

Acknowledgments

This microcomputer program is an enhanced version of an original Portland Cement Association mainframe program written by K. M. Kripanarayanan. Many of the original and fundamental algorithms are present in much of this microcomputer version.

Additional enhancements, the manual and software design, and microcomputer programming were created and performed by the Computer Programs Marketing Department of the Portland Cement Association.

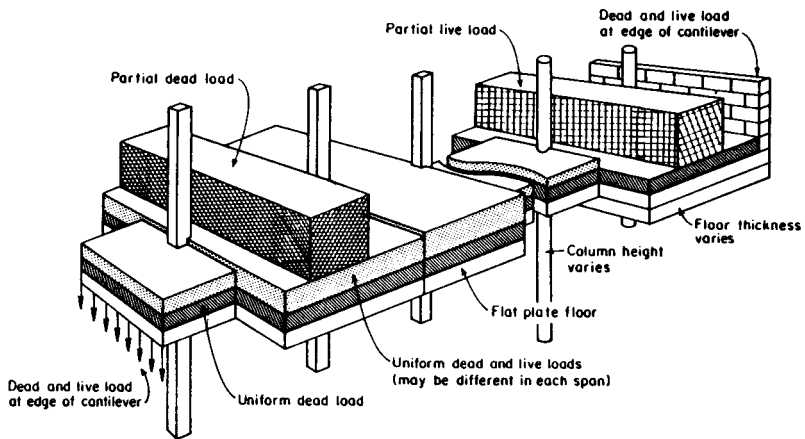
While the Portland Cement Association has taken every precaution to utilize the existing state of the art and to assure the correctness of the analytical solutions and design techniques used in this program, the responsibilities for modeling the structure, inputting data, applying engineering judgement 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.

© 1999 Portland Cement Association. All rights reserved.

MS-DOS is a trademark of Microsoft, Inc.
Microsoft and Windows are registered trademarks of the Microsoft Corporation.

ADOSS Features

ADOSS, the microcomputer version of PCA's mainframe program-Analysis and Design of Slab Systems (ADOSS), analyzes and designs an interior or exterior two-dimensional equivalent frame according to the relevant provisions of the Code. ADOSS provides the following three options: the Strength Design Method, the Limit States Method, and the Alternate Design Method. The program handles both round and rectangular columns, while allowing for user-controlled fixity conditions of all supports. The height of each column may differ, and the slab depth may vary from span to span. Lateral loads may be input into the program either as concentrated story loads, or as joint moments from STMFR II (PCA's frame analysis program) or from any other analysis program. The output includes the analysis of centerline and design moments and shears, punching shear stresses, cracked section deflection, and torsion design of transverse beams. Based on a realistic selection of reinforcement and bar lengths, the program provides a quantity estimate for concrete, reinforcement, and formwork.



Model elements and vertical loads acceptable to ADOSS

1 The Code refers to either the ACI 318-95 Code, or the CSA Standard A23.3-94.

Table of Contents

Installing the Program

Installing ADOSS	1-1
Running ADOSS	1-2
System Requirements	1-2
Backing up the Original ADOSS Set	1-3
Default Options Dialog Box	1-3
Setting the Default Data Directory	1-3
Customer Information	1-5
File Transfer and Change Disk Dialog Boxes	1-5
View README Document	1-5
Running ADOSS	1-6

Method of Solution

Geometric Checks	2-1
Code Checks	2-2
Drop Panel Dimensions	2-2
Column Capital Dimensions	2-4
T-Beam Dimensions	2-4
Waffle Rib Dimensions	2-5
Minimum Slab Thickness of Flat Plate, Flat Slab and Beam-Supported Slab Systems	2-6
Minimum Thickness for Waffle Slab Systems	2-8
Minimum Thickness of Continuous Beams and One-Way Slab Systems	2-8

Special Considerations for Waffle Slabs	2-9
Material Properties	2-11
The Equivalent Frame Method	2-13
Stiffness Characteristics	2-14
Slab Beams	2-14
Columns	2-17
Loading	2-24
Self-Weight	2-25
Superimposed Loading	2-25
Lateral Loading	2-26
Loading Patterns	2-29
Column and Middle Strip Widths	2-33
Design Moments	2-35
Shear Analysis of Slabs	2-42
Critical Section for Interior Supports of Interior Frames	2-43
Critical Section for Exterior Supports of Interior Frames	2-44
Critical Section for Interior Supports of Exterior Frames	2-44
Critical Section for Exterior Supports of Exterior Frames	2-44
Computation of Allowable Shear Stress at Critical Section	2-46
Computation of Factored Shear Force at Critical Section	2-47
Computation of Unbalanced Moment at Critical Section	2-49
Computation of Shear Stresses at Critical Section	2-51
Shear Analysis of Longitudinal Beams	2-52
Shear and Torsion Analysis of Transverse Beams	2-55
Area of Reinforcement	2-59

ADOSS Graphics

Reinforcement Selection	2-61
Additional Reinforcement at Support	2-64
Additional Information in Span	2-66
Deflection Calculation	2-66
Deflections of Continuous Beams and One Way Slabs	2-70
Cracking	2-70
Deflection Computations	2-73
Material Quantities	2-74

Menu Descriptions

ADOSS Graphics Menu Descriptions	3-1
The File Menu	3-1
The Input Menu	3-4
The View Menu	3-6
The Design Menu	3-8
The Options Menu	3-9
The Window Menu	3-10
The Help Menu	3-11
The Control Menu	3-12

Operating the Program

Main Screen Components	4-1
Title Bar	4-1
Control Menu	4-1
Menu Line	4-2

Dialog Boxes	4-3
Edit Boxes	4-3
Menu and Dialog Controls	4-4
Enabled/Disabled Options	4-4
Radio Buttons	4-4
Check Boxes	4-4
Push Buttons	4-5
Working With Data Files	4-5
Creating a New Data File	4-5
Saving Your Data	4-6
Opening an Existing Data File	4-7
Modeling a Building Floor System	4-8
Including Labels with the Output	4-8
Creating the Floor System	4-9
Using the General Information Command	4-10
Defining the Material Properties	4-12
Defining Reinforcement Parameters	4-14
Defining the Structure Geometry	4-16
Defining the Slab Geometry	4-18
Defining the Rib Geometry	4-21
Defining Longitudinal Beam Geometry	4-22
Defining Column and Capital Geometry	4-26
Defining Drop Geometry	4-29
Defining Transverse Beam Geometry	4-31
Changing the Column Fixity	4-33

ADOSS Graphics

Specifying Loads on the Floor System	4-34
Specifying Uniform Surface Loads	4-34
Specifying Partial Surface, Concentrated, and Moment Loads	4-35
Specifying Lateral Loads	4-38
Changing the Live Load Reduction Factor	4-40
Changing the Load Factors	4-41
Applying More Lateral Load to the Column Strip	4-42
Viewing the Input Data	4-43
Viewing the Floor System with the Scale	4-43
Zooming in on the System	4-43
Changing the Isometric View Angle	4-44
Viewing Specific Member Types	4-45
Viewing Specific Load Types	4-46
Designing	4-47
Designing the Floor System	4-50
Designing the Column	4-50
Viewing and Printing Design Results	4-52
Viewing and Printing the Results	4-52
Viewing and Printing Shear and Moment Diagrams	4-53
Viewing and Printing the Deflected Shapes	4-55
Customizing ADOSS	4-56
Changing the Background Color	4-56
Changing the Member Colors	4-57
Changing the Load Colors	4-58
Changing the Default Units and Code	4-59

