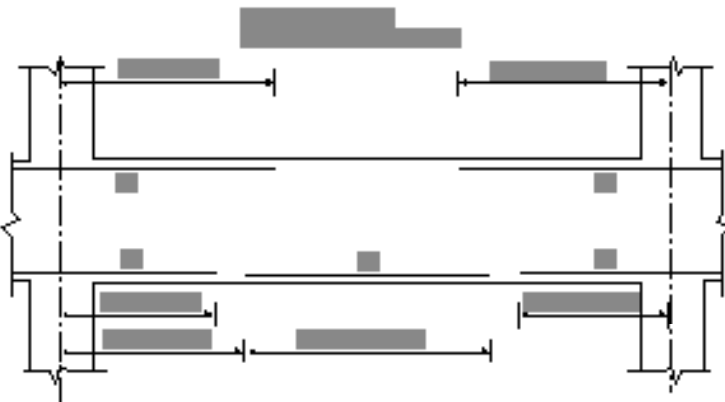


Figure 2-6—Discontinuous Bars

Continuous Bars

A continuous bar set, whether top or bottom, may consist of any one or more of the following types: single bar, two-lapped bars, or three-lapped bars (see Figure 2-7).

a) Single-bar set: consists of a group of bars that are continuous throughout the span from one end to the other. For example, a single bar set may contain 5-#7 bars that are continuous throughout the span.

b) Two-lapped bar set: consists of a number of bars on the left end of the span lapped with an equal number of bars on the right end of the span. Left and right bars may be of different sizes. For example, 4-#7 bars on the left end that are lapped with 4-#8 bars on the right end, form a set of four continuous bars.

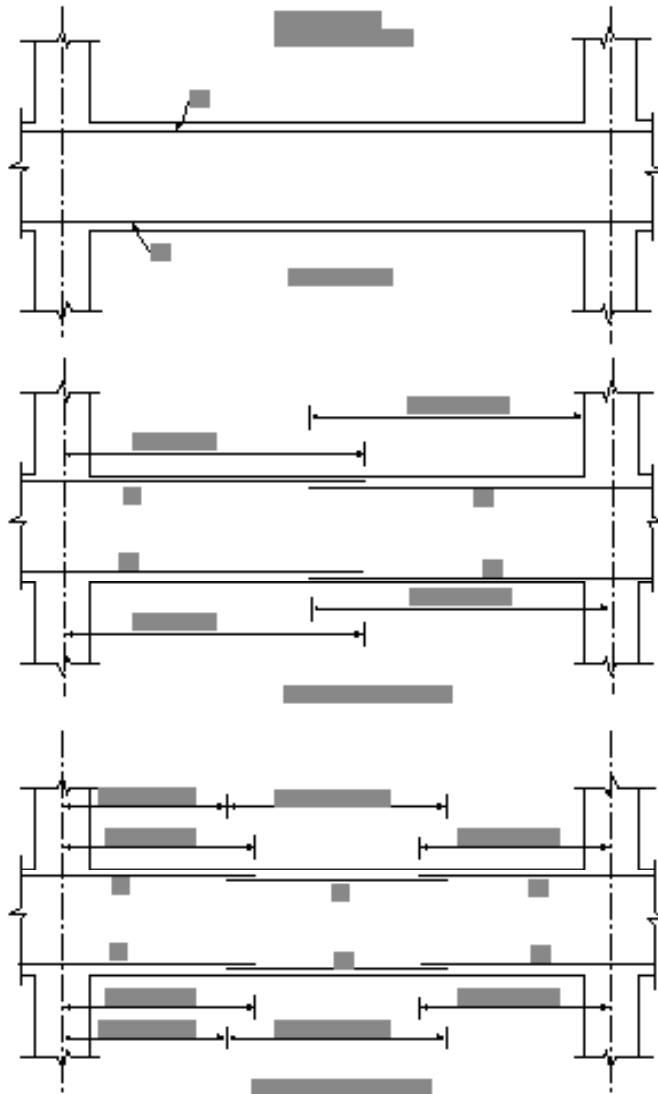


Figure 2-7—Continuous Bars

c) **Three-lapped bar set:** consists of a number of bars on the left end of the span, an equal number of bars on the right end of the span, and an equal number of bars in the center portion of the span. Center bars must lap the end bars. Left, right, and center bars may be of different sizes. For example, 3-#7 bars on the left end, that are lapped with 3-#6 bars in the middle which, in turn, are lapped with 3-#7 bars on the right end of the span, form a set of three continuous bars.

Bar sets may be in different layers and the distance from the top of the section, for top bars (the bottom of the section for bottom bars) to the centroid of the layer is referred to as the layer distance. Bars of different sets that exist in the same layer will have the same layer distance.

Stirrups

Stirrups are defined for each span in sets. You may define up to five stirrup sets per span. A stirrup set consists of a number of stirrups that have the same stirrup size, number of legs per stirrup, and center-to-center spacing between stirrups. Moving from the left-span end to the right, the location of the first stirrup from the face of the support must be defined. Consequently, stirrup sets are input in the order they exist along the span. Stirrups may not extend beyond the clear span length.

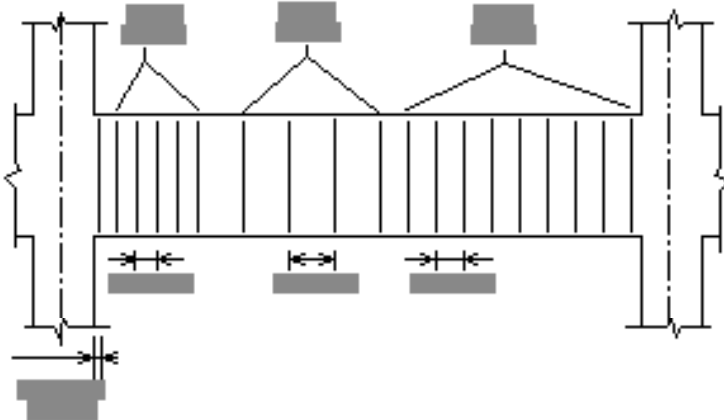


Figure 2-8—Stirrups

Bar Lengths and Splices

Bar lengths are computed, compared to the minimum requirements specified by the Code, and adjusted as necessary. The output bar lengths do not include hooks or bends.

The development length of longitudinal bars is the basic development length, l_{db} , multiplied by an applicable code case factor, F_{code} , and modifying factors, f_i .

$$d = l_{db} F_{code} \prod_{i=1}^7 f_i \quad \text{Eq. 2-17}$$

where l_{db} is computed per ACI 318-89, 12.2.2 [CAN3-A23.3-M84, 12.2.2], and denote multiplication of applicable modifying factors, f_i .

Besides²³:

$$l_{db} F_{code} \prod_{i=1}^2 0.03d_b \frac{f_y}{\sqrt{f_c}} \quad \text{Eq. 2-18}$$

If the above equation is not satisfied, PCABEAM prints an increased value of F_{code} to compensate for this requirement.

F_{code} depends on the following factors: concrete cover, bar lateral support, minimum transverse reinforcement, special transverse reinforcement, layer location, and clear bar spacing. Table 2.5 shows the code requirements specified for each F_{code} factor. For the CSA Standard, F_{code} is equal to 1.0.

The modifying factors, f_i , depend on the following parameters: concrete cover, clear bar spacing, stirrup size, aggregate weight, epoxy-coated reinforcement, and reinforcing bar yield strength. A total of seven different factors may be classified as shown in Table 2.6. Note that for a particular case only some factors may be applicable. Also, note that when epoxy-coated bars are used, the product $f_3 \times f_4$ need not exceed 1.7. For the CSA Standard, the product of the modifying factors less than 1.0 shall not be less than 0.6.

²³ ACI 318-89, 12.2.3.6

Table 2.5—Development Length Code Case Factor, F_{code}

			Code Requirements					
Code Case	F_{code}	Code Section	Concrete Cover	Bar Lateral Support	Min. Stirrup Spacing	Special Transverse Reinforcement	Layer Location	Bar Clear Spacing
1	1.0	12.2.3.1 (a)	minimum 7.7.1	7.10.5	11.5.4 & 11.5.5.3	–	–	$3d_b$
2	1.0	12.2.3.1 (b)	minimum 7.7.1	–	–	12.12.3.16	–	–
3	1.0	12.2.3.1 (c)	–	–	–	–	inner	$3d_b$
4	1.0	12.2.3.1 (d)	$2d_b$	–	–	–	–	$3d_b$
5	2.0	12.2.3.2	d_b	–	–	–	–	$2d_b$
6	1.4	12.3.3	All other conditions not included above					

Table 2.6—Modifying Development Length Factors, f_i

			Code Requirements						
i	f_i	Code Section	Concrete Cover	Bar Clear Spacing	Bar Size	Stirrup Size	Stirrup Spacing	Lateral Support	Bar Coating
1	0.80	12.2.3.4	$2.5d_b$	$5d_b$	#11	–	–	–	–
2	0.75	12.2.3.5	–	–	–	#4	4 in.	7.10.5	–
3	1.5	12.2.4.3	$3d_b$	$6d_b$	–				Epoxy
	1.2				All other conditions				
4	1.3	12.2.4.1	Top reinforcement						
5	1.3 or $6.7\sqrt{\frac{f_c}{f_{ct}}}$	12.2.4.2	Light aggregate						
6	$\frac{A_{sreq}}{A_{sprod}}$	Excess reinforcement							
7	$2 - \frac{60,000}{f_y}$	Yield strength							

Results

The output of PCABEAM is organized into tables and can be input echo, analysis results, design results, investigation results, quantity estimate tables, and deflection results. The amount of output tables depends on the options that you chose prior to a run.

Input echo tables are a printout of the input data and may sometimes include warning messages or notes regarding the conformance of the input data to the selected Code. The input echo consists of the applicable of the following: general information, material properties, beam and slab geometry, column geometry, vertical superimposed and partial loads, moment redistribution percentages, loading combinations and patterns, reinforcement design parameters, longitudinal reinforcement schedule, and transverse reinforcement schedule.

Analysis results

Service Loads Transferred to Columns. This table lists the service level axial load and moments in the columns for each of the dead, live, and lateral load cases. The moments listed in the table are joint centerline moments and the axial loads on the column represent the loading of a single floor only. Any axial loads on the column from the floors above must be added to this story's axial loads when designing the column.

Loading Pattern Moments and Shears at Supports. This table lists, for each support, the centerline moments and shears corresponding to the selected loading patterns. The sign convention used in this table is shown in Figure 2-2. Note that the results corresponding to pattern 1 are based on the condition of the live load acting on the adjacent spans. For example, moments and shears for joint 2 are based on the assumption of live loads acting on spans 2 and 3. If the moment redistribution option is active, the listed moments and shears (M_u and V_u) are those obtained after moment redistribution. M_u/M_o represents the ratios of the moment after moment redistribution to the moment before redistribution, in percentage (likewise for V_u/V_o).

Segmental Required Moments and Shears. This table lists moments and shears at 22 locations along each span for the selected loading patterns. If the moment redistribution option is active, the listed moments and shears are those obtained after moment redistribution. Each span is divided into 20 equal segments and the results are presented at the center of each segment in addition to the start and end of

a span.

Segmental Moment and Shear Envelopes. This table lists the minima and maxima values of the moments and shears (M_u and V_u) of all selected loading patterns along with the governing pattern number. Also listed are the service level minima and maxima values of moments and shears (M_s and V_s) of all selected gravity only loading patterns (1 through 4) along with the governing pattern. The ratios of M_u/M_s and V_u/V_s are also listed. All these values are listed at the 22 locations defined in the previous paragraph.

Design Results

Required Longitudinal Reinforcement at Critical Sections. This table lists, for each span, the location of the critical section for negative (Top) and positive (Bot) flexural design moments. Critical sections are measured from the left support centerline. The table also lists the effective width and depth of the section, the required moments at the critical sections along with the corresponding loading patterns, the required steel area, the layer distance, and the required number of bars and the bar size. This table may also contain notes related to the required area of steel. “Int” means that the steel area is needed to satisfy integrity requirements. “Min” means that the steel area is the minimum required. “Dob” means that the section is doubly reinforced. “Red” means that the steel area is resulting from moment redistribution. Other notes (denoted by numbers to the right of the table are explained at the bottom of the table.

Top Bars Development Lengths. This table lists, for each span and for each bar size in a span, the basic development length of the bar (Ldb), the bar clear cover (COV), the bar clear spacing (SPAC), the code case factor (CODE CASE), the remaining modifying factors for concrete cover and spacing (C/S), top bars (TOP), excess reinforcement (EXC), and epoxy coated bars (EPX), and the bar development length (Ld).

Bottom Bars Development Lengths. This table is similar to the Top Bars Development Lengths (explained above) except that it applies to the bottom bars.

Proposed Longitudinal Reinforcement Schedule. Based on the required area of steel at the different sections along a span and the bar selection criteria explained in Flexural Design, this table presents the proposed bar schedule. Bars are classified here as continuous and discontinuous and their corresponding lengths are also shown.

Provided Flexural Capacities. This table lists for each span, the location of all critical sections measured from the left support) and the design moment M_n at that section computed based on the proposed reinforcement. Also listed are the ratio of the design-to-required moment, the neutral axis depth (from the compression face), the reinforcement ratio (), the provided-to-maximum reinforcement ratio, the computed Z factor, the layer distance, and the number of bars and bar size.

Required Transverse Reinforcement at Beam Segments. This table lists for each span the steel required for shear reinforcement. Each span is divided into segments whose start and end locations are shown measured from the left support. For each segment, the required shear (V_u), the ratio of the required shear to the reduced concrete shear strength V_u/V_c , the governing load pattern, and the ratio of the required steel area to stirrup spacing A_v/s are shown. Based on A_v/s , the proposed stirrup number of legs, number, size, and spacing are also shown.

Proposed Transverse Reinforcement Schedule. Based on the Required Transverse Reinforcement at Beam Segments, and taking into account that PCABEAM provides minimum stirrups in regions where no shear reinforcement is required, this tables gives the proposed stirrup schedule. Again, each span is broken down into segments whose start and end distance (from the left support) are listed. For each segment the stirrup arrangement (number of legs, number of stirrups, size, and spacing) is shown along with the corresponding design shear, the ratio of the design shear to the required shear, and the governing load pattern.

Segmental Flexural and Shear Capacities. At 22 locations along a span, this table lists the negative (Top) and positive (Bot) design and ratio of design to required moment and shear. For the design option these values are based on the proposed longitudinal and transverse reinforcement schedule. For the investigation option, they are based on the input bars and stirrups.

Investigation Results

Top Bars Development Lengths. This table lists, for each span and for each bar size in a span, the basic development length of the bar (Ldb), the bar clear cover (COV), the bar clear spacing (SPAC), the code case factor (CODE CASE), the remaining modifying factors for concrete cover and spacing (C/S), top bars (TOP), excess reinforcement (EXC), and epoxy coated bars (EPX), and the bar development length (Ld).

Bottom Bars Development Lengths. This table is similar to the Top Bars Development Lengths (explained above) except that it applies to the bottom bars.

Provided Flexural Capacities. This table lists for each span, the location of all critical sections (measured from the left support) and the design moment M_n at that section computed based on the provided reinforcement. Also listed are the ratio of the design moment to the required moment, the neutral axis depth (from the tension face), the reinforcement ratio (), the ratio f_y / f_{max} , the computed Z factor*, the layer distance, and the number of bars and bar size.

Provided Shear Capacities. This table lists for each span, the different segments that contain the same amount of stirrups (Start and End are measured from the left support) and the design shear V_n in each segment. Also listed are the ratio of the design-to-required shear, the corresponding load pattern, and the stirrups number, number of legs, size, and spacing in each segment.

Segmental Flexural and Shear Capacities. At 22 locations along a span, this table lists the negative (Top) and positive (Bot) design and ratio of design-to-required moment and shear. These values are based on the input bars and stirrups.