Changing the Default Data Directory4-60

Changing the Margins of a Screen Print4-60

Manipulating the Windows4-61

 Working with Windows4-62

Getting On-Line Help4-63

Obtaining the ADOSS Version Number4-65

Output Description

Output Elements5-1

Program Version5-2

Description of Input Echo5-2

CODE Checks5-3

Analysis Elements5-3

Design Elements5-7

ADOSS Graphics

Conversion Factors

Conversion Factors – U.S. to SIA-1
U.S. Bar SizesA-2
SI Bar SizesA-3

References

ReferencesB-1

Examples

Example 1, Flat SlabC-2
Example 2, Beam-Supported SlabC-19
Example 3, Continuous BeamC-40

Chapter 1

Installing the Program

Installing ADOSS

1. Execute Windows and insert the ADOSS diskette into the proper floppy disk drive.
2. From the Program Manager File menus, 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 ADOSS is to be installed. The default directory is `C:\ADOSS`. 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.
5. The DEFAULT OPTIONS dialog box appears next. This allows you to customize ADOSS by setting the units of measure (US or Metric) and the Code (ACI or CSA). These defaults are saved in the PCAPROGS.INI file, located in the WINDOWS directory.
6. 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.
7. The setup program begins to copy files from the installation drive to the destination drive. Once the file transfer is done, setup adds the "ADOSS" icon to the Program Manager. A "PCA Programs" group box is created (if one does not exist) and the "ADOSS" icon is added.
8. After the setup program displays a successful installation message, the last dialog box displayed asks whether you would like to view the README.TXT file. This file contains information supplemental to the manual.

Running ADOSS

To execute ADOSS from within Windows:

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

To execute ADOSS from the DOS prompt:

- Change the directory to where the program is installed, type in ADOSS and press ENTER. This will execute Windows and load the ADOSS program as the application program.

System Requirements

To use ADOSS, a personal computer with the following hardware and software configuration is needed.

- MS-DOS operating system version 5.0 or higher.
- Microsoft Windows version 3.1 (running in Enhanced mode).

Minimum requirements:

- 4.0 MB of RAM memory with 1 M of free conventional memory.
- One floppy disk drive.
- A hard disk with at least 2.0 MB of free space.
- A monitor supported by Windows V3.1.
- A mouse or other pointing device supported by Windows V3.1.

Recommended Options:

- A printer supported by Windows V3.1
- A math co-processor.

Backing up the Original ADOSS Set

Before you install ADOSS, take time out to make a backup copy of the ADOSS disk. The license agreement states that you are allowed to make one backup copy of the disk. This backup set should be safely stored away and used only if the original disk is damaged in any way.

IMPORTANT: The license agreement states that only one backup could be made. This set is strictly a backup copy and it is not intended to be used on any other machine.

Default Options Dialog Box

This dialog box allows you to customize ADOSS to your needs by setting two default options; units of measure (US or SI) and code (ACI or CSA). This will eliminate the task of specifying these options for each ADOSS session.

These defaults are saved in the file PCAPROGS.INI located in the Windows directory. Each time ADOSS is executed, PCAPROGS.INI is read by ADOSS and the information is incorporated in the session. You can also change these settings in ADOSS by using the Default Unit/Codes command of the Options menu.

*Note: Selecting **Cancel** will set the units and code options to the default values. The default values are initially set to US units and ACI code.*

Setting the Default Data Directory

PCA Setup will prompt you for the default data directory. The default data directory is the directory where ADOSS will look for data when you elect to open a data file. The ADOSS default data directory is <ADOSS Directory>\DATA. Figure 1-1 displays the default data directory dialog box.

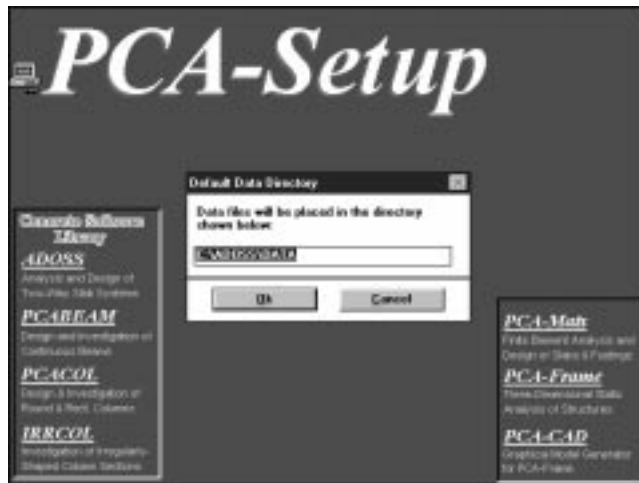


Figure 1-1 Default Data Directory dialog box

To use the default directory, press **Enter** or select **OK** . If the data directory does not exist, it will be created. The default data directory will be written to the PCAPROGS.INI file. You can change the default data directory in ADOSS by using the Default Data Directory command from the Options menu.

A different directory can be used for the data other than that given. Use the backspace key and enter the new directory where you want ADOSS to look for data. If this directory does not exist, it will be created.

*Note: Selecting **Cancel** will leave the default data directory unspecified. This approach is practical to the user who has multiple data directories and who does not want to specify a particular data directory. If the default data directory is left unspecified, ADOSS assumes the current directory as the default data directory.*

Customer Information

Record your firm name, city, and state. Be sure to tab to the city and state edit boxes instead of pressing enter. By selecting **Ok** or pressing **Enter**, you are confirming your choices. By selecting **Cancel**, changes made to this dialog box will not be saved.

If you select **Enter** a customer information verification box will be displayed prompting you to confirm your choices. Select **Enter** to save the information or select **Change** if you want to edit the information displayed.

*Note: Press **Enter** only after all the edit boxes are filled in.*

File Transfer and Change Disk Dialog Boxes

PCA Setup will begin to copy files from the drive the ADOSS Disk was placed in to the ADOSS Directory. The source path and destination path are displayed in this dialog box along with a file transfer progress scale.

View README Document

After PCA Setup displays a successful installation message, the last dialog box displayed will ask whether you would like to view the ADOSS README file.

This is a good opportunity to read this file which contains information supplemental to the manual. Selecting **Yes**, will cause PCA Setup to execute the viewing utility, PCA View, and load the README file. Selecting **No** will simply terminate the PCA Setup program.

IMPORTANT: A message box will appear acknowledging a successful installation of ADOSS V7.00. If PCA Setup modified the AUTOEXEC.BAT file, reboot your system in order for the changes made to the AUTOEXEC.BAT file to take effect. If PCA Setup did not modify the AUTOEXEC.BAT file, then you are ready to run ADOSS V7.00.

Running ADOSS

There are a number of ways to execute ADOSS from within Windows. One way is to select the ADOSS Icon in the “PCA Programs” group window from within the Program Manager and press **Enter** . Another way is to select the Run command under the File menu in the Program Manager. A dialog box will be displayed prompting for a command line. Type “ADOSS” in the edit box and press **Enter** . Lastly, you can execute ADOSS from the File Manager by selecting an ADOSS data file with an .ADS extension and pressing **Enter** . PCA Setup creates this application association between the application program, ADOSS, and the data files which have the .ADS extension.

Chapter 2

Method of Solution

The user should be aware of the assumptions made by the program during the design stage. These include details regarding loading, strip widths, reinforcement selection, deflection computations, material quantities, etc.

Geometric Checks

The ADOSS program provides geometric to avoid an analysis with an inconsistent system. The dimensions of the slabs, drops, and column capitals are checked and modified to produce a legitimate system.

If the slab cantilever length is less than one-half the column dimension in the direction of analysis c_1 , or less than the extension of the transverse beam into the cantilever, the cantilever length will be increased to the larger of these two lengths. If the slab width is less than one-half the column dimension transverse to the direction of analysis, c_2 , or less than one-half the longitudinal beam width, the slab width will be increased to the larger of these two widths.

If the drop panel lengths extend beyond the end of the slab cantilevers, the drop panel lengths will be reduced so that they extend only to the cantilever tip. The drop panel will be shifted forward or backward in the transverse direction when the slab strip width on either side of the column is less than one-half the drop panel width.

If a column capital contains an extension/depth ratio greater than 1, the extension will be set equal to the capital depth. If, by drawing lines extending the column capital through the drop or beam to the underside of the slab, the column capital extension falls outside the edge of the drop or beam at the slab soffit, the extension will be modified such that these lines extend only up to the edge of the drop or beam at the slab soffit (Figure 2-1).

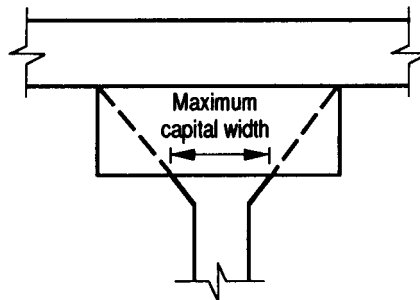


Figure 2-1 Maximum capital width

Code Checks

Prior to an analysis, ADOSS checks that the input values for the drop panel dimensions, column capital dimensions, T-beam flange widths, waffle slab rib dimensions, and slab and beam depths meet the applicable code values.

Drop Panel Dimensions

A valid ACI 318 drop must extend in each direction at least one-sixth the center-to-center span length in that direction and have a drop depth below the slab of at least one-quarter the input slab depth (Figure 2-2). ACI slabs that contain valid drops are allowed a 10% decrease in minimum slab depth.¹ The drop depth of an invalid drop will not be used in the calculation of the depth used to reduce the amount of reinforcement required at a column,² and will not be used in the calculation of moment of inertia for deflection computations.

For CSA A23.2 drop panels, the extension of the drop panel beyond the edge of the column or column capital is limited to one quarter of the clear span. No minimum is specified for this dimension. The maximum depth of the drop panel below the slab is limited to the input slab depth.

1 ACI 318-95, 9.5.3.2

2 ACI 318-95, 13.3.7

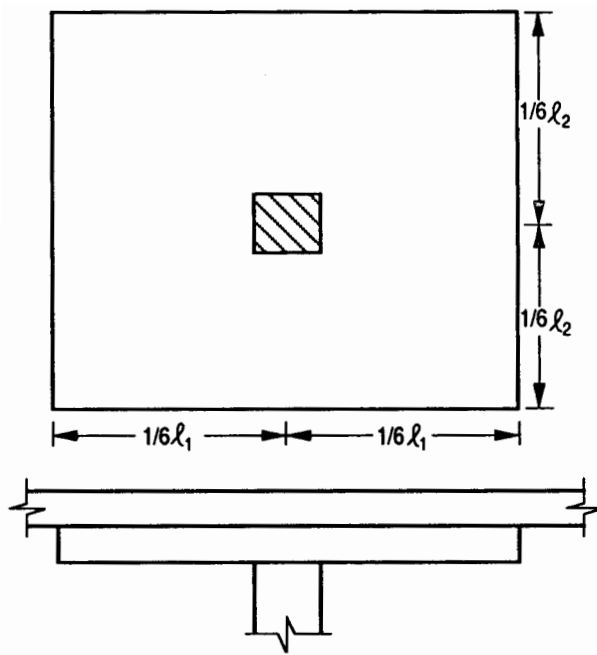


Figure 2-2 Valid drop dimensions

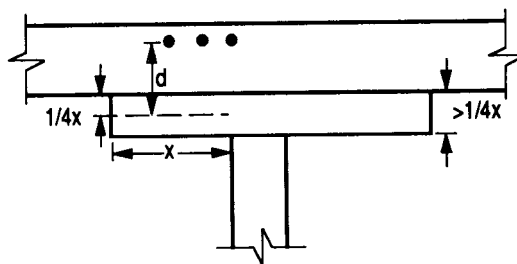


Figure 2-3 Excess drop depth

If the valid drop depth is greater than one-quarter the distance from the edge of the drop panel to the face of the column or column the excess depth will not be considered in the calculation of the effective depth used to reduce the amount of reinforcement required at a column (Figure 2-3).³

The input drop dimensions will be used for self-weight computations, when computing slab stiffness and to determine moments and shears when computing punching shear around a column.

Column Capital Dimensions

If a column capital contains an extension/depth ratio greater than 1, the extension will be set equal to the capital depth. If, by drawing lines extending the column capital through the drop or beam to the underside of the slab, the column capital extension falls outside the edge of the drop or beam at the slab soffit, the extension will be modified such that these lines extend only up to the edge of the drop or beam at the slab soffit (Figure 2-1).

The modified column capital dimensions will be used when computing column stiffness.

T-Beam Dimensions

For the ACI Code the effective T-beam flange width will not exceed one-quarter the beam span length for the ACI code. For beams that have slabs on both sides, the maximum flange overhang will not exceed eight times the slab thickness or the input slab width on that side. For beams that have slab on one side only, the maximum flange overhang will not exceed one-twelfth the beam span length, six times the slab thickness, or the input slab width on the overhang side (Figure 2-4)⁴

For the CSA Standard the effective flange width of T-beams is based on the overhanging flange widths on each side of the web which are not greater than 1/10 of the span length, 12 times the slab thickness or 1/2 the clear distance to the next web. Where a slab exists on only one side of the beam, the overhanging flange width is limited to 1/12 of the span of the beam, 6 times the slab thickness, and 1/2 the clear distance to the next web.

3 ACI 318-95, 13.3.7.3; CSA A23.3-94, 13.11.6

4 ACI 381-95, 8.10.2-8.10.3; CSA-A23.3-94, 10.3.3-10.3.4

ADOSS Graphics

These T-beam dimensions will be used for computing the stiffness. The entire slab width, however, will be used when computing self-weight and superimposed surface dead and live loads.