Deflections

Segmental Uniform Service Load Deflections. At 22 locations along each span, the deflections due to dead and total (dead + live) loads are listed as well as the ratio of dead-to-total load deflection. These deflections are based on cracked section properties.

Maximum Uniform Service Load Deflections. This table lists the maximum deflections due to dead and total (dead + live) loads with their corresponding locations. Also listed are the cracking moment (M_{cr}), the ratio M_a/M_{cr} *, the gross moment of inertia (I_{gt}), the ratio of cracking-to-gross moment of inertia (I_{cr}/I_{gt}), and the effective moment of inertia (I_{eff}).

Material Quantity Estimate

* When $M_a/M_{cr} \leq 1.0$, the Z factor is set equal to zero and I_{eff} is set equal to I_{gt} .

Concrete Quantity Estimate. This table gives an estimate of the total area of the input frame, the area of formwork, and the volume of concrete.

Longitudinal Reinforcement Quantity Estimate. This table gives an approximate estimate of the longitudinal reinforcing bars, their weight and total length. These figures are based on the proposed (Design option) or input (Investigation option) longitudinal bars.

Transverse Reinforcement Quantity Estimate. This table gives an approximate estimate of the transverse reinforcing bars and their total weight. These figures are based on the proposed (Design option) or input (Investigation option) transverse bars.

Chapter 3

Menu Descriptions

This chapter gives a brief description of the PCABEAM window and a summary of different commands you will encounter using PCABEAM in the Microsoft Windows environment.

The PCABEAM Window

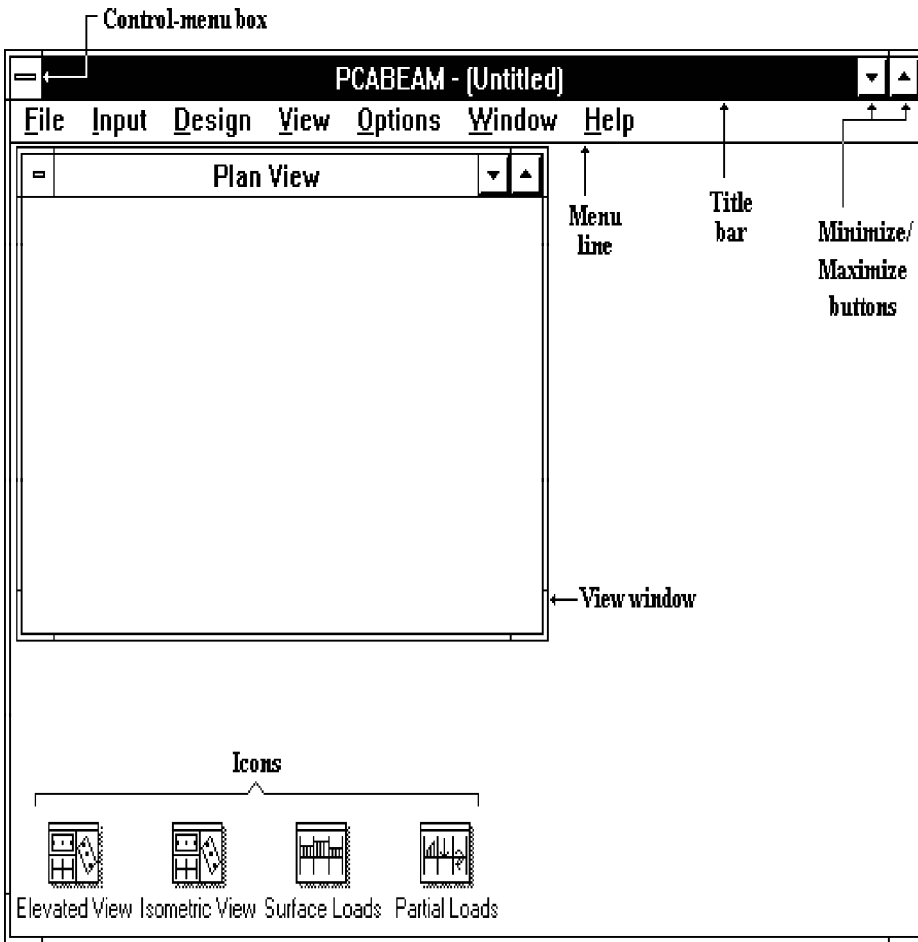
Most windows have certain components in common, such as a title bar and menus. Not all windows, however, have every component. To learn more about how to use window elements, refer to your Microsoft Windows manual. The following is a list of the PCABEAM window components.

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. To access the Control-menu box using the keyboard, press ALT + - (hyphen).

Minimize/Maximize buttons

The Minimize and Maximize buttons are located in the upper-right hand corner of the window. The Minimize button (represented by a down arrow) shrinks the window to an icon. The Maximize button (represented by an up arrow) enlarges the window so that it fills the entire desktop. After you maximize the window, the Maximize button is replaced by the *Restore button* (contains both an up and a down arrow); clicking on the Restore button returns the window to its previous size.



The PCABEAM Window

Title bar

The Title bar shows the name of the application or document (PCABEAM in this case) along with the name of the current data file in use. If the data current 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 and lists the available menus. A menu contains a list of commands, or actions, that may be executed with Windows. In PCABEAM, there are seven main pull-down menus: **File**, **Input**, **Design**, **View**, **Options**, **Window**, and **Help**.

View window

The View window (open) contains a graphical display of any of the following: model geometry (plan, elevation, or isometric views), applied loads (surface and partial), or any requested diagrams after a solution (shear, moment, and deflection).

Icons

The Icons are located along the bottom of the PCABEAM window. These are the minimized view windows.

Working With Menus

To access a menu item using the mouse, place the arrow cursor on 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 that menu. For example, to open the **F**ile menu, press ALT + F.

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

To choose a command from a selected menu, click the left-mouse button on the command, or use the UP or DOWN ARROW key to select the command, and press ENTER.

Menu Conventions

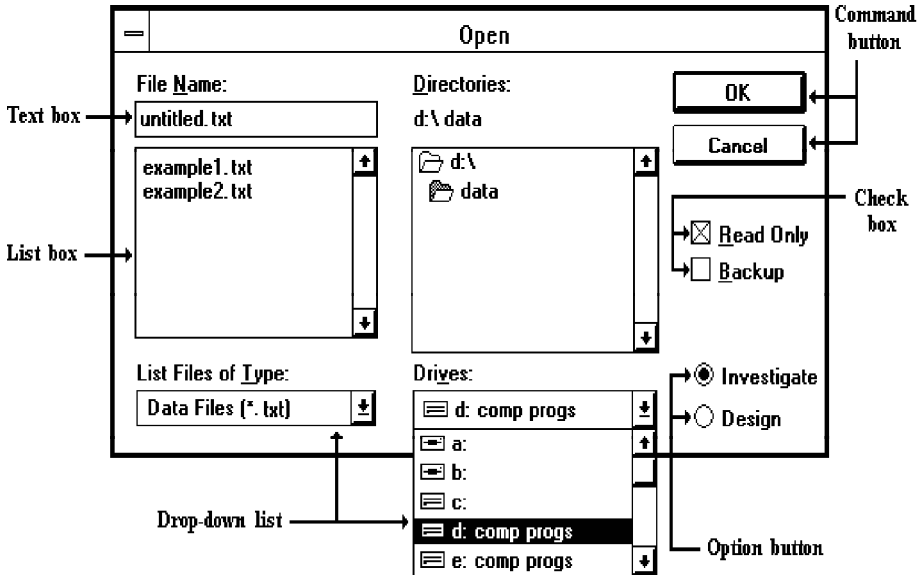
The following menu conventions are used in PCABEAM.

<u>Menu Convention</u>	<u>Meaning</u>
Dimmed (or not visible) command	You may not be able to use the command at the current time.
An ellipsis (...) following a command	A dialog box appears when this command is chosen. The dialog box contains options that need to be completed by the user.
A check mark (✓) next to a command	The command is in effect. To remove the check mark (by selecting the command), the command is no longer in effect.
A triangle (▶) next to a command	When this command is chosen, a cascading menu appears listing additional commands.

Working With a Dialog Box

A dialog box is displayed to request or supply information. An ellipsis (...) after a menu command indicates a dialog box will appear when selected. For example, selecting the **Open** command from the **File** menu causes the OPEN dialog box to appear.

There are several types of dialog box options. To move from one option to another, press **TAB** (move to next option) or **SHIFT + TAB** (move to previous option).



Command button

A Command button (or Push button) is labeled to do exactly what it says. The **OK** and **CANCEL** buttons are common command buttons. After you have completed your input within a dialog box has been completed, click once with the left-mouse button on the **OK** button or press **ALT + O** to save the changes. If, while working in a dialog box, you want to disregard the data you have input, click on the **CANCEL** button or press **ALT + C** to exit and disregard the changes.

Option button

An Option button (or Radio button) is one of a group of related options. Select only one option in a group at a time. When an option is selected, it contains a black dot. To select an option with the mouse, click the option button; using the keyboard, press TAB to move to the group of options and use the arrow keys to select the button.

Check box

A 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. Using the mouse, click a blank check box to select it and click a selected check box to clear it. Using the 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

A 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 typed starts at the insertion point. If the box already contains text, this text is selected, and any text typed replaces it. 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 to 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 and click on the item. Using the keyboard, use the arrow keys to scroll to the item you want and press ENTER.

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. Select another item in a similar way as in using a list box.

Working With a List Box

A list box displays a list of items. If there are more items than can fit in the box, scroll bars are provided to move quickly through the list.

The scroll bar is usually located to the right (vertical scroll bar) or at the bottom (horizontal scroll bar) of a list box. It has two directional arrows at either end (up and down, or left and right) in addition to a small rectangle called the scroll box. The location of the scroll box with respect to the length of the bar indicates the proportion of the amount of information in view to the whole list.

To scroll through the information displayed in a list box one line at a time, click the up or down scroll arrow, or, using the keyboard, press the arrow key that points in the direction you want to scroll. Another way is to click inside the scroll bar above or below the scroll box. This scrolls the list one screen at a time (the keyboard equivalent is pressing the PGUP or PGDN key). If you drag the scroll box in the scroll bar, the section of the list that moves into view depends on where you position the scroll 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 on the item.
- Using the keyboard, use the directional arrow keys to scroll to the item.

To select multiple sequential items in a list:

- Click the first item you want, and drag the cursor to the last item you want. Alternatively, click on the first item you want, press and hold down the SHIFT key, and click on the last item you want. 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. 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.

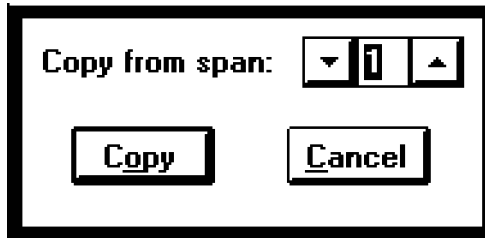
To select multiple nonsequential items in a list:

- Press and hold down the CTRL key, and click each item you want. To cancel a

selection, press and hold down the CTRL key, and click the item again. There are no keyboard equivalents for this procedure.

Using the COPY FROM command button:

- Some dialog boxes (such as the BEAMS & SLABS GEOMETRY, COLUMNS GEOMETRY, and SURFACE LOADS, etc.) have a COPY FROM command button. This button is provided to allow copying of data from one item to another in a list box.
- Select the item (or items) to copy data to by using one of the selection methods explained above.
- Choose the COPY FROM button. A dialog box appears.



- In the text box, enter the number of the item to copy from or use the up or down arrows provided on each side of the text box to cycle through available items.
- Press ENTER or choose COPY. The dialog box is closed and the list box is updated to reflect the changes.

For example, you have six columns and they all have the same height and geometry. Enter the data needed for the first column, select the second through fifth columns from the list box, choose the COPY FROM button, type 1 in the text box, and press ENTER. The first column's data are copied to the rest of the columns.

Working With View Windows

After inputting the data, you may view a plan, elevation, or isometric view of the model, as well as the applied surface and partial loads. The default open view at startup is the Plan view. All other views are displayed as icons near the bottom of the PCABEAM window.

The Isometric View is a three-dimensional view of the model rotated -45 degrees about the X-axis and 45 degrees about the Z-axis. This viewing angle may be changed from the **View/Preferences** command. The Plan View is a view looking down at the model whereas, the Elevated View is looking at the model from the side.

The Surface Loads and Partial Loads windows give a scaled graphical representation of the applied loads on all spans.

There may be more View windows displayed for a particular problem depending on the shear, moment, or deflection diagrams you requested to view.

To select a View window or icon:

- Click on any part of the window or icon to select it. Using the keyboard, press CTRL+TAB to cycle through all open windows and icons, or, select the window from the **Window** menu. If there are more than nine windows in the menu, a **More Windows** command appears. Selecting this command displays a list to choose from.

To open a View window:

- Double click on the icon that represents the View window you want opened. Using the keyboard, open the icon's control-menu box by pressing ALT+- (hyphen) and select **Restore**.

To minimize a View window to an icon:

- Click on the window's Minimize button (the down arrow located on the upper right hand corner of the window). Using the keyboard, open the windows Control-menu box by pressing ALT+- (hyphen) and select **Minimize**.

To maximize a View window:

- Click on the window's Maximize button (the up arrow located on the upper right hand corner of the window). This maximizes the window to occupy the full screen. Using the keyboard, open the windows Control-menu box by pressing ALT+- (hyphen) and select **Maximize**. To restore the window to its size, click on the Restore button or select **Restore** from the windows Control-menu box.

To close a View window:

- From the windows Control-menu box, select **Close**. Plan View, Elevation View, Isometric View, Surface loads, and Partial loads windows may not be closed. Selecting the **Close** command will minimize these windows to icons.

Manipulating View windows:

When you have too many open View windows, they may overlap each other making it difficult to locate a particular window. You can manipulate the windows in the following ways.

- From the **Windows** menu, select **Cascade**. This layers all open View windows one on top of the other with the title bar of each visible.
- From the **Windows** menu, select **Tile All Windows**. This divides the screen into equal parts and displays all open windows without overlapping.
- From the **Windows** menu, select **Tile View Windows**. The three View windows (Plan, Elevated, and Isometric Views) arrange on the screen without overlapping.

To learn more about resizing, moving, shrinking, enlarging, and closing windows, refer to *Working with Windows* in the Microsoft Windows Manual.

PCABEAM Menu Descriptions

PCABEAM uses pull-down menus to access commands that allow input of the properties, geometry, reinforcement, loading, manipulate the look of the program window, ultimately run the program, review, and print the results. This section gives a summary of each command in all the pull-down menus.

The File Menu

The **File** menu is used for opening, saving, retrieving data files, printing results, and exiting the program.

New

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

Open

Loads an existing PCABEAM data file.

Save

Saves the changes made to the current data file under that same filename. The new data overwrites the old data, and the old data cannot be retrieved. The **Save** command will only be available if the file has been modified since the last file save. If a data file is untitled, the **SAVE AS** dialog box appears.

Save As

Allows you to name or rename a data file. Use **Save As** when you want to save both the original data and any changes you've currently made to the data. The original data remains under the old filename. If a file of the same name exists, the program asks if you would like to overwrite the file.

Revert

Discards any changes made to the data and returns to the version of data saved in the original file. This option is only available if the file has been modified since the last file save and the file has been saved prior to editing.

File
New
Open...
Save
Save As...
Revert
Print Results...
Print Screen...
Exit

Print Results

Allows you to send the results of a successful run to the active printer.

Print Screen

Allows you to prints the contents of any view window (Plan, Elevated, or Isometric View, Partial Loads, Diagrams, etc.).

Exit

Ends the PCABEAM session and returns to the Program Manager. If any changes were made to the data and have not been saved, PCABEAM prompts you whether you want to save or abandon any changes made before you exit.