Waffle Rib Dimensions

Waffle slab rib dimensions will be considered valid if the rib width is at least 4 in., the depth is no more than 3-1/2 times the rib width, and the clear spacing between ribs does not exceed 30 in. (Figure 2-5). When valid ribs exist, the ACI code permits the nominal concrete shear strength, V_c , to be increased by 10%.⁵

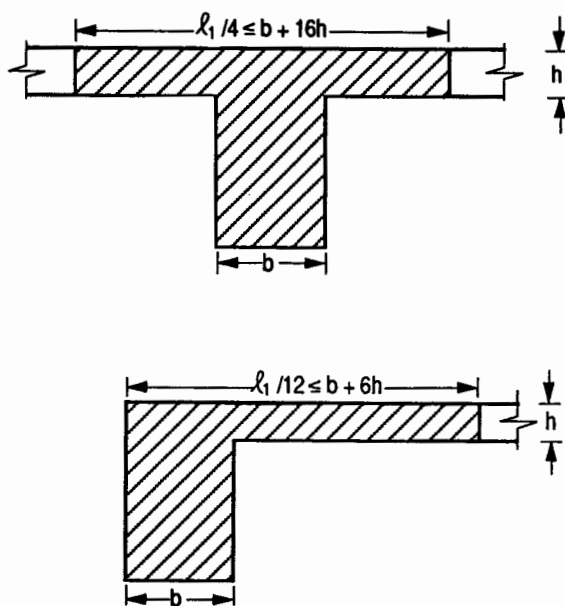


Figure 2-4 Allowable T-beam flange widths

5 ACI 318-95, 8.11

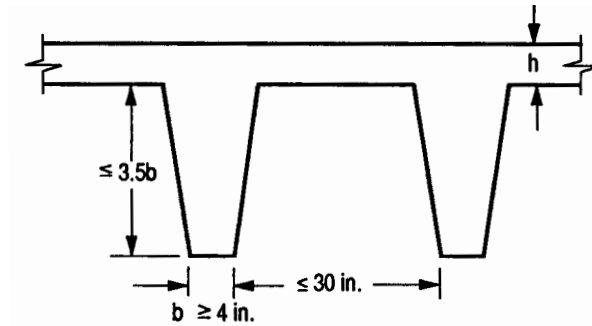


Figure 2-5 Valid rib dimensions

Minimum Slab Thickness of Flat Plate, Flat Slab and Beam-Supported Slab Systems

For ACI 318-95 Code, the minimum thickness of slabs with or without interior beams spanning between supports and having a ratio of long to short span not exceeding 2 is:⁶

$$h = \frac{\ell_n \left(0.8 + \frac{f_y}{200,000} \right)}{36 + 5\beta (\alpha_m - 0.2)} \quad 0.2 < \alpha_m \leq 2.0 \quad \text{Eq. 2-1}$$

but not less than

$$h = \frac{\ell_n \left(0.8 + \frac{f_y}{200,000} \right)}{36 + 9\beta} \quad \alpha_m > 2.0 \quad \text{Eq. 2-2}$$

For CSA A23.3-94

$$h_s \geq \frac{\ell_n (0.6 + f_y/1000)}{30 + 4\beta\alpha_m} \quad \alpha_m \text{ taken } \leq 2.0$$

6 318-95, 9.5.3.3; CSA-A23.3-94, 13.3.5

where

- l_n = clear span in the direction of analysis.
- β = ratio of the clear spans in long to short direction.
- α_m = average value of α , the ratio of flexural stiffness of a beam section to the flexural stiffness of a width of slab bounded laterally by centerlines of adjacent panels on either side of the beam, for all beams supporting the edges of a slab panel.

For the design of ACI slabs without beams spanning between supports the minimum thickness shall conform to ACI 318 Table 9.5(c) For flat slabs that contain valid drops, Table 9.5 (c) reduces the minimum thickness by approximately 10%.⁷ (See Figure 2-2).

The minimum depth in a span that contains a discontinuous edge will be increased by 10% if the edge beam provided has a stiffness ratio, α , of less than 0.80.⁸ The first and last spans are considered to contain a discontinuous edge as well as a span that contains an exterior edge.

For the CSA Standard the minimum thickness is:

- a) for flat plates and slabs with column capitals

$$h_s > l_n (0.6 + f_y / 1000) / 30$$

- b) for slabs with drop panels

$$h_s \geq \frac{l_n (0.6 + f_y / 1000)}{30 \left[1 + \left(\frac{2X_d}{l_n} \right) \left(\frac{h_d - h_s}{h_s} \right) \right]}$$

Eq. 2-3

where $X_d / (l_n/2)$ is the smaller of the values determined in the two directions, X_d shall not be taken greater than $l_n/4$ and $h_d - h_s$ shall not be greater than h_s .

7 ACI 318-95, 9.5.3.2

8 ACI 318-95, 9.5.3.3 (d)

For flat plate systems, the minimum allowable thickness can in no case be less than 5.0 in. For two-way flat slab systems with drops, described above, the minimum allowable thickness can in no case be less than 4.0 in., otherwise 5.0 in.⁹ For supported slab systems supported by beams with an a_m greater than or equal to 2.0, the minimum allowable thickness can in no case be less than 3.5 in., otherwise 5.0 in.¹⁰ is taken on the minimum allowable thickness.

A message will be printed with the output showing the required slab depth for any span where the depth does not meet or exceed code requirements.

Minimum Thickness for Waffle Slab Systems

The minimum slab thickness allowed for waffle slabs is one-twelfth the clear rib spacing, or 2 in.¹¹

A message will be printed with the output showing the required slab depth for the system if the depth does not meet or exceed code requirements.

Minimum Thickness of Continuous Beams and One-Way Slab Systems

The minimum depth of continuous beam systems and one-way slabs, made of normal weight concrete and using Grade 60 reinforcing bars below which deflections must be computed is shown in Table 2.1.¹² Note: l in Table 2.1 is center-to-center span lengths for ACI 318 and clear span lengths for CSA-A23.3.

For concrete density in the range of 90–120 pcf, the above minimum thicknesses are increase 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 thicknesses are increased by $(0.4 + f_y/100)$.¹³

A message will be printed with the output showing the required beam or slab depth for any span where the depth does not meet or exceed code requirements.

9 ACI 318-95, 9.5.3.2

10 ACI 318-95, 9.5.3.3

11 ACI 318-95, 8.11.6.1

12 ACI 318-95, 9.5.2.1; CSA-A23.3 94, 9.8.5.2, Table 9-1

13 ACI 318-95, Table 9.5(a)

Table 2.1 – Minimum Thickness of Continuous Beam and One-Way Slab Systems

Minimum Thickness, h				
Floor System	Interior Span of a Three Span System	First and Last Interior Span	Other Interior Spans	Cantilever
Continuous Beams	ℓ16	ℓ18.5	ℓ21	ℓ8
One-Way Slabs	ℓ20	ℓ24	ℓ28	ℓ16

Special Considerations for Waffle Slabs

For the purposes of analysis and design, ADOSS replaces the waffle with solid slabs of equivalent moment of inertia, weight, and punching shear resistance.

The equivalent thickness based on system weight is used to compute the system self-weight. This thickness, h_w , is given by:

$$h_v = \frac{V_{mod}}{A_{mod}} \tag{Eq. 2-4}$$

where

V_{mod} = the volume of one waffle module.

A_{mod} = the plan area of one waffle module.

The equivalent thickness based on moment of inertia is used to compute slab stiffness. The ribs spanning in the transverse direction are not considered in the stiffness computations. This thickness, h_{MI} , is given by:

$$h_{MI} = \left(\frac{12MI}{B_{rib}} \right)^{1/3} \tag{Eq. 2-5}$$

where

MI = moment of inertia of one waffle section between centerlines of ribs.

b_{rib} = the center-to-center distance of two ribs (clear rib spacing plus rib width).

The drop depth for waffle slab systems is set equal to the rib depth; therefore, an input drop depth of 0 is sufficient. The equivalent drop depth based on moment of inertia, d_{MV} , is given by:

$$d_{drop} = h_{slab} + h_{rib} - h_{MI} \quad \text{Eq. 2-6}$$

where

h_{slab} = overall thickness of slab.

h_{rib} = rib depth below slab.

h_{MI} = equivalent slab thickness based on moment of inertia.

A drop depth entered for a waffle slab system other than 0 will be added to d_{MV} , thus extending below the ribs.

The equivalent thickness based on shear area is used to compute the area of concrete section resisting shear transfer, A_c around the drop. The equivalent slab thickness, h_v , used to compute A_c , is given by:

$$h_v = \frac{A_{rib}}{b_{rib}} + d_{reinf} \quad \text{Eq. 2-7}$$

where:

A_{rib} = the entire rib area below the slab plus the slab thickness minus the distance to the reinforcement centroid, d_{reinf} , within the rib width, (i.e., the slab depth between the ribs is not considered as contributing to shear resistance.)

b_{rib} = the center-to-center distance of two ribs (clear rib spacing plus rib width).

d_{reinf} = the distance to reinforcement centroid from the slab top at the support.

Material Properties

By entering the concrete density and compressive strength of the members, default values for the other concrete properties are determined. The slab, column, and beam members may have different concrete properties.

The density of concrete is used to determine the type of concrete, modulus of elasticity, and self-weight.

For the ACI Code, the concrete type is used to determine the default value of f_{ct} , the average split tensile strength of concrete. The concrete type is determined in accordance with Table 2.2.

Note: The CSA Standard does not use f_{ct}

Table 2.2 –Default Concrete Types

Density (w)		Type
pcf	kg/m3	
$w \geq 130$	$w \geq 2000$	Normal
$105 < w < 130$	$1700 < w < 2000$	Sand-Lightweight
$w \leq 105$	$w \leq 1700$	All-Lightweight

Once f'_c the compressive strength of concrete, is input, various parameters are set to their default values.

The modulus of elasticity is computed as:¹⁴

$$E_c = 33w^{1.5}\sqrt{f'_c}$$

Eq. 2-8

14 ACI 318-95, 8.5.1; CSA-A23.3-p4, 8.6.2.2

where

w = the density of the concrete.

For CSA A23.3 $E_c = (3300\sqrt{f'_c} + 6900)\left(\frac{\gamma}{2300}\right)^{1.5}$

The square root of f'_c is limited to 100 psi for the computation of shear strength provided by concrete, V_c , and development lengths.¹⁵

$$f_{ct} = 6.7\sqrt{f'_c} \tag{Eq. 2-9}$$

Equations 2-8 and 2-9 are used internally and cannot be modified.

The average split tensile strength is used to compute the modulus of rupture and reinforcement development lengths. For normal weight concrete, the default value of f_{ct} used by ADOSS is set equal to:¹⁶

$$f_{ct} = 6.7\sqrt{f'_c} \tag{Eq. 2-10}$$

In no case will $f_{ct}/6.7$ exceed $\sqrt{f'_c}$. f_{ct} will be modified according to the concrete type. Table 2.3 shows the default values for f_{ct} for different concrete types. No interpolation is performed for partial sand replacement.

Table 2.3 – Default Average Splitting Tensile Strength

Type	fct
Normal	6.7
Sand-Lightweight	<i>(0.85)</i> 6.7 $\sqrt{f'_c}$
All-Lightweight	<i>(0.75)</i> 6.7 $\sqrt{f'_c}$

¹⁵ ACI 318-95, 11.1.2

¹⁶ ACI 318-95, 9.5.2.3

ADOSS Graphics

The modulus of rupture is used to determine the cracking moment when computing the effective moment of inertia for deflection computations. The default value of f_r , modulus of rupture, is set equal to:

For ACI 318

$$f_r = 1.12 f_{ct} \quad \text{Eq. 2-11}$$

If f_{ct} is that given in Eq. 2-10, Eq. 2-11 is equal to:¹⁷

$$f_r = 7.5 \sqrt{f'_c} \quad \text{Eq. 2-12}$$

For CSA A23.3 $f_r = 0.6 \sqrt{f'_c}$ for beams and columns and $f_r = \frac{0.6 \sqrt{f'_c}}{2}$ for slabs.

There is no limit imposed on f_r . Entering a large value of f_r will produce deflections based on gross properties, (i.e., uncracked sections).

The default values for the longitudinal reinforcement yield strength, f_y , and shear reinforcement yield strength, f_{yv} , if applicable, are set equal to 60 ksi (400 MPa).

The Equivalent Frame Method

The equivalent frame method, as described in the Code,¹⁸ is used by ADOSS for both analysis and design.

The Code specifies procedures for the analysis and design of slab systems reinforced for flexure in more than one direction, with or without beams between the supports. A two-way slab¹⁹ system, including the slab and its supporting beams, columns, and walls may be designed by either of the following procedures:

- The Direct Design Method
- The Equivalent Frame Method

¹⁷ ACI 318-95, Eq.9-9; CSA A23.3-94, 8.6.4 & 13.3.6

¹⁸ ACI 318-95, Chapter 13; CSA A23.3-94, Clause 13

¹⁹ Implies a slab supported by isolated supports which permits the slab to deflect in two orthogonal directions.

ADOSS uses the Equivalent Frame Method of analysis. It should be noted that this method is based on extensive analytical and experimental studies conducted at the University of Illinois. Note also that there are no restrictions on the number of slab spans or on dead-to-live load ratios in this method of analysis.

The first step in the frame analysis is to divide the three-dimensional building into a series of two-dimensional frames extending to the full height of the building. Horizontal members for each frame are formed by slab strips as shown in Figure 2-6. For vertical loads, each story (floor and/or roof) may be analyzed separately with the supporting columns being considered fixed at their remote ends. (Figure 2-7).

Stiffness Characteristics

The stiffnesses and carryover factors are determined for the horizontal members (the slab beams) and the vertical members (the equivalent columns) using column analogy, after dividing each member into 220 equal segments.

Slab Beams

The moment of inertia of the slab beam elements *between* the faces of the columns (or column capitals) is based on the uncracked section of the concrete including beams or drop panels. The moment of inertia *from* the face of the column (or capital) to the centerline of the column (or capital) is considered *finite* and is dependent on the transverse dimensions of the panel and support. This reduced stiffness (as compared to the infinite stiffness assumed in previous codes) is intended to soften the slab at the joint to account for the flexibility of the slab away from the support. This is consistent with provisions of the Code.²⁰ Figure 2-8 shows the changes in stiffness between a slab, and a drop panel, and a column (or capital).