The Input Menu

The **Input** menu includes commands that enable you to enter all the data needed for a particular problem such as properties, geometry, reinforcement, and loads.

Auto Input

This command is designed to make the input process easier. By selecting **Auto Input**, a logical sequence of dialog boxes is automatically displayed.

Input	
Auto Input	
L abels...	
G eneral I nformation...	^I
M aterial P roperties...	^M
G eometry	▶
R einforcement	▶
L oads	▶
M oment R edistribution...	

Labels

Allows you to enter identifying labels for your input file. This information will be printed with the output and is intended to help you organize your data.

General Information

Allows you to input the number of spans and select the units of measure, the code to conform to, the run option (design or investigation), moment redistribution option, and the frame type and exposure.

Material Properties

Allows you to input the material properties for concrete and reinforcement.

Geometry

Contains a cascading menu with commands that allow you to define the frame geometry.

G eometry	B eams & S labs... ^B
	C olumns... ^C

Beams & Slabs

Allows you to input the span lengths and widths as well as the section geometry for each span.

Columns

Allows you to input the length, section geometry, and end fixities for the columns above and below the design floor.

Reinforcement

Contains a cascading menu with commands that allow you to input reinforcement data.

Reinforcement

Design Parameters...

Top Continuous Bars...
Top Discontinuous Bars...
Bot Continuous Bars...
Bot Discontinuous Bars...
Stirrups...

Design Parameters

Allows you to input parameters that will set the criteria for designing the continuous beam system. *This command is available for the Design option only.*

Top Continuous Bars

Allows you to input top flexural bars that are continuous throughout the span. *This command is available for the Investigation option only.*

Top Discontinuous Bars

Allows you to input top flexural bars that are not continuous throughout the span. *This command is available for the Investigation option only.*

Bot Continuous Bars

Allows you to input bottom flexural bars that are continuous throughout the span. *This command is available for the Investigation option only.*

Bot Discontinuous Bars

Allows you to input bottom flexural bars that are not continuous throughout the span. *This command is available for the Investigation option only.*

Stirrups

Allows you to input the transverse (shear) reinforcement. *This command is available for the Investigation option only.*

Loads

Contains a cascading menu with commands that allow you to enter the applied loads on the frame. Load factors may also be modified.

Loads**F**actors & Patterns...**S**urface...**P**artial **D**ead Loads...**P**artial **L**ive Loads...**L**ateral **M**oments...**Factors & Patterns**

Allows you to modify the dead, live, and lateral load factors and specify the load patterns to be used.

Surface

Allows you to input unfactored dead and live surface loads (force per unit area) for each span.

Partial Dead Loads

Allows you to input uniform (force per unit length), concentrated (point), and trapezoidal dead loads on any span.

Partial Live Loads

Allows you to input uniform (force per unit length), concentrated (point), and trapezoidal live loads on any span.

Lateral Moments

Allows you to input concentrated moments resulting from a lateral load analysis applied at each joint.

Moment Redistribution

Allows you to input percentages of the moments to be considered in moment redistribution.

The Design Menu

The **Design** menu contains commands that enable you to select the output options, perform the design or investigation, and pass necessary information to the PCACOL program for column design.

Design

Output Options...

Solve...

PCACOL...

Output Options

Allows you to suppress portions of the output. PCABEAM prints or outputs the selected items only. To take effect, the solver must be executed after selecting this command.

Solve

Executes the solver portion of PCABEAM and sends the output to the results file that may be viewed and/or printed.

PCACOL

Transfers data to and executes the PCACOL program to design any column that is part of the input frame. *This command will not be available if the PCACOL program is not found on the system.*

The View Menu

The **View** menu commands enable you to control what is displayed on the screen to suit your viewing needs, view the results file, and view results diagrams.

Redraw

Redraws (or refreshes) the View windows and resets the zoom.

Scale

Shows or hides a reference scale along the left and top edges of the three View windows.

Axis

Shows or hides axis representation in the bottom left-hand corner of the three View windows.

Zoom

Enables you to zoom in (enlarge) on any portion of the displayed model for closer verification.

Preferences

Allows you to control what is to be displayed in the windows and specify the angle at which to view the Isometric View.

Results File

Allows viewing of the results after a successful run.

Shear & Moment Diagrams

Allows you to view the shear and moment diagrams for a specific loading pattern and a specific span after a successful run.

View

Redraw

Scale

Axis

Zoom

Preferences...

Results File...

Shear/**M**oment Diagram...

Envelope/**C**apacity Diagram...

Deflection Diagram...

Envelope & Capacity Diagrams

Allows you to view the shear and moment envelopes and capacity diagrams for a specific span after a successful run.

Deflection Diagrams

Allows you to view the deflection diagram of a specific span after a successful run.

The Options Menu

The **Options** menu allows you to change the window colors, the startup options of PCABEAM, and the page layout.

Color

Allows you to select different color settings for the window background, displayed members, and displayed loads.

Default Units & Code

Allows you to change the startup default units and code that were set during the installation of the PCABEAM program. The choices set here become defaults the next time PCABEAM is executed.

Default Data Directory

Allows you to modify the startup data directory that was set during the installation of the PCABEAM program.

Page Layout

Allows you to change the page margins.

Options
Color...
Default Units/Code...
Default Data Directory...
Page Layout...

The Window Menu

The **Window** menu allows you to manipulate the different view windows.

Cascade

Displays all open windows in the same size, arranging them on top of each other so that the Title bar of each is visible.

Tile All Windows

Arranges all open windows so that no window overlaps the other.

Tile View Windows

Arranges all open View windows so that no window overlaps the other.

Arrange Icons

Aligns all icons at the bottom of the PCABEAM window.

Remaining Commands...

The remaining menu commands is a list of the windows that are available for viewing.

Window

Cascade

Tile All Windows

Tile View Windows

Arrange Icons

1 Plan View

2 Elevated View

3 Isometric View

4 Surface Loads

5 Partial Loads

The Help Menu

The **Help** menu includes commands that enable you to obtain on-line help for the PCABEAM program. The help screens in the programs are closely related with Chapters 3 and 4 of this manual.

Index

Lists all the available help topics. Click on any underlined topic and a help screen appears with the information about that item. More information on the help system can be obtained from the Microsoft Windows manual.

Commands

Lists all the available commands found in the PCABEAM program, similar to this chapter. Click on any underlined command and a help screen appears with information about that command.

Procedures

Lists several procedures on operating PCABEAM, similar to Chapter 4. Click on any underlined command and a help screen appears with information about that procedure.

Using Help

Provides help on using the Microsoft Windows help system.

About PCABEAM

Displays a dialog box that gives you copyright information, the version number of the PCABEAM program, your firm name, city, and state.

Help**Index****Commands****Procedures****Using Help****About PCABEAM...**

Chapter 4

Operating the Program

This chapter is divided into sections that follow the order that commands and options appear under the menu items.

Working With the File Menu Commands

Creating a New Data File

When PCABEAM 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.

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

Selecting the **New** command automatically executes the **Auto Input** command.

Opening an Existing Data File

- From the **File** menu, select **Open**. The dialog box of Figure 4-1 appears.

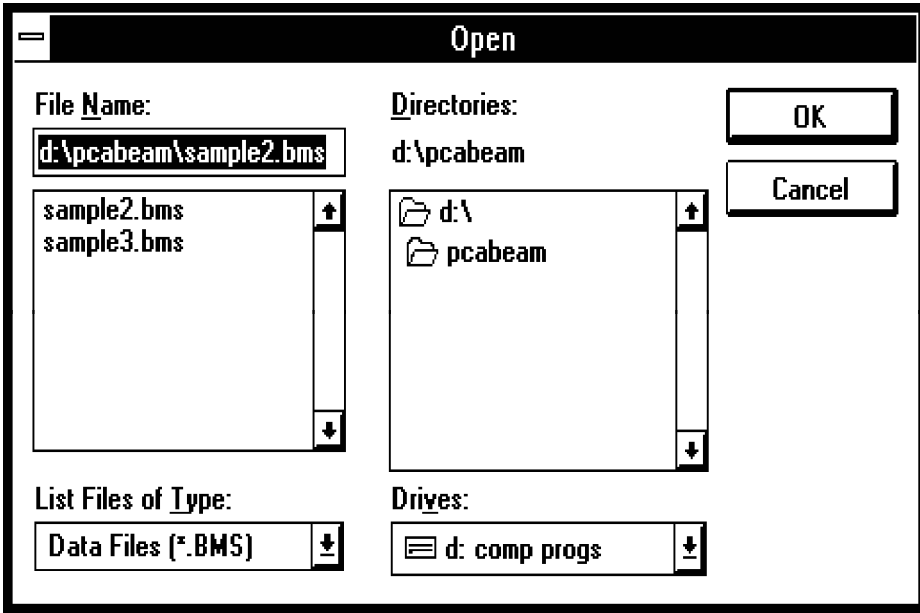


Figure 4-1—The Open dialog box

- All files with a BMS 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 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 type its name in the text box.
- Press ENTER or choose the OK button.

Saving the Data

In PCABEAM you can save the data with the same filename or with a new filename.

To save your data with the same filename:

- If you are editing a data set that has previously been saved under a filename, select the **Save** command from the **File** menu to save the changes to the same file.

To give your data a new or different filename:

- From the **File** menu, select **Save As**. The dialog box of Figure 4-2 appears. All files with a BMS extension contained in the current drive and directory are displayed in the FILE NAME list box.

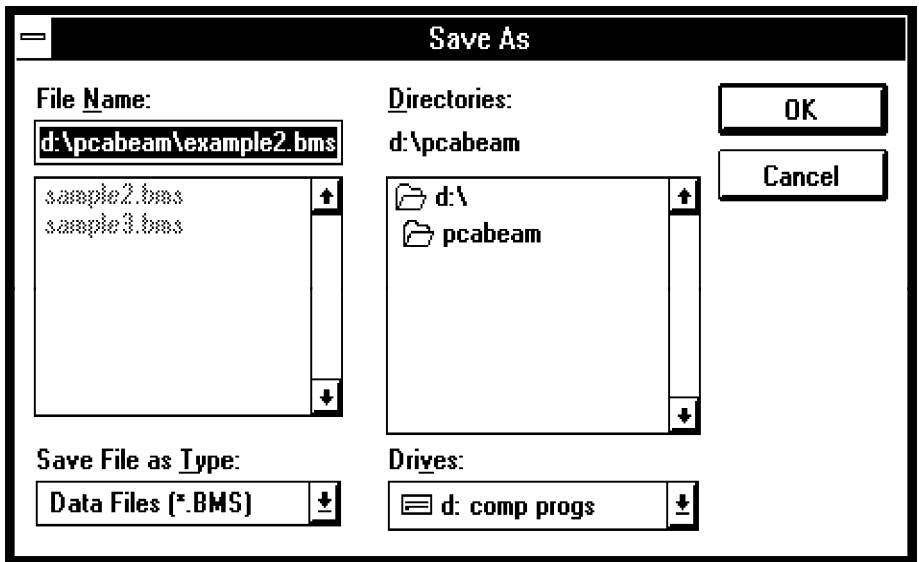


Figure 4-2—The Save As dialog box

- 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 list box, type a new filename. You do not have to supply a file extension, .BMS will be appended to the specified file.
- Press ENTER or choose the OK button.

Printing the Results

Once a successful run has been performed, you may print the results file. PCABEAM sends the results to the active printer set in the Control Panel. (Refer to your Microsoft Windows manual on using the Control Panel.)

- From the **File** menu, select **Print Results**. The dialog box of Figure 4-3 appears. The options in this dialog box may vary depending on the installed printer.

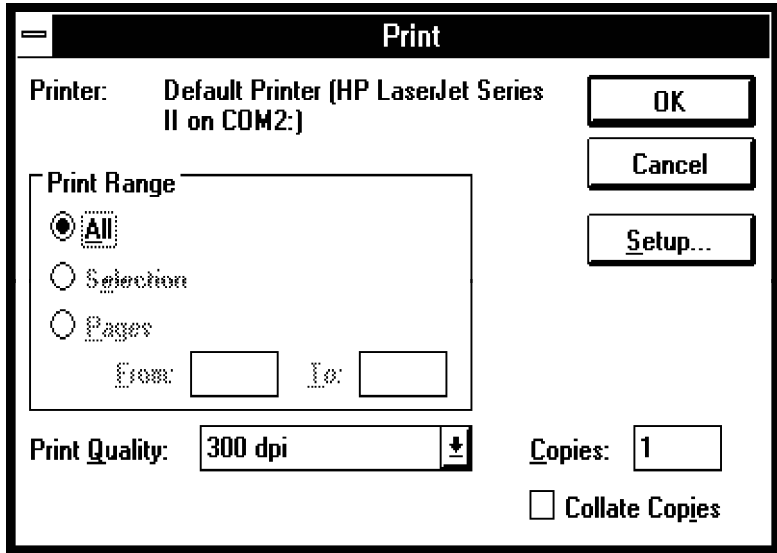


Figure 4-3—The Print dialog box

- Choose OK to start printing. A message box appears informing you that the results file is being printed. Choose the CANCEL button to terminate the print job.

Printing the Screen

- From the **File** menu, select **Print Screen**. The dialog box of Figure 4-4 appears.

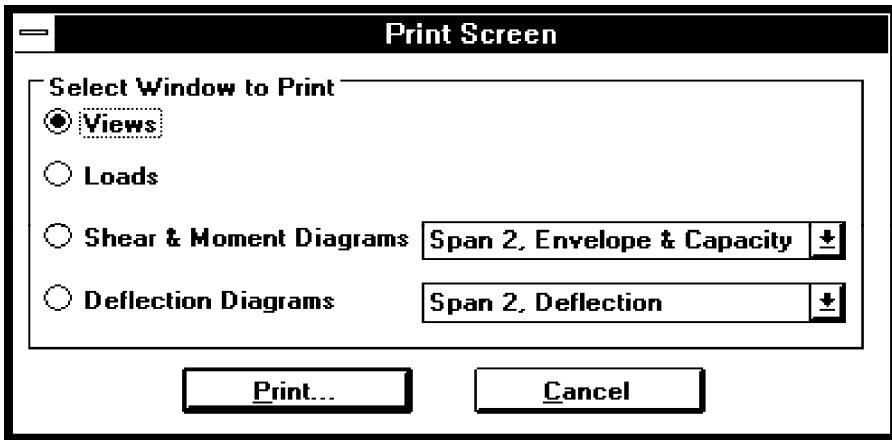


Figure 4-4—The Print Screen dialog box

- Choose **VIEWS** to print the three views (Plan, Elevation, and Isometric).
- Choose **LOADS** to print the surface, partial dead, and partial live loads.
- Choose **SHEAR & MOMENT DIAGRAMS** to print any of the diagrams. Open the drop-down list to select the diagram to be printed.
- Choose **DEFLECTION DIAGRAMS** to print any of the deflection diagrams. Open the drop-down list to select the diagram to be printed.
- Press **ENTER** or choose the **PRINT** button. The dialog box of Figure 4-3 appears. The options in this dialog box may vary depending on the installed printer.
- Choose **OK** to start printing. A message box appears informing you that the views are being printed. Choose the **CANCEL** button to terminate the print job.

Working With the Input Menu Commands

Auto Input

PCABEAM has an automatic input feature that walks you through the input process in a logical order. Selecting the **Input/Auto Input** command opens the necessary dialog boxes to complete the input of a problem.

- From the **Input** menu, select **Auto Input**.
- The **LABELS** dialog box is the first to appear in an input session. Complete the input fields in this box and choose the **Ok** button.
- Consequently, other dialog boxes appear in the order they are displayed in the **Input** menu. Complete the input needed in each dialog box and choose the **Ok** button.
- To abort **Auto Input**, choose the **CANCEL** button in any active dialog box. A message box informing you that **Auto Input** is aborted appears. Press **ENTER** or choose **Ok**.