²⁰ ACI 318-95, 13.7.3; CSA-A23.-94, 13.9.2.3

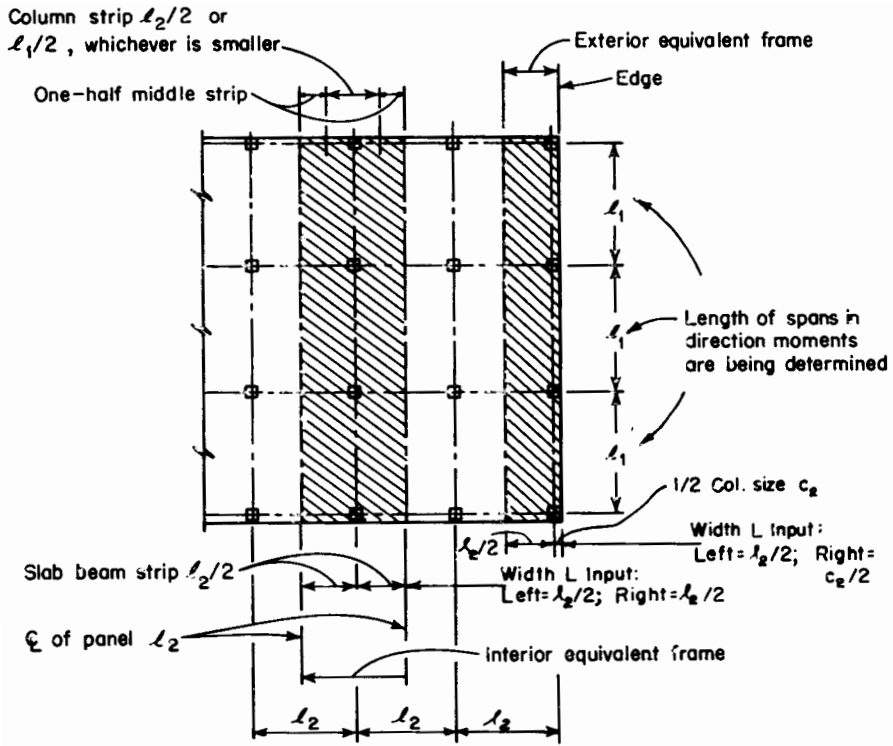


Figure 2-6 Design strips

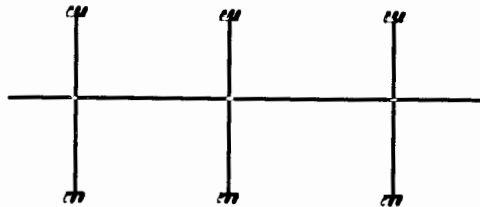


Figure 2-7 Analytical model for vertical loads for a typical story

Columns

The computation of the column stiffness is more complicated as it utilizes the concept of an equivalent column. Theoretical slab studies have shown that the positive moment in a slab may increase under pattern loads, even if rigid columns are used, because of the flexibility of the slab away from the column. However, if a two-dimensional frame analysis is applied to a structure with rigid columns, pattern loads will have little effect. To account for this difference in behavior between slab structures and frames, the equivalent column torsional member, as shown in Figure 2-9, runs transverse to the direction in which the moments are being determined. The transverse slab beam can rotate even though the column may be infinitely stiff, thus permitting moment distribution between adjacent panels. It is seen that the stiffness of the equivalent column is affected by both the flexural stiffness of the columns and the torsional stiffness of the slabs or beams framing into the columns. Note that the method of computation of column stiffness is in accordance with the requirements of the Code.²¹ Figure 2-10 shows a schematic representation of the stiffness of typical columns.

The column stiffness is based on the column height, ℓ_c , measured from mid-depth of the slab above, to the mid-depth of the slab below. ADOSS calculates the stiffness of the column below the design slab, taking into account the design slab system at its top end. ADOSS calculates the stiffness of the column above the design slab taking only the slab depth into account at its top end; column capitals, beams, or drops are ignored.

The computation of the torsional stiffness of the member requires several simplifying assumptions. The first step is to assume dimensions of the transverse torsional slab-beam members. Assumptions for dimensions of typical torsional members are shown in Figure 2-11.