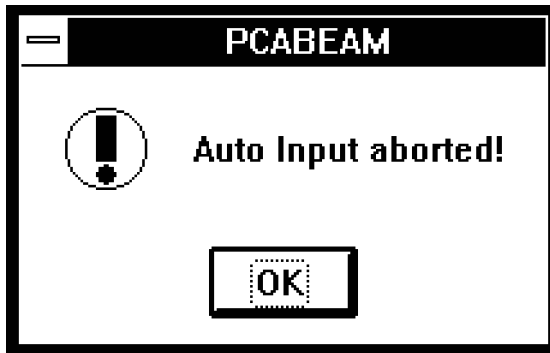
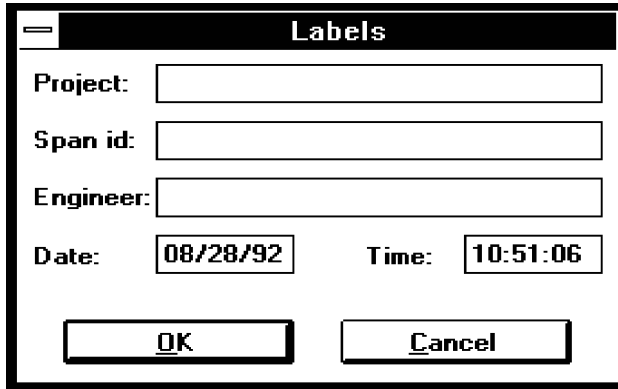


Figure 4-5

Inputting Labels

To keep track of data and output, PCABEAM allows you to specify the project id, span id, engineer, date, and time (date and time are automatically input when a new data file is started). These labels are printed with the output and are intended to help organize data.

- From the **Input** menu, select **Labels**. The dialog box of Figure 4-6 appears.



The image shows a dialog box titled "Labels". It has a standard window title bar with a close button on the left. The dialog contains five input fields arranged vertically. The first three are "Project:", "Span id:", and "Engineer:", each followed by an empty rectangular text box. The fourth is "Date:" followed by a text box containing "08/28/92". The fifth is "Time:" followed by a text box containing "10:51:06". At the bottom of the dialog are two buttons: "OK" on the left and "Cancel" on the right.

Figure 4-6—The Labels dialog box

- Enter the labels in the corresponding text boxes.
- Press ENTER or choose the OK button.

Using the General Information Command

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

- From the **Input** menu, select **General Information**. The dialog box of Figure 4-7 appears.

General Information

Number of spans:

Units

U.S. in-lb
 SI metric

Moment redistribution

No redistribution
 Input redistribution

Code

ACI 318-89
 CAN3.A23.3-M84

Frame type

Interior
 Spandrel

Run option

Design
 Investigation

Frame exposure

Interior
 Exterior

Figure 4-7—The General Information dialog box

- In the NUMBER OF SPANS text box, enter the total number of spans, including the two cantilevers on each end. This number must be between 3 and 14 (inclusive).
- From the UNITS options group, select the units of measure. US IN-LB and SI METRIC are the two choices in PCABEAM. The SI METRIC option uses metric reinforcement.
- From the CODE options group, select the building standard to conform to: ACI 318-89 or CAN3-A23.3-M84.
- From the RUN OPTION options group, select the run option you want the program to perform, DESIGN or INVESTIGATION.
- From the MOMENT REDISTRIBUTION options group, specify whether you want PCABEAM to perform moment redistribution or not, by selecting the corresponding option.
- From the FRAME TYPE options group, specify whether the frame is INTERIOR or SPANDREL. Note that spandrel (exterior) beams have different integrity reinforcement requirements (ACI Code only).
- From the FRAME EXPOSURE options group, specify INTERIOR or EXTERIOR exposure type. This option affects the exposure factor, z .
- Press ENTER or choose the OK button.

Note: If, after inputting data and executing the solver, you change the run option from Design to Investigation, PCABEAM will save the design problem data and start a new file with the same data. You will be prompted to use the design results for investigation input.

Inputting Material Properties

The **Material Properties** command allows you to input the concrete and reinforcing steel material properties.

- From the **Input** menu, select **Material Properties**. The dialog box of Figure 4-8 appears.

Material Properties			
Concrete			
	Beams	Columns	
Unit weight:	<input type="text" value="C"/>	<input type="text" value="0"/>	pcf
Type:	Lt Wt	Lt Wt	
Compressive strength, f_c :	<input type="text" value="0"/>	<input type="text" value="0"/>	ksi
Tensile strength, f_{ct} :	<input type="text" value="0.0"/>	<input type="text" value="0.0"/>	psi
Modulus of rupture, f_r :	<input type="text" value="0.0"/>	<input type="text" value="0.0"/>	psi
Reinforcement			
	Bars	Stirrups	
Yield strength, f_y :	<input type="text" value="60"/>	<input type="text" value="60"/>	ksi
<input type="checkbox"/> Epoxy-coated bars			
<input type="button" value="OK"/>		<input type="button" value="Cancel"/>	

Figure 4-8—The Material Properties dialog box

- Enter the concrete unit weight for beams and columns in the corresponding text boxes. Note that the concrete type (light, sand, or normal weight) is displayed under the text boxes.
- Enter the concrete compressive strength, f_c , for beams and columns, in the corresponding text boxes. The concrete tensile strength, f_{ct} , and modulus of rupture, f_r , are automatically computed and displayed in their corresponding text boxes.

- Enter the concrete tensile strength, f_{ct} , (if different than the computed value) for beams and columns in the corresponding text boxes.
- Enter the concrete modulus of rupture, f_r , (if different than the computed value) for beams and columns in the corresponding text boxes.
- Enter the reinforcing steel yield strength, f_y , for bars and stirrups in the corresponding text boxes.
- Select or clear the EPOXY-COATED BARS check-box option. If this option is selected (an X in its box), then the bars will be considered epoxy-coated. This option affects the development length calculations (ACI Code).
- Press ENTER or choose the OK button.

Inputting Beams and Slabs Geometry

- From the **Input** menu, select **Geometry**. From the cascading menu, select **Beams & Slabs**. The dialog box of Figure 4-9 appears. All input fields, except span numbers, are initially set to zero.

Span no.	Span Length	Beam Width	Beam Depth	Slab Thick.	Slab Width Left	Slab Width Right
1	0.0	0.0	0.0	0.0	0.0	0.0
2	0.0	0.0	0.0	0.0	0.0	0.0
3	0.0	0.0	0.0	0.0	0.0	0.0
4	0.0	0.0	0.0	0.0	0.0	0.0
5	0.0	0.0	0.0	0.0	0.0	0.0
6	0.0	0.0	0.0	0.0	0.0	0.0
7	0.0	0.0	0.0	0.0	0.0	0.0

Figure 4-9—The Beams & Slabs Geometry dialog box

- In the **SPAN NO.** text box, enter the number of the span whose geometry you want to input. The highlight in the list box moves to the span specified and that span information is displayed in the remaining text boxes.
- In the **LENGTH** text boxes, enter the span length. Span length is measured from column centerline to column centerline for interior spans, and from slab edge to column centerline for cantilevers.

- In the **BEAM** group, enter the beam width and depth in the corresponding text boxes. Beam depth is measured from the top of the slab. For one-way slab systems, these values are zero.
- In the **SLAB** group, enter the slab thickness and its widths (in the transverse direction) to the left and to the right of the column centerline, in the corresponding text boxes. For rectangular beam sections, these values are zero. For T- or L-beam sections, the slab widths must be greater than or equal to half the beam width.
- Press **ENTER** or choose the **REPLACE** button. The list box at the right is updated.
- Repeat the four steps above for each span. You may use the **COPY FROM** button as a shortcut (see *Working With a Dialog Box*).
- Choose the **OK** button.

Inputting the Columns Geometry

- From the **Input** menu, select **Geometry**. From the cascading menu, select **Columns**. The dialog box of Figure 4-10 appears. All input fields, except column numbers, are initially set to zero.

Column no	loc	c ₁ (in)	c ₂ (in)	Height (ft)	Stiff. Factor
1	Abv	0.0	0.0	0.0	100.0
	Blw	0.0	0.0	0.0	100.0
2	Abv	0.0	0.0	0.0	100.0
	Blw	0.0	0.0	0.0	100.0
3	Abv	0.0	0.0	0.0	100.0
	Blw	0.0	0.0	0.0	100.0
4	Abv	0.0	0.0	0.0	100.0
	Blw	0.0	0.0	0.0	100.0
5	Abv	0.0	0.0	0.0	100.0
	Blw	0.0	0.0	0.0	100.0
6	Abv	0.0	0.0	0.0	100.0
	Blw	0.0	0.0	0.0	100.0

Figure 4-10—The Columns Geometry dialog box

- In the COLUMN NO. text box, enter the number of the column whose geometry you want to input. The highlight in the list box moves to the column specified and that column information is displayed in the remaining text boxes.
- In the DEPTH, C1 text box under ABOVE, enter the depth of the column above the design floor. Column depth is measured in the direction of analysis. For round columns, enter the column diameter.

- In the WIDTH, c2 text box under ABOVE, enter the width of the column above the design floor. Column width is measured perpendicular to the direction of analysis. For round columns, enter zero.
- In the HEIGHT text box under ABOVE, enter the height of the column above the design floor. This is the distance from the top of the design floor to the top of the floor above.
- In the DEPTH, c1 text box under BELOW, enter the depth of the column below the design floor. Column depth is measured in the direction of analysis. For round columns, enter the column diameter.
- In the WIDTH, c2 text box under BELOW, enter the width of the column below the design floor. Column width is measured perpendicular to the direction of analysis. For round columns, enter zero.
- In the HEIGHT text box under BELOW, enter the height of the column below the design floor. This is the distance from the top of the design floor to the top of the floor below.
- In the STIFFNESS REDUCTION FACTOR text box, enter a factor (in percentage) by which the column stiffness will be reduced. The default is 100% (full column stiffness is considered). A zero value indicates *apin* support, and, for exterior supports, 999 indicates full fixity.
- Press ENTER or choose the REPLACE button. The list box at the right is updated.
- Repeat the nine steps above for each column. You may use the COPY FROM button as a shortcut (see *Working With a Dialog Box*).
- Choose the OK button.

Inputting the Reinforcement Design Parameters

The **Input/Reinforcement/Design Parameters** command allows you to input parameters that govern the design. *This command is available for the Design option only.*

- From the **Input** menu, select **Reinforcement**. From the cascading menu, select **Design Parameters**. The dialog box in Figure 4-11 appears.

	Top Bars	Bottom Bars	
Minimum bar size:	4	4	
Maximum bar size:	14	14	
Clear vertical cover:	1.5	1.5	in
Clear side cover:	1.5	1.5	in
Min. clear spacing:	1	1	in
Max. center-to-center spacing:	18	18	in

Stirrups	
Minimum	Maximum
Size:	3 5
Spacing:	2 24 in
<input type="radio"/> Open <input checked="" type="radio"/> Closed	

OK Cancel

Figure 4-11—The Reinforcement Design Parameters dialog box

- In the **MINIMUM BAR SIZE** text boxes, enter the minimum bar sizes for top and bottom longitudinal (flexural) bars.
- In the **MAXIMUM BAR SIZE** text boxes, enter the maximum bar sizes for top and bottom longitudinal (flexural) bars. Maximum bar size must be smaller than or equal to minimum bar size.
- In the **CLEAR VERTICAL COVER** text boxes, enter the clear concrete vertical cover to the top and bottom longitudinal (flexural) bars.
- In the **CLEAR SIDE COVER** text boxes, enter the clear concrete horizontal (side) cover to the top and bottom longitudinal (flexural) bars. This will be used by the program when trying to fit the bars within the section.

- In the *MIN. CLEAR SPACING* text boxes, enter the minimum clear spacing between bars for the top and bottom longitudinal (flexural) bars. This will be used by the program when trying to fit the bars within the section.
- In the *MAX. CENTER-TO-CENTER SPACING* text boxes, enter the maximum center-to-center spacing between bars for the top and bottom longitudinal (flexural) bars. This will be used by the program when trying to fit the bars within the section.
- In the *SIZE* text boxes, enter the minimum and maximum bar sizes for transverse reinforcement (stirrups).
- In the *SPACING* text boxes, enter the minimum and maximum center-to-center spacing between stirrups.
- Specify whether stirrups are open or closed by selecting the corresponding option. This option affects the integrity reinforcement requirements (ACI Code only).
- Press *ENTER* or choose the *OK* button.

Inputting Top Continuous Bars

The **Input/Reinforcement/Top Continuous Bars** command allows you to define sets of continuous top longitudinal bars in any span. *This command is available for the Investigation option only.*

- From the **Input** menu, select **Reinforcement**. From the cascading menu, select **Top Continuous Bars**. The dialog box in Figure 4-12 appears.

Span no.	Layer Dist.	No. of bars	Left Size	Left Length	Right Size	Right Length	Center Size	Center Length	Start

Figure 4-12—The Top Continuous Bars dialog box

- In the **SPAN NO.** text box, enter the number of the span you want to specify bars for.
- In the **LAYER DISTANCE** text box, enter the distance from the top of the beam to the centroid of the bar set.
- In the **NO. OF BARS** text box, enter the total number of bars in the set.

- From the SET TYPE options group, select one of the types explained in Chapter 2 by selecting the corresponding option¹.
- If you selected SINGLE BAR set type, enter the bar size in the corresponding text box. The bar length will default to the span length.
- If you selected TWO-LAPPED BARS set type, enter the size and length of the left and right bars in the corresponding text boxes. The sum of the left and right bar lengths must be greater than or equal to the span length.
- If you selected THREE-LAPPED BARS set type, enter the size and length of the left and right bars along with the size, length, and start distance of the center bar in the corresponding text boxes. The sum of the left, right, and center bar lengths must be greater than or equal to the span length.
- Choose the INSERT button. The data are added to the list box at the bottom.
- Repeat the seven steps above for each continuous bar set you have in the same span. You may define up to five bar sets in a span. To input bar sets for another span, repeat the eight steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

¹ For cantilevers, you may specify a single-bar set only.

Inputting Top Discontinuous Bars

The **Input/Reinforcement/Top Discontinuous Bars** command allows you to define sets of discontinuous top longitudinal (flexural) bars in any span. *This command is available for the Investigation option only.*

- From the **Input** menu, select **Reinforcement**. From the cascading menu, select **Top Discontinuous Bars**. The dialog box in Figure 4-13 appears.

Span no.	Layer distance:	in	Left:	No. of bars	Size	Length (ft)	Right:	No. of bars	Size	Length (ft)
1	0		0	0	0	0	0	0	0	0

Span no.	Layer Distance	Left	Right				
No.	Distance	No.	Size	Length	No.	Size	Length

Figure 4-13—The Top Discontinuous Bars dialog box

- In the **SPAN NO.** text box, enter the number of the span you want to specify bars for.
- In the **LAYER DISTANCE** text box, enter the distance from the top of the beam to the centroid of the bar set.

- In the LEFT¹ text boxes, enter the number of bars, bar size, and bar length, under the corresponding titles. Bar length specified here is measured from the centerline of the left support.
- In the RIGHT² text boxes, enter the number of bars, bar size, and bar length, under the corresponding titles. Bar length specified here is measured from the centerline of the right support.
- Choose the INSERT button. The data are added to the list box at the bottom.
- Repeat the four steps above for each discontinuous bar set you have in the same span. You may define up to five bar sets in a span. To input bar sets for another span, repeat the five steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

1 Not applicable for left cantilever.

2 Not applicable for right cantilever.

Inputting Bottom Continuous Bars

The **Input/Reinforcement/Bot Continuous Bars** command allows you to define sets of continuous bottom longitudinal (flexural) bars in any span. *This command is available for the Investigation option only.*

- From the **Input** menu, select **Reinforcement**. From the cascading menu, select **Bot Continuous Bars**. The dialog box in Figure 4-14 appears.