²¹ ACI 318-95, 13.7.4; CSA-A23-94, 13.9

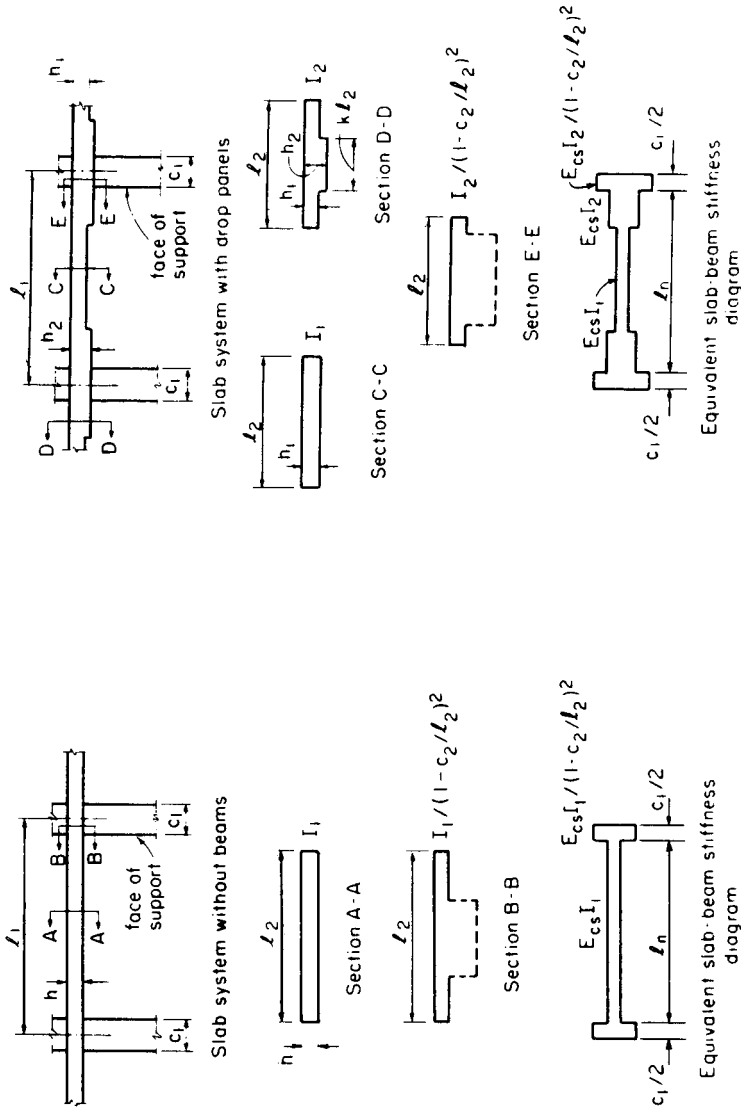


Figure 2-8 Sections for calculating slab-beam stiffness, K_{sb}

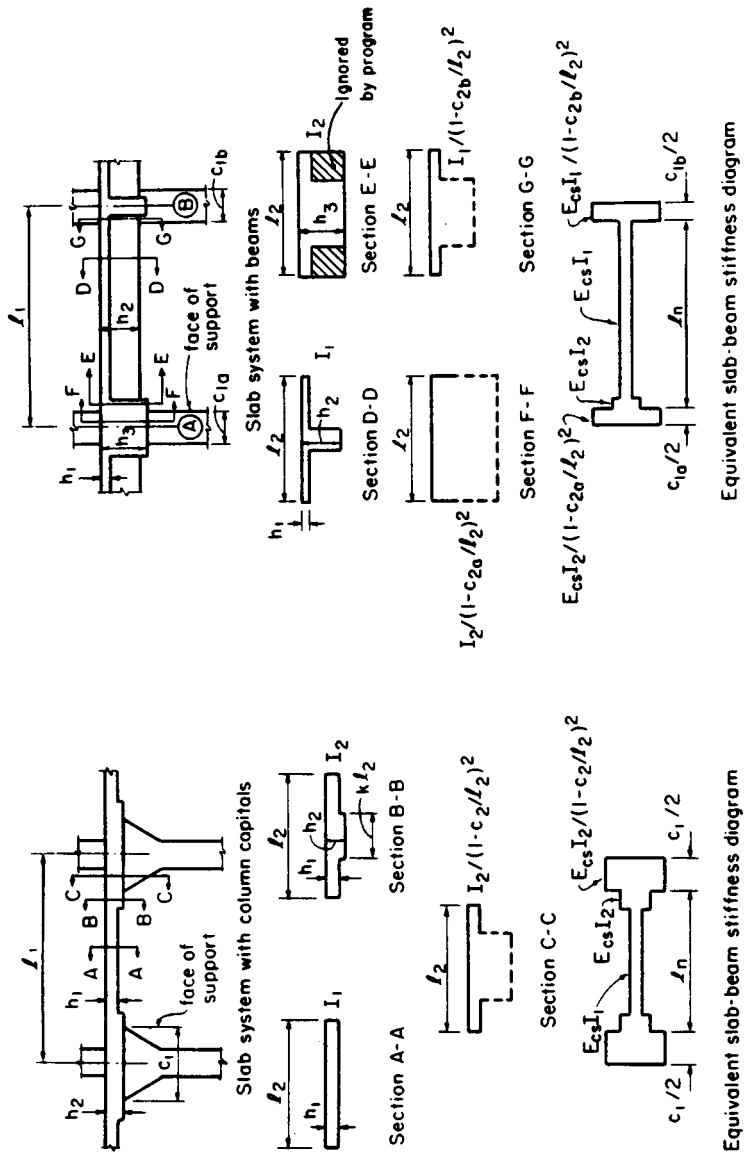


Figure 2-8 continued

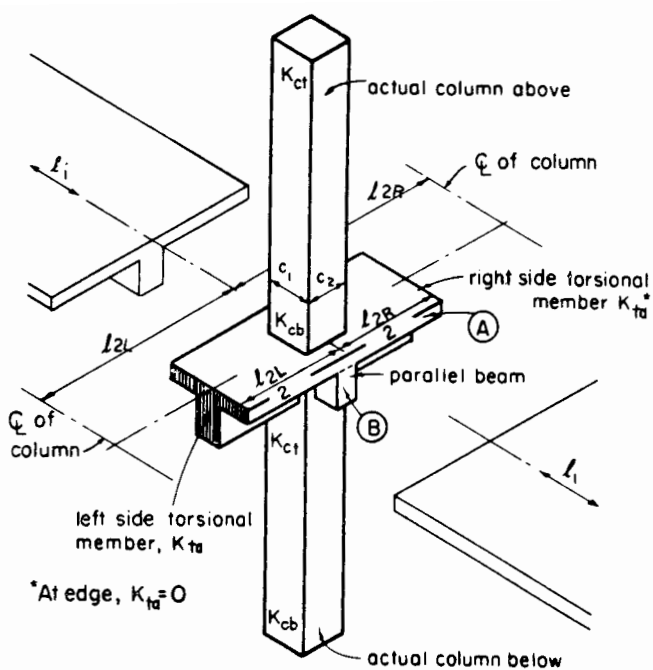


Figure 2-9 The equivalent column

The stiffness, K_t of the torsional member is given by the following expression:²²

$$K_t = \sum \frac{9E_{cs}C}{l_2 \left(1 - \frac{c_2}{l_2}\right)^3} \quad \text{Eq. 2-13}$$

where

- E_{cs} = modulus of elasticity for slab concrete.
- C = cross-sectional constant defining torsional properties; see

²² ACI 318-95, Eq. 13-6; CSA-A23.3-94, Eq. 13.13

Eq. 2-14. It is a conservatively low approximation of the torsional rigidity of rectangular sections when assuming elastic behavior. For the CSA Standard ℓ_2 is taken as the smaller of ℓ_2 or ℓ_1 .

c_2 = size of rectangular column or capital measured transverse to the direction in which moments are being determined.

ℓ_2 = For ACI 318 length of span transverse to ℓ_1 , measured on each side of the column. For CSA A23.3 ℓ_2 is taken as the smaller of ℓ_2 or ℓ_1 .

The constant C is evaluated for the cross section by dividing it into separate rectangular parts and by carrying out the following summation:²³

$$C = \sum \left(1 - 0.63 \frac{x}{y} \right) \frac{x^3 y}{3} \quad \text{Eq. 2-14}$$

where

x = short overall dimension of the rectangular part of a cross section.

y = long overall dimension of the rectangular part of a cross section.

As a result of Eq. 2-13, walls²⁴ running the full width of a slab ($c_2 = \ell_2$) cannot be modeled by the Equivalent Frame Method.

When beams frame into the column in the direction of analysis, the value of K_t as computed in Eq. 2-13 is multiplied by the ratio of the moment of inertia of the slab with the beam (I_{sb}) to the moment of inertia of the slab without the beam (I_s), as shown:

$$K_{ta} = K_t \frac{I_{sb}}{I_s} \quad \text{Eq. 2-15}$$

²³ CSA-A23.3-94, Eq. 13-14

²⁴ Instead walls can be modeled as long supports less than the full design width of the slab. To obtain a uniform distribution of the end moment along the column and middle strips, the width of the wall must be greater than 75% of the design strip.

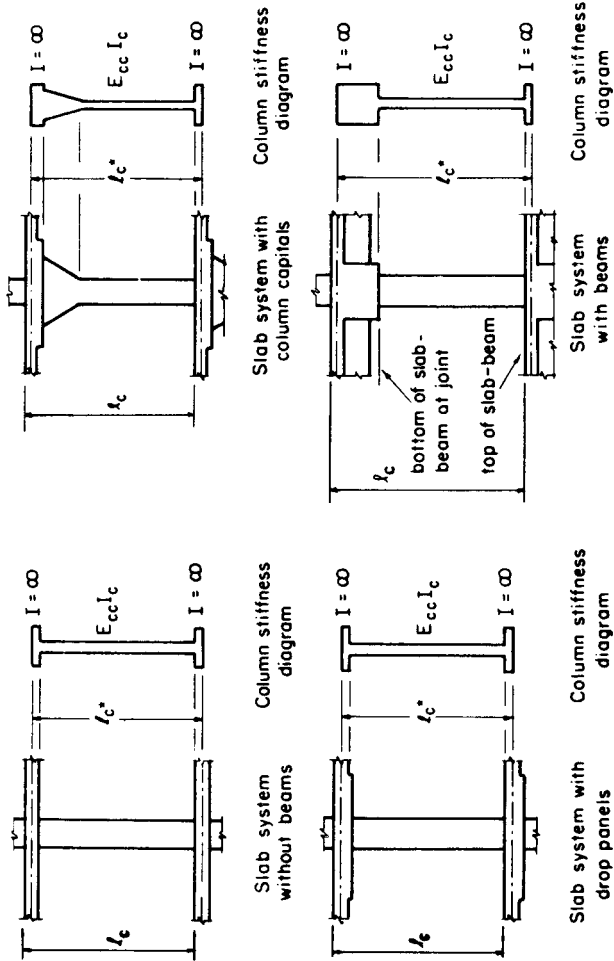


Figure 2-10 Sections for calculating the stiffness (k_c) of the column below the design floor (l_c -input, l_c^* -computed)