Span no.	Layer Dist.	No. of bars	Left Size	Left Length	Right Size	Right Length	Center Size	Center Length	Start

Figure 4-14—The Bottom Continuous Bars dialog box

- In the **SPAN NO.** text box, enter the number of the span you want to specify bars for.
- In the **LAYER DISTANCE** text box, enter the distance from the bottom of the beam to the centroid of the bar set.

- In the NO. OF BARS text box, enter the number of bars in the set.
- From the SET TYPE options group, select one of the types explained in Chapter 2 by selecting the corresponding option¹.
- If you selected SINGLE BAR set type, enter the bar size in the corresponding text box. The bar length will default to the span length.
- If you selected TWO-LAPPED BARS set type, enter the size and length of the left and right bars in the corresponding text boxes. The sum of the left and right bar lengths must be greater than or equal to the span length.
- If you selected THREE-LAPPED BARS set type, enter the size and length of the left and right bars along with the size, length, and start distance of the center bar in the corresponding text boxes. The sum of the left, right, and center bar lengths must be greater than or equal to the span length.
- Choose the INSERT button. The data are added to the list box at the bottom.
- Repeat the seven steps above for each continuous bar set you have in the same span. You may define up to five bar sets in a span. To input bar sets for another span, repeat the eight steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

1 For cantilevers, you may specify a single-bar set only.

Inputting Bottom Discontinuous Bars

The **Input/Reinforcement/Bot Discontinuous Bars** command allows you to define sets of discontinuous bottom longitudinal (flexural) bars in any span. *This command is available for the Investigation option only.*

- From the **Input** menu, select **Reinforcement**. From the cascading menu, select **Bot Discontinuous Bars**. The dialog box in Figure 4-15 appears.

Bottom Discontinuous Bars											
		No. of bars	Size	Length (ft)	Start (ft)						
Span no.:	<input type="text" value="1"/>	Left: <input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>							
Layer distance:	<input type="text" value="0"/> in	Right: <input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>							
		Center: <input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>						
Span no.	Layer Distance	----- Left -----			----- Right -----			----- Center -----			
		No.	Size	Length	No.	Size	Length	No.	Size	Length	Start

Figure 4-15—The Bottom Discontinuous Bars dialog box

- In the **SPAN NO.** text box, enter the number of the span you want to specify bars for.
- In the **LAYER DISTANCE** text box, enter the distance from the top of the beam to the centroid of the bar set.

- In the LEFT¹ text boxes, enter the number of bars, bar size, and bar length, under the corresponding titles. Bar length specified here is measured from the centerline of the left support.
- In the RIGHT² text boxes, enter the number of bars, bar size, and bar length, under the corresponding titles. Bar length specified here is measured from the centerline of the right support.
- In the CENTER³ text boxes, enter the number of bars, bar size, length, and start distance under the corresponding titles. Start distance is measured from the centerline of left support.
- Choose the INSERT button. The data are added to the list box at the bottom.
- Repeat the five steps above for each discontinuous bar set you have in the same span. You may define up to five bar sets in a span. To input bar sets for another span, repeat the six steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

1 Not applicable for left cantilever.

2 Not applicable for right cantilever.

3 Not applicable for left or right cantilever.

Inputting Beam Stirrups

The **Input/Reinforcement/Stirrups** command allows you to input sets of stirrups (transverse reinforcement) in any span. A stirrup set consists of a group of stirrups of equal size, spacing, and number of legs. You may specify up to six sets of stirrups per span. *This command is available for the Investigation option only.*

- From the **Input** menu, select **Reinforcement**. From the cascading menu, select **Stirrups**. The dialog box in Figure 4-16 appears.

Span No.	1st stir. loc.	Set No.	No. of stirrups	Size	No. of legs	Spacing

Figure 4-16—The Stirrups dialog box

- In the SPAN NO. text box, enter the number of the span you want to specify stirrups for.
- In the 1ST STIRRUP LOC text box, enter the distance from the face of left support¹ to the first stirrup.
- Enter the number of stirrups, stirrup size, and the number of legs per stirrup, in the corresponding text boxes.
- Choose the INSERT button. The data are added to the list box at the bottom.
- Repeat the two steps above for each stirrup set you have in the same span. You may define up to six stirrup sets in a span. To input stirrup sets for another span, repeat the four steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

1 For the left cantilever, this is measured from the cantilever edge.

Inputting Load Factors and Loading Patterns

The **Input/Loads/Factors & Patterns** command allows you to change the magnification factors applied to the loads. The default values depend on the Code selected using the **General Information** command.

- From the **Input** menu, select **Loads**. From the cascading menu, select **Factors & Patterns**. The dialog box in Figure 4-17 appears.

Load Factors & Patterns

Load Factors

$U_1 = 1.4 \text{ Dead} + 1.7 \text{ Live}$

$U_2 = 0.75 (1.4 \text{ Dead} + 1.7 \text{ Live} + 1.7 \text{ Lateral})$

$U_3 = 0.9 \text{ Dead} + 1.3 \text{ Lateral}$

Loading Patterns

Gravity	Gravity + Lateral
<input checked="" type="radio"/> 1 through 4	<input type="radio"/> 4 through 8
<input type="radio"/> 4 Only	<input type="radio"/> 1 through 8

Pattern live load factor (1 - 3): 75 %

OK **Cancel**

Figure 4-17—The Load Factors & Patterns dialog box

- Enter the value for the load factor you want to change in its text box. *Note that the dead and live load factors of the first load case, U1, are linked to the dead and live load factors of the second load case, U2; therefore, different factors for the dead and live loads cannot be input for the two cases.*
- From the LOADING PATTERNS options group, select the type of pattern(s) to be considered. For gravity loads only, you may select patterns 1 through 4 or pattern 4 (all spans loaded). For combined gravity and lateral loads, you may select patterns 4 through 8 or patterns 1 through 8. *You must select one of the options under GRAVITY + LATERAL to be able to input lateral load moments.*
- In the corresponding text box, enter the percentage of live load to be considered for pattern loading.
- Press ENTER or choose the OK button.

Inputting Surface Loads

- From the **Input** menu, select **Loads**. From the cascading menu, select **Surface**. The dialog box in Figure 4-18 appears.

Span no.	Loads (psf)	
	Dead	Live
1	0.0	0.0
2	0.0	0.0
3	0.0	0.0
4	0.0	0.0
5	0.0	0.0
6	0.0	0.0
7	0.0	0.0

Figure 4-18—The Surface Loads dialog box

- In the SPANNO. text box, enter the number of the span whose surface loads you want to input. The highlight in the list box moves to the span specified and that span loads are displayed in the remaining text boxes.
- Enter the dead and live surface loads in the corresponding text boxes.
- Press ENTER or choose the REPLACE button. The list box at the right is updated.
- Repeat the three steps above for each span you have. You may use the COPY FROM button as a shortcut (see *Working With a Dialog Box*).
- Choose the OK button.

Inputting Partial Dead Loads

- From the **Input** menu, select **Loads**. From the cascading menu, select **Partial Dead Loads**. The dialog box in Figure 4-19 appears.

Span no.	Load no.	Type	W_a	W_b	L_a	L_b

Figure 4-19—The Partial Dead Loads dialog box

- In the SPAN NO. text box, enter the number of the span whose surface loads you want to input.
- Select a partial load type, UNIFORM, CONCENTRATED, or TRAPEZOIDAL. The remaining text boxes change depending on the selected load type.

- If you select UNIFORM, enter the load magnitude, its start location, and its end location in the W_A , L_A , and L_B text boxes, respectively. Load start and end locations are measured from the centerline of the left support¹.
- If you select CONCENTRATED, enter the load magnitude and location (measured from the centerline of the left support¹) in the W_A and L_A text boxes, respectively.
- If you select TRAPEZOIDAL, enter the load magnitude on its left end, its magnitude on its right end, its start location, and its end location in the W_A , W_B , L_A , and L_B text boxes, respectively. Load start and end locations are measured from the centerline of the left support¹.
- Choose the INSERT button. The data are added to the list box on the right.
- Repeat the five steps above for each load you have on the span. To enter loads on another span, repeat the six steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

¹ For the left cantilever, L_A and L_B are measured from the cantilever edge.

Inputting Partial Live Loads

- From the **Input** menu, select **Loads**. From the cascading menu, select **Partial Live Loads**. The dialog box in Figure 4-20 appears.

Span no.	Load no.	Type	W_a	W_b	L_a	L_b

Figure 4-20—The Partial Live Loads dialog box

- In the SPAN NO. text box, enter the number of the span whose surface loads you want to input.
- Select a partial load type, UNIFORM, CONCENTRATED, or TRAPEZOIDAL. The remaining text boxes change depending on the selected load type.

- If you select UNIFORM, enter the load magnitude, its start location, and its end location in the W_A , L_A , and L_B text boxes, respectively. Load start and end locations are measured from the centerline of the left support¹.
- If you select CONCENTRATED, enter the load magnitude and location (measured from the centerline of the left support¹) in the W_A and L_A text boxes, respectively.
- If you select TRAPEZOIDAL, enter the load magnitude on its left end, its magnitude on its right end, its start location, and its end location in the W_A , W_B , L_A , and L_B text boxes, respectively. Load start and end locations are measured from the centerline of the left support¹.
- Choose the INSERT button. The data are added to the list box on the right.
- Repeat the five steps above for each load you have on the span. To enter loads on another span, repeat the six steps above.
- To edit an entry that has been added to the list box, select that entry from the list box, modify its values, and choose the REPLACE button.
- To delete an entry that has been added to the list box, select that entry from the list box and choose the DELETE button.
- Choose the OK button.

¹ For the left cantilever, L_A and L_B are measured from the cantilever edge.

Inputting Lateral Load Moments

The **Input/Loads/Lateral Moments** command is available only if you selected patterns 4 THROUGH 8 or 1 THROUGH 8 in the **FACTORS & PATTERNS** dialog box.

- From the **Input** menu, select **Loads**. From the cascading menu, select **Lateral Moments**. The dialog box in Figure 4-21 appears.

Joint no.	Direction	Left	Right	Top	Bottom
1	[->]	0.00	0.00	0.00	0.00
	[<-]	0.00	0.00	0.00	0.00
2	[->]	0.00	0.00	0.00	0.00
	[<-]	0.00	0.00	0.00	0.00
3	[->]	0.00	0.00	0.00	0.00
	[<-]	0.00	0.00	0.00	0.00
4	[->]	0.00	0.00	0.00	0.00
	[<-]	0.00	0.00	0.00	0.00

Figure 4-21—The Lateral Moments dialog box

- In the **JOINT NO.** text box, enter the number of the beam-column joint whose lateral load moments you want to input. Moments input at the joint are the moments resulting from a lateral load analysis. The analysis could be in either direction, left-to-right or right-to-left.
- Enter the moment to the left, right, top, and bottom of the joint resulting from an analysis under lateral loads from left-to-right, in the corresponding text boxes.

The sum of the left and right moments must be equal to the sum of the top and bottom moments. *Note that the default signs are shown outside the text boxes.*

- Moments of the same magnitude but of opposite direction are automatically assigned to the joint for the right-to-left direction. If different, modify their values.
- Press ENTER or choose the REPLACE button. The list box at the right is updated.
- Repeat the four steps above for each joint. You may use the COPY FROM button as a shortcut (see Working With a Dialog Box).
- Choose the OK button.

Inputting Moment Redistribution Factors

- From the **Input** menu, select **Moment Redistribution**. The dialog box of Figure 4-22 appears. All input fields, except joint numbers, are originally set to zero. *This command is available only if you specified INPUT REDISTRIBUTION in the GENERAL INFORMATION dialog box.*

Joint no.	Percentage Left	Percentage Right
1	0.0	10.0
2	10.0	10.0
3	10.0	10.0
4	10.0	10.0
5	10.0	0.0

Figure 4-22—The Moment Redistribution dialog box

- Input the joint number and the percentages of moment to be redistributed at the left and the right of the joint, in the corresponding text boxes.
- Press ENTER or choose the REPLACE button. The list box at the right is updated.
- Repeat the two steps above for each joint. You can use the COPY FROM button as a shortcut (see *Working With a Dialog Box*).
- Choose the OK button.

Working With the Design Menu Commands

Selecting Output Options

- From the **Design** menu, select **Output Options**. Depending on the run option, one of two dialog boxes in Figure 4-23 appears.
- From the list of check-box options, select the options that you want printed. A selected option has an X in its box.
- Press ENTER or choose the Ok button.

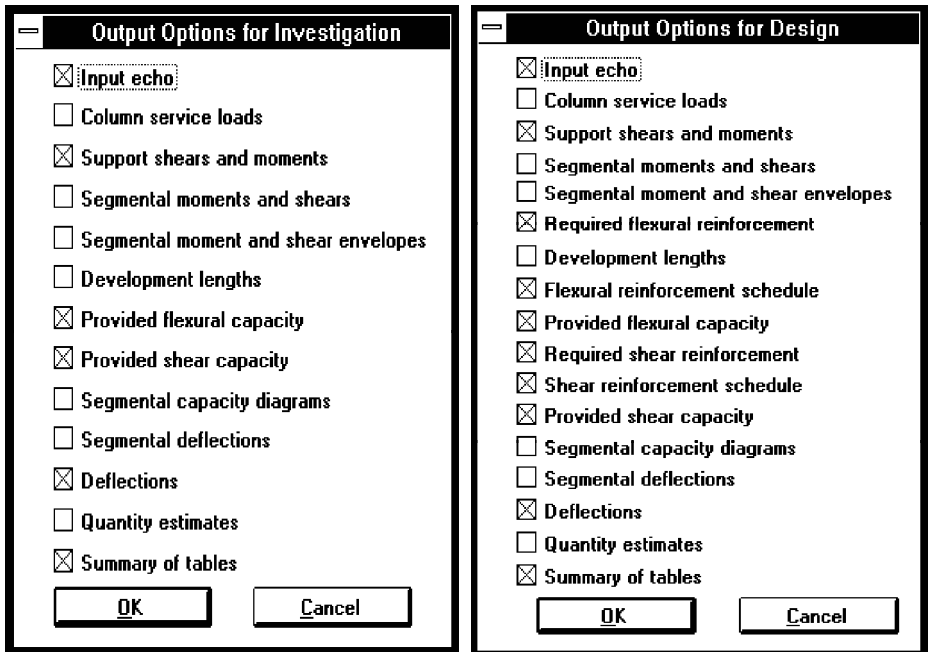


Figure 4-23 —The Output Options dialog boxes

Executing the Solver

- From the **Design** menu, select **Solve**. If any data required to analyze and design or investigate the system are still needed, PCABEAM displays a *Need more data* message and the dialog box of the required data appears.
- The PCABEAM Solver window is brought to the desktop and the solver messages are displayed. When finished, the Solver window is closed and control is returned to PCABEAM.
- Text output is sent to a file (same filename as the input file with an OUT extension). This file may be viewed and/or printed. Data for shear, moment, and deflection diagrams, however, are kept in memory. To save this data, you must use the **File/Save** command.

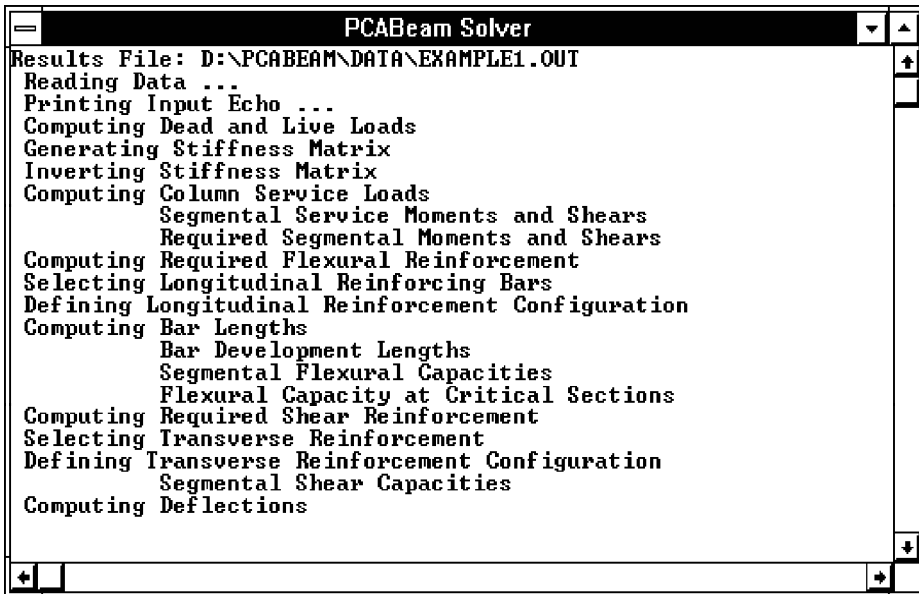


Figure 4-24—The PCABEAM Solver Window

Designing Columns

If the PCACOL program is found on the system, the **Design/PCACOL** command is made available after executing a run. PCACOL is PCA's reinforced concrete column section design and investigation program. If the program is installed and its directory is included in the PATH statement, PCABEAM will find it. Otherwise the **Design/PCACOL** command will be grayed.

- From the **Design** menu, select **PCACOL**. The dialog box of Figure 4-25 appears.

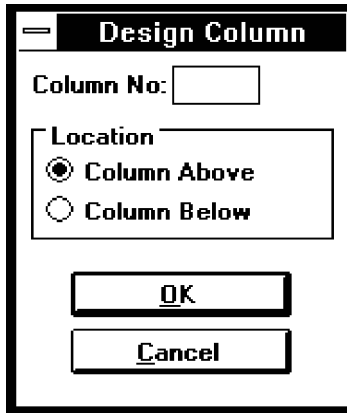


Figure 4-25—The Design Column dialog box

- In the COLUMN NO. text box, enter the number of the column you want to design.
- From the LOCATION options group, select COLUMN ABOVE or COLUMN BELOW.
- Press ENTER or choose the Ok button.
- Alternatively, you may double click the left-mouse button on the desired column from the Elevated View window.
- PCACOL is executed and the needed data are transferred to it. Enter any additional data, then design the column following the procedure described in the PCACOL documentation.
- When you exit PCACOL, you are returned to PCABEAM.

Working With the View Menu Commands

Defining the Zoom Command

The **View/Zoom** command enables you to enlarge any of the Plan, Elevation, or Isometric Views for closer inspection of the model geometry. This command is used for display purposes only and does not affect the model geometry.

- From the **View** menu, select **Zoom**.
- In any of the View windows, using the left-mouse button, marquee select a portion of the graphical image. When you release the mouse button, the selected area occupies the entire View window.
- To reset the zoom action, select **Redraw** from the **View** menu.

Setting the View Preferences

- From the **View** menu, select **Preferences**. The dialog box of Figure 4-26 appears.

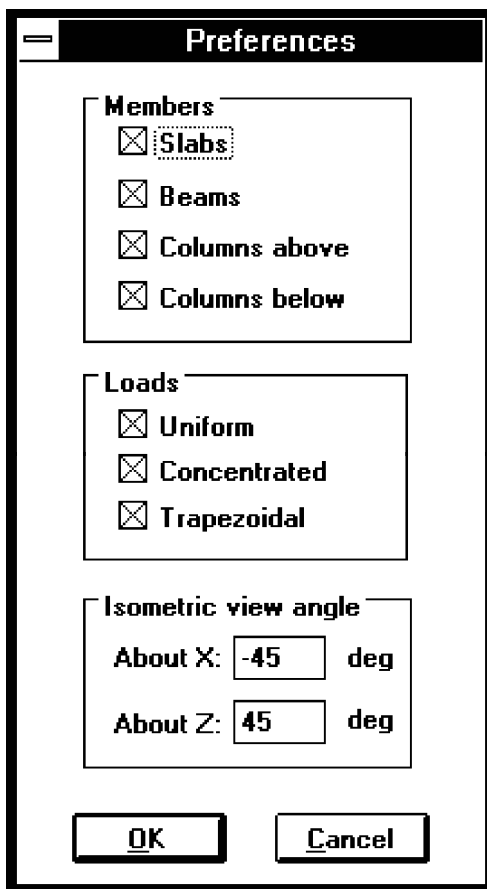


Figure 4-26—The Preferences dialog box

- Select (or clear) the structural members and the loads that you want to show (or hide) in the View windows.

- In the corresponding text boxes, enter the isometric view angle. You may change the angles of rotation about the X and Z axes. Angle of rotation is measured in degrees and is defined using the right-hand rule.
- Press ENTER or choose the OK button.

Viewing the Results

Once a successful run has been performed, you may view the results file.

- From the **View** menu, choose **Results File**. PCABEAM executes the PCAVIEW program, a Windows 3.x utility that enables viewing of text based files. The PCAVIEW window is brought to the top and the results file is in its workplace.
- You can page through the file by clicking the mouse on the vertical scroll bar, or by pressing the PGUP or PGDN keys.
- When done viewing the results, choose **Exit** from the menu line.
- It may be advantageous to maximize the PCAVIEW window so that more data can be viewed.

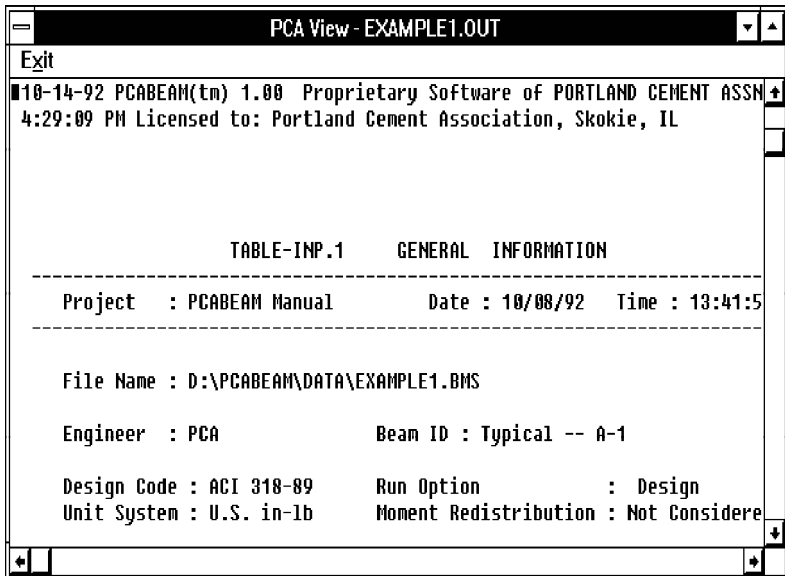


Figure 4-27—The PCAVIEW Window

Viewing Shear & Moment Diagrams

After a successful run, you can view the shear and moment diagrams.

- From the **View** menu, select **Shear & Moment Diagrams**. The dialog box of Figure 4-28 appears.

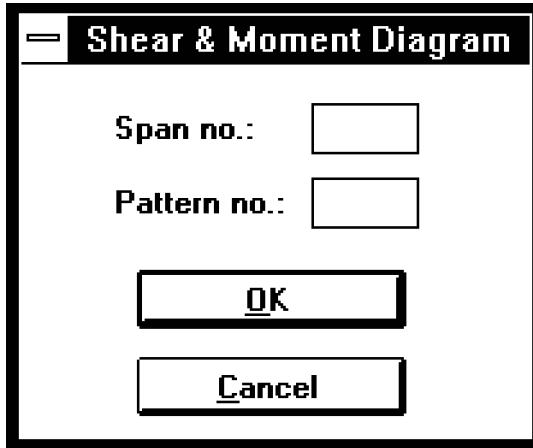


Figure 4-28—The Shear & Moment Diagram dialog box

- Enter the span number and the pattern number in the corresponding text boxes.
- Press ENTER or choose the OK button.

A new window is opened displaying the diagrams for the specified span and load pattern. The window is *movable* and *resizable*. Moving the mouse cursor over the gridded area of the window changes the shape of the cursor to a *cross*. Moving the cross over the graphs enables you to read the location and shear/moment values. These values are displayed at the bottom left-hand corner of the window.

Viewing Shear & Moment Envelope and Capacity Diagrams

After a successful run, you can view the shear and moment envelope and capacity diagrams.

- From the **View** menu, select **Envelope & Capacity Diagrams**. The dialog box of Figure 4-29 appears.

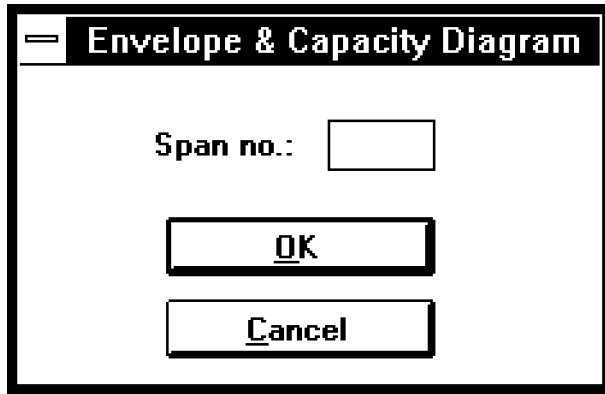


Figure 4-29—The Envelope & Capacity Diagram dialog box

- Enter the span number in the SPAN NO. text box.
- Press ENTER or choose the OK button.

A new window is opened displaying the diagrams for the specified span. The window is *movable* and *resizable*. Moving the mouse cursor over the gridded area of the window changes the shape of the cursor to a *cross*. Moving the cross over the graphs enables you to read the location and shear/moment values. These values are displayed at the bottom left-hand corner of the window.

Viewing Deflection Diagrams

After a successful run, you can view the deflection diagram of any span.

- From the **View** menu, select **Deflection Diagram**. The dialog box of Figure 4-30 appears.

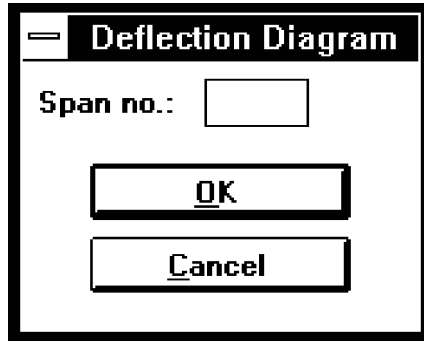


Figure 4-30—The Deflection Diagram dialog box

- Enter the span number in the SPAN NO. text box.
- Press ENTER or choose the OK button.

A new window is opened displaying the deflection diagrams for the specified span due to dead, live, and total loads. The window is *movable* and *resizable*.

Working With the Options Menu Commands

Changing the View Color

- From the **Options** menu, select **Color**. The dialog box of Figure 4-31 appears.

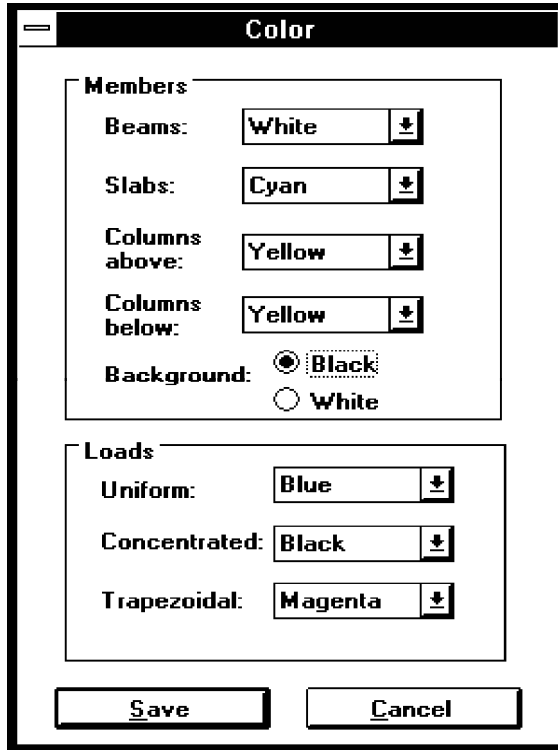


Figure 4-31—The Color dialog box

- Open the drop-down list box associated with the item you want to change the color for, and select a new color from the list. You may change the color of the members displayed in the Plan, Elevation, and Isometric View windows, and the color for the loads displayed in the Partial Loads window.

- Choose a background color, `BLACK` or `WHITE`. The background color affects the Plan, Elevation, and Isometric View windows only.
- Press `ENTER` or choose the `SAVE` button. The color settings are saved in the `PCAPROGS.INI` file.

Changing the Default Units and Code

- From the **Options** menu, select **Default Units & Code**. The dialog box of Figure 4-32 appears.

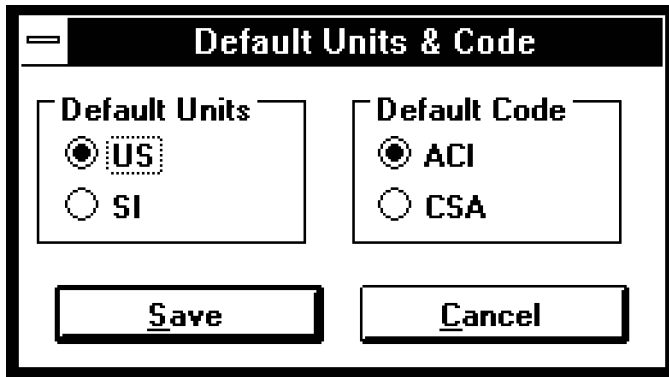


Figure 4-32—The Default Units & Code dialog box

- Select the default units, US or SI, and the default code, ACI or CSA.
- Press ENTER or choose the SAVE button. These settings are saved in the PCAPROGS.INI file and are used by PCABEAM as defaults on startup.

Changing the Default Data Directory

- From the **Options** menu, select **Default Data Directory**. The dialog box of Figure 4-33 appears.

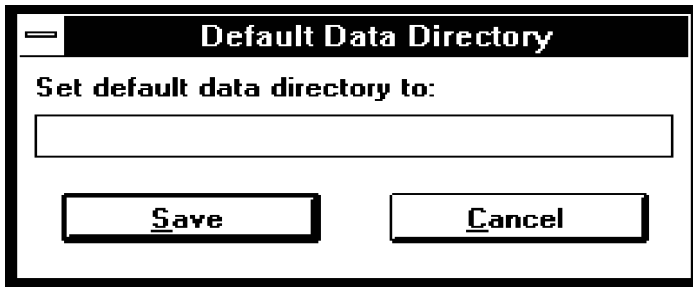


Figure 4-33—The Default Data Directory dialog box

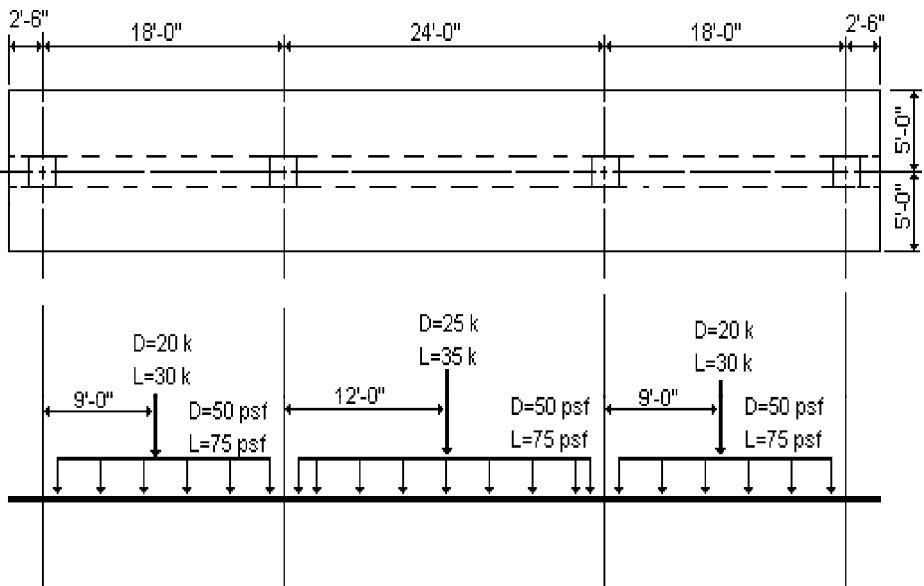
- In the text box, enter the full path of the directory where PCABEAM should store input data files.
- Press ENTER or choose the SAVE button. The default data directory is saved in the PCAPROGS.INI file.

Chapter 5

Examples

Example 1

The system shown in the following figure consists of five spans including a cantilever on each end. All beams are 24 in. \times 20 in. and the slab is 6 in. thick. Span length and widths are shown in the figure. Columns are 24 in. \times 24 in. and the height of the columns below is 10 ft while the columns above are 12 ft in height. The system will be analyzed and designed under the dead and live loads shown below.



1. From the **Input** menu, select **Labels**. In the dialog box, input the project, engineer, and span id labels. Choose Ok.
2. From the **Input** menu, select **General Information**. In the dialog box, input 5 for No. OF SPANS. Leave all other options to their default settings of USIN.-LB units, ACI 318-89 code, DESIGN run option, NO REDISTRIBUTION, INTERIOR frame type, and INTERIOR frame exposure. Choose Ok.

General Information

Number of spans:

Units

U.S. in-lb
 SI metric

Moment redistribution

No redistribution
 Input redistribution

Code

ACI 318-89
 CAN3.A23.3-M84

Frame type

Interior
 Spandrel

Run option

Design
 Investigation

Frame exposure

Interior
 Exterior

3. From the **Input** menu, select **Material Properties**. In the dialog box, input 145 for the beams and columns concrete unit weight. Input 4 for the beams and columns concrete compressive strength. Choose Ok.

Material Properties			
Concrete			
	Beams	Columns	
Unit weight:	<input type="text" value="145"/>	<input type="text" value="145"/>	pcf
Type:	Normal	Normal	
Compressive strength, f'c:	<input type="text" value="4"/>	<input type="text" value="4"/>	ksi
Tensile strength, fct:	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	psi
Modulus of rupture, fr:	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	psi
Reinforcement			
	Bars	Stirrups	
Yield strength, fy:	<input type="text" value="60"/>	<input type="text" value="60"/>	ksi
<input type="checkbox"/>	Epoxy-coated bars		
<input type="button" value="OK"/>		<input type="button" value="Cancel"/>	

4. From the **Input** menu, select **Geometry/Beams & Slabs**. A dialog box appears.
 - For span 1, input 2.5 for length, 24 and 20 for beam width and depth respectively, and 6, 5, and 5 for the slab thickness, width left, and width right, respectively. Choose **REPLACE**. The list box at the right is updated and **SPAN NO.** increments to 2.
 - Input span 2 data (similar to span 1 but use 18 for span length) and choose **REPLACE**. The list box is updated and **SPAN NO.** increments to 3.
 - Input span 3 data (similar to span 2 but use 24 for span length) and choose **REPLACE**.

SPAN NO. increments to 4.

Beams & Slabs Geometry

Span no.:

Length: ft

Beam

Width: in

Depth: in

Slab

Thickness: in

Width left: ft

Width right: ft

Replace

Copy from

Span no.	Span Length	Beam Width	Beam Depth	Slab Thick.	Slab Width Left	Slab Width Right
1	2.5	24.0	20.0	6.0	5.0	5.0
2	18.0	24.0	20.0	6.0	5.0	5.0
3	24.0	24.0	20.0	6.0	5.0	5.0
4	18.0	24.0	20.0	6.0	5.0	5.0
5	2.5	24.0	20.0	6.0	5.0	5.0

OK

Cancel

- Choose the COPY FROM button. In the dialog box input 2 in the COPY FROM SPAN text box and choose COPY. The list box is updated.
 - Input 5 for SPAN NO. and choose the COPY FROM button. In the dialog box input 1 in the COPY FROM SPAN text box and choose COPY. The list box is updated.
 - Choose Ok.
5. From the **Input** menu, select **Geometry/Columns**. A dialog box appears.
- For Column 1, input 24, 24, and 12, for the column above depth, width, and height. Input 24, 24, and 10 for the column below depth, width, and height. Input 100 for STIFFNESS REDUCTION FACTOR. Choose REPLACE. The list box at the right

is updated and COLUMN NO. increments to 2.

☰

Columns Geometry

Column Geometry

Column no.:

Above Below

Depth, c1: in

Width, c2: in

Height: ft

Stiffness Reduction Factor:

%

Column no	loc	c ₁ (in)	c ₂ (in)	Height (ft)	Stiff. Factor
1	Abv	24.0	24.0	12.0	100.0
	Blw	24.0	24.0	10.0	100.0
2	Abv	24.0	24.0	12.0	100.0
	Blw	24.0	24.0	10.0	100.0
3	Abv	24.0	24.0	12.0	100.0
	Blw	24.0	24.0	10.0	100.0
4	Abv	24.0	24.0	12.0	100.0
	Blw	24.0	24.0	10.0	100.0

- From the list box, select columns 2 through 4 and choose COPY FROM. In the dialog box, input 1 in the COPY FROM COLUMN text box and choose COPY. The list box is updated.
- Choose OK.
- 6. From the **Input** menu, select **Reinforcement/Design Parameters**. In the dialog box, input the data shown in the following figure and choose Ok.
- 7. From the **Input** menu, select **Loads/Factors & Patterns**. Leave all parameters to their default values (see following figure) and choose Ok.
- 8. From the **Input** menu, select **Loads/Surface**. A dialog box appears.
 - For span 1, input 50 and 75 for DEAD and LIVE respectively. Choose REPLACE.

The list box at the right is updated and the SPAN NO. increments to 2.

- From the list box, select spans 2 through 5 and choose COPY FROM. In the dialog box, input 1 in the COPY FROM SPAN text box and choose COPY. The list box is updated.
- Choose Ok.

Reinforcement Design Parameters

	Top Bars	Bottom Bars																	
Minimum bar size:	<input type="text" value="5"/>	<input type="text" value="5"/>	<div style="text-align: center;">Stirrups</div> <table style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="width: 50%;"></th> <th style="width: 50%; text-align: center;">Minimum</th> <th style="width: 50%;"></th> <th style="width: 50%; text-align: center;">Maximum</th> </tr> </thead> <tbody> <tr> <td>Size:</td> <td style="text-align: center;"><input type="text" value="3"/></td> <td style="text-align: center;"><input type="text" value="5"/></td> <td style="text-align: center;"><input type="text" value="5"/></td> </tr> <tr> <td>Spacing:</td> <td style="text-align: center;"><input type="text" value="3"/></td> <td style="text-align: center;"><input type="text" value="24"/></td> <td style="text-align: center;">in</td> </tr> <tr> <td colspan="4" style="text-align: center;"> <input type="radio"/> Open <input checked="" type="radio"/> Closed </td> </tr> </tbody> </table>		Minimum		Maximum	Size:	<input type="text" value="3"/>	<input type="text" value="5"/>	<input type="text" value="5"/>	Spacing:	<input type="text" value="3"/>	<input type="text" value="24"/>	in	<input type="radio"/> Open <input checked="" type="radio"/> Closed			
	Minimum			Maximum															
Size:	<input type="text" value="3"/>	<input type="text" value="5"/>		<input type="text" value="5"/>															
Spacing:	<input type="text" value="3"/>	<input type="text" value="24"/>		in															
<input type="radio"/> Open <input checked="" type="radio"/> Closed																			
Maximum bar size:	<input type="text" value="10"/>	<input type="text" value="11"/>																	
Clear vertical cover:	<input type="text" value="2"/>	<input type="text" value="2"/>	in																
Clear side cover:	<input type="text" value="1.5"/>	<input type="text" value="1.5"/>	in																
Min. clear spacing:	<input type="text" value="1"/>	<input type="text" value="1"/>	in																
Max. center-to-center spacing:	<input type="text" value="12"/>	<input type="text" value="12"/>	in																

Load Factors & Patterns

Load Factors

U₁ = Dead + Live

U₂ = (Dead + Live + Lateral)

U₃ = Dead + Lateral

Loading Patterns

<p>Gravity:</p> <p><input checked="" type="radio"/> 1 through 4</p> <p><input type="radio"/> 4 Only</p>	<p>Gravity & Lateral:</p> <p><input type="radio"/> 4 through 8</p> <p><input type="radio"/> 1 through 8</p>
--	--

Pattern live load factor (1 - 3): %

Surface Loads

Span no.:

Dead: psf

Live: psf

Span no.	Loads (psf)	
	Dead	Live
1	50.0	75.0
2	50.0	75.0
3	50.0	75.0
4	50.0	75.0
5	50.0	75.0

9. From the **Input** menu, select **Loads/Partial Dead Loads**. A dialog box appears.
 - Input 2 for SPAN NO., choose CONCENTRATED, and input 20 and 9 for WA and LA respectively. Choose INSERT. The data are added to the list box at the right.
 - Input 3 for SPAN NO., choose CONCENTRATED, and input 25 and 12 for WA and LA respectively. Choose INSERT. The data are added to the list box at the right.
 - Input 4 for SPAN NO., choose CONCENTRATED, and input 20 and 9 for WA and LA respectively. Choose INSERT. The data are added to the list box at the right.
 - Choose OK.
10. From the **Input** menu, select **Loads/Partial Live Loads**. A dialog box appears.
 - Input 2 for SPAN NO., choose CONCENTRATED, and input 30 and 9 for WA and LA respectively. Choose INSERT. The data are added to the list box at the right.
 - Input 3 for SPAN NO., choose CONCENTRATED, and input 35 and 12 for WA and LA respectively. Choose INSERT. The data are added to the list box at the right.

Partial Dead Loads

Span no.:

Uniform
 Concentrated
 Trapezoidal

W_a : kips
 W_b : kips
 L_a : ft
 L_b : ft

Span no.	Load no.	Type	W_a	W_b	L_a	L_b
2	1	CONC	20.0		9.0	
3	1	CONC	25.0		12.0	
4	1	CONC	20.0		9.0	

- Input 4 for SPAN NO., choose CONCENTRATED, and input 30 and 9 for W_a and L_a respectively. Choose INSERT. The data are added to the list box at the right.
 - Choose OK.
11. From the **Design** menu, select **Output Options**. In the dialog box, select the items you want included in the output file and choose OK.
 12. From the **File** menu, select **Save As**. In the dialog box, input a file name (e.g. EX1_D.BMS) in the FILE NAME text box and choose OK.

Partial Live Loads

Span no.	Load no.	Type	W_a	W_b	L_a	L_b
2	1	CONC	30.0		9.0	
3	1	CONC	35.0		12.0	
4	1	CONC	30.0		9.0	

Span no.:

Uniform
 Concentrated
 Trapezoidal

W_a : kips
 W_b : kips
 L_a : ft
 L_b : ft

13. From the **Design** menu, select **Solve**. The PCABEAM Solver window opens and the solver messages are displayed. When the solver is done, the Solver window is closed.
14. From the **View** menu, select **View Results**. The PCAVIEW window opens and occupies the entire desktop. Use the scroll bars or the PGUP, PGDN, or the ARROW keys to scroll through the results file. Select **Exit** from the Menu bar to close PCAVIEW and return to PCABEAM.
15. To view diagrams, select **Shear & Moment Diagrams**, **Envelope & Capacity Diagrams**, or **Deflection Diagrams** from the **View** menu. In the dialog box input a span number and choose Ok. The requested diagram is displayed.

16. You may print the results file by selecting **Print Results** from the **File** menu. To print the Views or any of the diagrams you selected to view, use the **Print Screen** command from the **File** menu.

After viewing the results, you may decide to investigate the input beams under the same loads but with a modified reinforcement configuration.

17. From the **Input** menu, select **General Information**. In the dialog box change the run option to INVESTIGATION. *Do not change any of the other options.* Choose OK. A message box asks if design results should be used as investigation input. Choose YES.
18. From the **Input** menu, select the different commands under **Reinforcement** to modify the reinforcement configuration computed by the program. For more on how to use these commands, see Example 2.
19. Repeat steps 13 and subsequent to perform the investigation and view the results.

Example 2

This example depicts the different cross-sections that you may model using PCABEAM. The system consists of six spans out of which two cantilevers do not exist (leaving four interior spans). Span 2 (the first interior span) is a one-way slab with an 8-in. thick slab. The third span is a 16-ft long T-beam with an 18 in. \times 20 in. beam and 8-in. thick slab. Span four is a 16-ft long L-beam with an 18 in. \times 20 in. beam and 8-in. thick slab. Span five is a 20 ft long rectangular 18 in. \times 20 in. beam. No cantilevers are considered in this example.

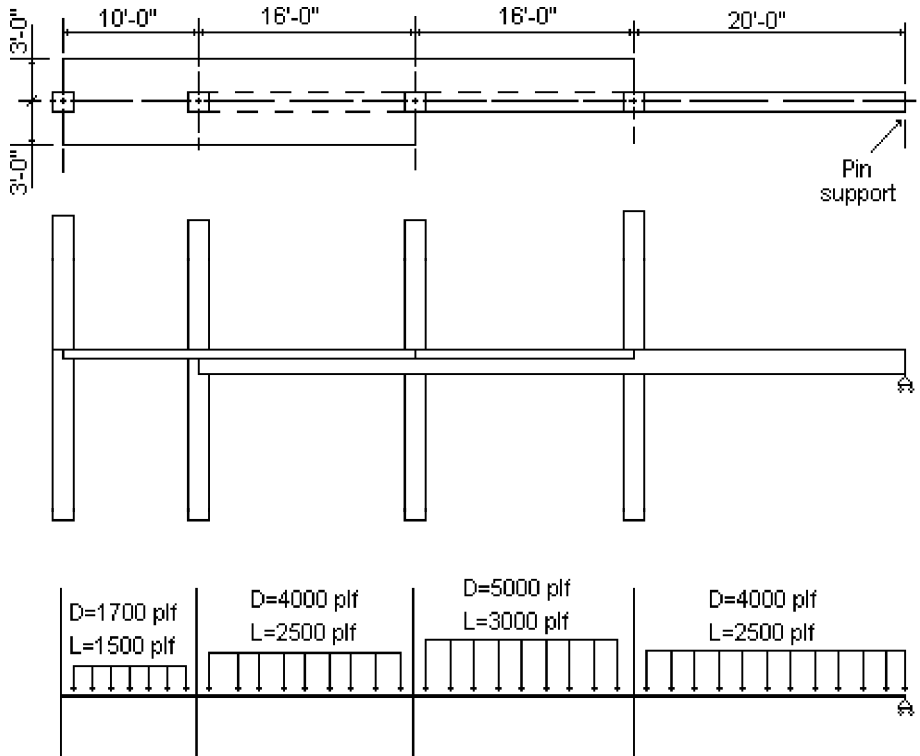
The first four columns are 18 in. \times 18 in., 10-ft high below, and 12-ft high above the design floor. To model the pin support at column 5, a zero column will be input with a 0% stiffness factor. Note that since column 1 is 18 in. \times 18 in., PCABEAM will still force a 0.75-ft long cantilever for span 1. However, this span will not have any sectional properties. The reinforcement of all spans is shown in the following tables. The system will be analyzed under the shown loads and its capacity will be investigated.

Longitudinal Bars

Span	Top Cont.	Top Left	Top Right	Bottom
2	7-#5	—	—	7-#5
3	—	8-#6 \times 4'-0"	8-#6 \times 4'-6"	5-#6
4	—	8-#6 \times 4'-0"	8-#8 \times 6'-0"	5-#6
5	—	8-#9 \times 6'-0"	6-#7 \times 5'-0"	6-#8

Stirrups

Span	# of legs	Size	Number & Spacing
2	—	—	—
3	2	4	21 @ 8"
4	2	4	4 @ 4", 17 @ 8", 4 @ 4in.
5	2	4	12 @ 4", 20 @ 8", 5 @ 4"



1. From the **Input** menu, select **Labels**. In the dialog box, input the project, engineer, and span id labels. Choose Ok.
2. From the **Input** menu, select **General Information**. In the dialog box, input 5 for No. of SPANS. and change the run option to INVESTIGATION. Leave all other options to their default settings of US IN.-LB units, ACI 318-89 code, No REDISTRIBUTION, INTERIOR frame type, and INTERIOR frame exposure. Choose Ok.

General Information

Number of spans:

Units

U.S. in-lb
 SI metric

Moment redistribution

No redistribution
 Input redistribution

Code

ACI 318-89
 CAN3.A23.3-M84

Frame type

Interior
 Spandrel

Run option

Design
 Investigation

Frame exposure

Interior
 Exterior

3. From the **Input** menu, select **Material Properties**. In the dialog box, input 145 for the beams and columns concrete unit weight. Input 4 for the beams and columns concrete compressive strength. Choose Ok.

Material Properties			
Concrete			
	Beams	Columns	
Unit weight:	<input type="text" value="145"/>	<input type="text" value="145"/>	pcf
Type:	Normal	Normal	
Compressive strength, f'c:	<input type="text" value="4"/>	<input type="text" value="4"/>	ksi
Tensile strength, fct:	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	psi
Modulus of rupture, fr:	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	psi
Reinforcement			
	Bars	Stirrups	
Yield strength, fy:	<input type="text" value="60"/>	<input type="text" value="60"/>	ksi
<input type="checkbox"/>	Epoxy-coated bars		
<input type="button" value="OK"/>		<input type="button" value="Cancel"/>	

4. From the **Input** menu, select **Geometry/Beams & Slabs**. A dialog box appears.
 - For span 1, leave all input data to their default 0 values and choose **REPLACE**. The list box at the right is updated and **SPAN NO.** increments to 2.
 - For span 2, input 10, 0, 0, 8, 3, and 3 for span length, beam width, beam depth, slab thickness, width left, and width right, respectively. Choose **REPLACE**. The list box is updated and **SPAN NO.** increments to 3.
 - For span 3, input 16, 18, 20, 8, 3, and 3 for span length, beam width, beam depth, slab thickness, width left, and width right, respectively. Choose **REPLACE**. The list box is updated and **SPAN NO.** increments to 4.

Beams & Slabs Geometry

Span no.:

Length: ft

Beam

Width: in

Depth: in

Slab

Thickness: in

Width left: ft

Width right: ft

Replace

Copy from

Span no.	Span Length	Beam Width	Beam Depth	Slab Thick.	Slab Width	
					Left	Right
1	0.8	0.0	0.0	0.0	0.0	0.0
2	10.0	0.0	0.0	8.0	3.0	3.0
3	16.0	18.0	20.0	8.0	3.0	3.0
4	16.0	18.0	20.0	8.0	3.0	0.8
5	20.0	18.0	20.0	0.0	0.0	0.0
6	0.0	0.0	0.0	0.0	0.0	0.0

OK

Cancel

- For span 4, input 16, 18, 20, 8, 3, and 0.75 for span length, beam width, beam depth, slab thickness, width left, and width right, respectively. Choose REPLACE. The list box is updated and SPAN NO. increments to 5.
 - For span 5, input 20, 18, 20, 0, 0, and 0 for span length, beam width, beam depth, slab thickness, width left, and width right, respectively. Choose REPLACE. The list box is updated and SPAN NO. increments to 6.
 - For span 6, leave all input data to their default 0 values and choose REPLACE. The list box is updated. Choose Ok.
5. From the **Input** menu, select **Geometry/Columns**. A dialog box appears.

Columns Geometry

Column Geometry

Column no.:

Above Below

Depth, c1: in

Width, c2: in

Height: ft

Stiffness Reduction Factor:

%

Column no	loc	c ₁ (in)	c ₂ (in)	Height (ft)	Stiff. Factor
1	Abv	18.0	18.0	10.0	100.0
	Blw	18.0	18.0	12.0	100.0
2	Abv	18.0	18.0	10.0	100.0
	Blw	18.0	18.0	12.0	100.0
3	Abv	18.0	18.0	10.0	100.0
	Blw	18.0	18.0	12.0	100.0
4	Abv	18.0	18.0	10.0	100.0
	Blw	18.0	18.0	12.0	100.0
5	Abv	0.0	0.0	0.0	0.0
	Blw	0.0	0.0	0.0	0.0

- For Column 1, input 18, 18, and 10, for the column above depth, width, and height. Input 18, 18, and 12 for the column below depth, width, and height. Input 100 for STIFFNESS REDUCTION FACTOR. Choose REPLACE. The list box at the right is updated and COLUMN NO. increments to 2.
- From the list box, select columns 2 through 4 and choose COPY FROM. In the dialog box, input 1 in the COPY FROM COLUMN text box and choose COPY. The list box is updated.
- From the list box, select column 5, input 0 in every text box (pin support), and choose REPLACE. The list box at the right is updated.
- Choose OK.

6. From the **Input** menu, select **Reinforcement/Top Continuous**. In the dialog box, input 2 for SPAN NO., 2 for LAYER DISTANCE, and 7 for NO. OF BARS. Choose SINGLE BAR Set type. Input 5 for BAR SIZE and choose INSERT. The data are added to the list box at the bottom. Choose OK.

Top Continuous Bars

Span no.:

Layer distance: in

No. of bars:

Set type

Single bar

Two-lapped bars

Three-lapped bars

Bar size Length (ft)

Insert

Replace

Delete

Span no.	Layer Dist.	No. of bars	----- Left -----	--- Right ---	----- Center -----
			Size Length	Size Length	Size Length Start
2	2.00	7	5 10.00		

OK

Cancel

7. From the **Input** menu, select **Reinforcement/Top Discontinuous**. A dialog box appears.
 - Input 3 for SPAN NO. and 2.5 for LAYER DISTANCE. Input 8, 6, and 4.0 for Left bars number, size, and length. Input 8, 6, and 4.5 for Right bars number, size, and length. Choose INSERT. The data are added to the list box at the bottom.
 - Input 4 for SPAN NO. and 2.5 for LAYER DISTANCE. Input 8, 6, and 4.0 for Left bars number, size, and length. Input 8, 8, and 6.0 for Right bars number, size, and length. Choose INSERT. The data are added to the list box at the bottom.

Top Discontinuous Bars

	No. of bars	Size	Length (ft)	
Span no.: <input style="width: 80px;" type="text" value="5"/>	Left: <input style="width: 40px;" type="text" value="8"/>	<input style="width: 40px;" type="text" value="6"/>	<input style="width: 40px;" type="text" value="4"/>	<input type="button" value="Insert"/>
Layer distance: <input style="width: 80px;" type="text" value="2.5"/> in	Right: <input style="width: 40px;" type="text" value="8"/>	<input style="width: 40px;" type="text" value="6"/>	<input style="width: 40px;" type="text" value="4.5"/>	<input type="button" value="Replace"/>
				<input type="button" value="Delete"/>

Span no.	Layer Distance	----- Left -----			----- Right -----		
		No.	Size	Length	No.	Size	Length
3	2.50	8	6	4.00	8	6	4.50
4	2.50	8	6	4.00	8	8	6.00
5	2.50	8	9	6.00	6	7	5.00

- Input 5 for SPAN NO. and 2.5 for LAYER DISTANCE. Input 8, 9, and 6.0 for Left bars number, size, and length. Input 6, 7, and 5.0 for Right bars number, size, and length. Choose INSERT. The data are added to the list box at the bottom.
 - Choose OK.
8. From the **Input** menu, select **Reinforcement/Bot Continuous**. A dialog box appears.
- Input 2 for SPAN NO., 2 for LAYER DISTANCE, and 7 for No. OF BARS. Choose SINGLE BAR Set type. Input 5 for BAR SIZE and choose INSERT. The data are added to the list box at the bottom.
 - Input 3 for SPAN NO., 2.5 for LAYER DISTANCE, and 5 for No. OF BARS. Choose SINGLE BAR Set type. Input 6 for BAR SIZE and choose INSERT. The data are added to the list box at the bottom.

Bottom Continuous Bars

Span no.:	<input type="text" value="2"/>	Set type <input checked="" type="radio"/> Single bar <input type="radio"/> Two-lapped bars <input type="radio"/> Three-lapped bars	Bar size	<input type="text" value="5"/>	Length (ft)	<input type="text" value="10"/>
Layer distance:	<input type="text" value="2"/> in					
No. of bars:	<input type="text" value="7"/>					

Insert

Replace

Delete

Span no.	Layer Dist.	No. of bars	----- Left ----- Size Length	--- Right --- Size Length	----- Center ----- Size Length Start
2	2.00	7	5 10.00		
3	2.50	5	6 16.00		
4	2.50	5	6 16.00		
5	2.50	6	8 20.00		

OK

Cancel

- Input 4 for SPAN NO., 2.5 for LAYER DISTANCE, and 5 for NO. OF BARS. Choose SINGLE BAR Set type. Input 6 for BAR SIZE and choose INSERT. The data are added to the list box at the bottom.
 - Input 5 for SPAN NO., 2.5 for LAYER DISTANCE, and 6 for NO. OF BARS. Choose SINGLE BAR Set type. Input 8 for BAR SIZE and choose INSERT. The data are added to the list box at the bottom.
 - Choose Ok.
9. From the **Input** menu, select **Reinforcement/Stirrups**. A dialog box appears.
- Input 3 for SPAN NO., 3 for 1ST STIRRUP LOCATION. Input 21, 4, 2, and 8.0 for NO. OF STIRRUPS, SIZE, NO. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.

Stirrups

Span no.:

1st stirrup loc.: in

No. of stirrups:

Size:

No. of legs:

Spacing: in

Insert

Replace

Delete

Span No.	1st stir. loc.	Set No.	No. of stirrups	Size	No. of legs	Spacing
3	3.00	1	21	4	2	8.00
4	3.00	1	4	4	2	4.00
		2	17	4	2	8.00
		3	4	4	2	4.00
5	3.00	1	12	4	2	4.00
		2	20	4	2	8.00
		3	5	4	2	4.00

OK

Cancel

- Input 4 for SPAN NO., 3 for 1ST STIRRUP LOCATION. Input 4, 4, 2, and 4.0 for No. OF STIRRUPS, SIZE, No. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.
- Input 17, 4, 2, and 8.0 for No. OF STIRRUPS, SIZE, No. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.
- Input 4, 4, 2, and 4.0 for No. OF STIRRUPS, SIZE, No. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.
- Input 5 for SPAN NO., 3 for 1ST STIRRUP LOCATION. Input 12, 4, 2, and 4.0 for No. OF STIRRUPS, SIZE, No. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.

- Input 20, 4, 2, and 8.0 for No. OF STIRRUPS, SIZE, No. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.
 - Input 5, 4, 2, and 4.0 for No. OF STIRRUPS, SIZE, No. OF LEGS, and SPACING, respectively. Choose INSERT. The data are added to the list box at the right.
 - Choose Ok.
10. From the **Input** menu, select **Loads/Factors & Patterns**. Leave all parameters to their default values (see following figure) and choose Ok.

Load Factors & Patterns

Load Factors

$U_1 =$ **Dead** + **Live**

$U_2 =$ (**Dead** + **Live** + **Lateral**)

$U_3 =$ **Dead** + **Lateral**

Loading Patterns

<p>Gravity:</p> <p><input checked="" type="radio"/> 1 through 4</p> <p><input type="radio"/> 4 Only</p>	<p>Gravity & Lateral:</p> <p><input type="radio"/> 4 through 8</p> <p><input type="radio"/> 1 through 8</p>
--	--

Pattern live load factor (1 - 3): %

11. From the **Input** menu, select **Loads/Partial Dead Loads**. A dialog box appears.
- Input 2 for SPAN NO., choose UNIFORM, and input 1700, 0.75, and 9.25 for WA, LA, and LB respectively. Choose INSERT. The data are added to the list box at the right.

Partial Dead Loads

Span no.	Load no.	Type	W_a	W_b	L_a	L_b
2	1	UNIF	1700.0	0.8	9.3	
3	1	UNIF	4000.0	0.8	15.3	
4	1	UNIF	5000.0	0.8	15.3	
5	1	UNIF	4000.0	0.8	20.0	

Span no.:

Uniform
 Concentrated
 Trapezoidal

W_a : lb/ft
 W_b : lb/ft
 L_a : ft
 L_b : ft

- Input 3 for SPAN NO., choose UNIFORM, and input 4000, 0.75, and 15.25 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.
 - Input 4 for SPAN NO., choose UNIFORM, and input 5000, 0.75, and 15.25 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.
 - Input 5 for SPAN NO., choose UNIFORM, and input 4000, 0.75, and 20.0 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.
 - Choose OK.
12. From the **Input** menu, select **Loads/Partial Live Loads**. A dialog box appears.
- Input 2 for SPAN NO., choose UNIFORM, and input 1500, 0.75, and 9.25 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.

Partial Live Loads

Span no.:

Uniform
 Concentrated
 Trapezoidal

W_a : lb/ft
 W_b : lb/ft
 L_a : ft
 L_b : ft

Span no.	Load no.	Type	W_a	W_b	L_a	L_b
2	1	UNIF	1500.0		0.8	9.3
3	1	UNIF	2500.0		0.8	15.3
4	1	UNIF	3000.0		0.8	15.3
5	1	UNIF	2500.0		0.8	20.0

- Input 3 for SPAN NO., choose UNIFORM, and input 2500, 0.75, and 15.25 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.
- Input 4 for SPAN NO., choose UNIFORM, and input 3000, 0.75, and 15.25 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.
- Input 5 for SPAN NO., choose UNIFORM, and input 2500, 0.75, and 20.0 for W_a , L_a , and L_b respectively. Choose INSERT. The data are added to the list box at the right.
- Choose Ok.

13. From the **Design** menu, select **Output Options**. In the dialog box, select the items you want included in the output file and choose **Ok**.
14. From the **File** menu, select **Save As**. In the dialog box, input a file name (e.g. EX2_I.BMS) in the **FILE NAME** text box and choose **Ok**.
15. From the **Design** menu, select **Solve**. The PCABEAM Solver window opens and the solver messages are displayed. When the solver is done, the Solver window is closed.
16. From the **View** menu, select **View Results**. The PCAView window opens and occupies the entire desktop. Use the scroll bars or the **PGUP**, **PGDN**, or the **ARROW** keys to scroll through the results file. Select **Exit** from the Menu bar to close PCAView and return to PCABEAM.
17. To view diagrams, select **Shear & Moment Diagrams**, **Envelope & Capacity Diagrams**, or **Deflection Diagrams** from the **View** menu. In the dialog box input a span number and choose **Ok**. The requested diagram is displayed.
18. You may print the results file by selecting **Print Results** from the **File** menu. To print the Views or any of the diagrams you selected to view, use the **Print Screen** command from the **File** menu.

Your comments help us improve our products.

If you have any suggestions as to how we can improve our program, please mail or fax your comments to:

Computer Programs Marketing
Portland Cement Association
5420 Old Orchard Road
Skokie, Illinois 60077-1083
Fax: (708) 966-9781

PCA may use your comments in any appropriate way without incurring obligation. Naturally, you may also use the same information.

Technical Support

To be able to serve you better, we ask that you fax or mail any technical questions you have about any of our programs. Please make copies of the form and use it whenever you need technical support.