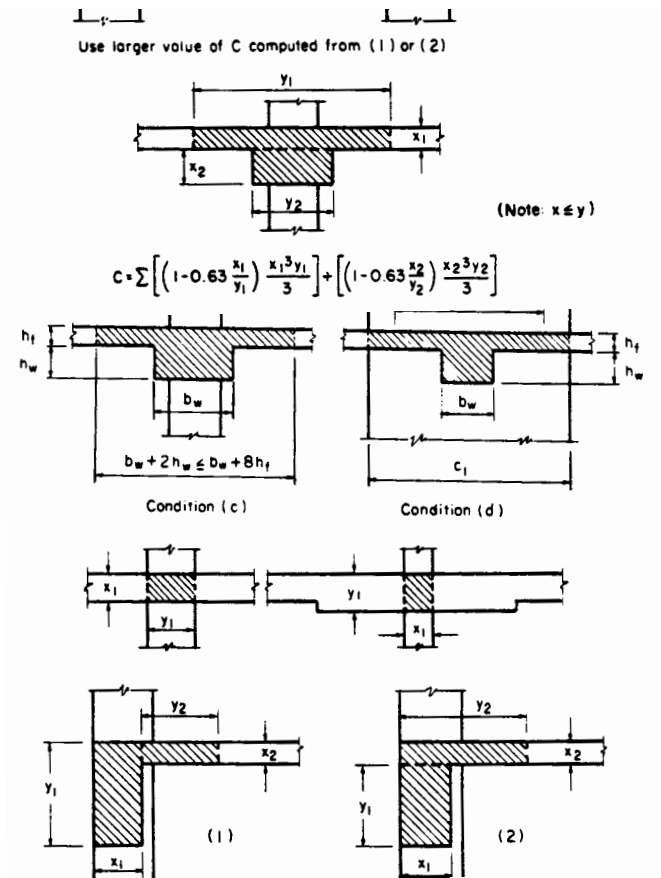


Figure 2-11 Section of the attached torsional members

ADOSS Graphics

With reference to Figure 2-9, I_s is computed from part A, whereas I_{sb} is computed from both parts A and B.

Knowing the column stiffness, K_c , and the stiffness of the attached torsional member, K_t , the stiffness of the equivalent column, K_{ec} , is computed from the equation:

$$K_{ec} = \frac{K_{ct} + K_{cb}}{1 + \frac{K_{ct} + K_{cb}}{K_{ta} + K_{tb}}}$$

Eq. 2-16

Loading

All applied loads are input as unfactored loads. There are no limitations imposed on the ratio of dead to live loads in the Equivalent Frame Method. Results of gravity load and lateral load analyses may be combined, however, the effects of cracking and reinforcement on stiffness must be accounted for in the lateral load analysis.

Self-Weight

The self-weight of the floor system is computed internally by ADOSS. The weights of the slabs, drops, and longitudinal and transverse beams are considered in the self weight computations. Only the concrete weight is considered, the reinforcement weight is ignored. The weight of longitudinal beams is ignored starting at the column centerline, for a length equal to one-half c_1 , the column dimension in the direction of analysis. This will produce slightly less self-weight than actually present for beams wider than c_2 , the column's transverse dimension.

Superimposed Loading

All superimposed vertical loading is considered to act over the entire transverse width of the slab. Vertical loads acting over less than 1/20 of the span length in the direction of analysis will be averaged over 1/20 the span length by ADOSS. Concentrated moments are converted into vertical loads producing a couple which acts over 1/20 the span center-to-center length in the direction of analysis and over the entire transverse width of the slab. It may be convenient to consider surface loading on continuous-beam systems as partial uniform loads. The surface loading on T-beams acts over the entire input slab width, even if it is greater than the code

allowed T-beam flange width. For slab systems with beams, loads supported directly by the beam (such as the weight of the beam stem or a wall supported directly by the beams) are also assumed to be distributed over the entire transverse width of the strip. An additional analysis may be required, with the beam section designed to carry these loads in addition to the portion of the slab moments assigned to the beam.

Lateral Loading

For lateral loads, each frame should be analyzed as a unit for the entire height of the building (Figure 2-12). Computer programs, such as PCA-FRAME, are available for performing such analyses. It should be realized that, for lateral load analysis, slab-beam elements may have a reduced stiffness due to cracking as well as other assumptions made for the effective slab width used for the lateral analysis. The moments obtained from such an analysis may then be input into the analytical model to determine the appropriate design moments under combined vertical and lateral loads.

An alternate but approximate means of computing the effect of lateral loads incorporated in this program is the use of a modified "portal method" to determine the centerline moments. This method however, is *not* recommended since it uses uncracked stiffness for slab-beam elements to resist lateral loads. This modified portal method is based on the following assumptions:

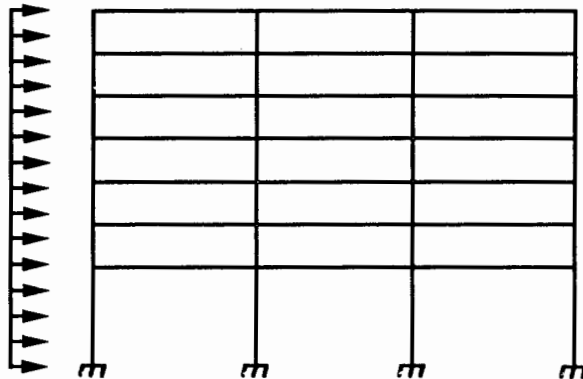


Figure 2-12 Analytical model for lateral loads

ADOSS Graphics

- shear carried by the column is proportional to its flexural stiffness.
- inflection points exist at midheight of columns.

According to the procedure used in ADOSS, the lateral load is divided in proportion to the relative stiffnesses of each column. Moments due to each force are found for columns above and below. Note that for positive lateral loads (acting from left to right) these moments have a positive sign and are balanced at each joint by moments acting on the slab strip. These balancing moments are found on the basis of the stiffness of the slab strips on both sides of the joint. Note that for positive lateral loads (acting from left to right) these moments have a negative sign. Figure 2-13 shows the resulting moments from such an analysis. See Figure 5-1 for the sign convention adopted by ADOSS.

By default, ADOSS distributes the effect of lateral load moment, the difference between the total load moments, vertical plus lateral, and the vertical only moment envelopes, to the column strip and middle strip according to the code distribution factors computed for vertical loads (see Tables 2.4 through 2.6 later in this chapter). However, you may assign a greater percentage of this difference directly to the column strip. The remainder will be distributed to the middle strip and the moment within the column strip will be distributed between the beam and slab according to the code, if applicable.

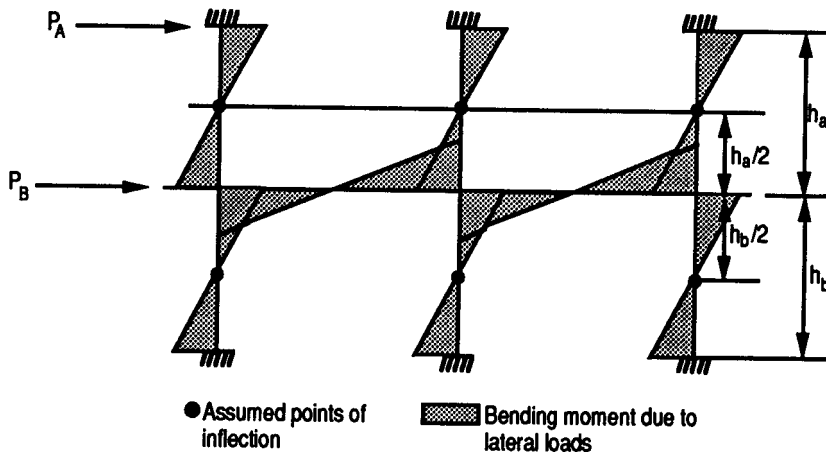


Figure 2-13 Approximate lateral load analysis

Loading Patterns

The analysis of floor systems requires the consideration of several loading combinations. For example, the two adjacent spans loaded may produce the maximum shear stress around a column, while the alternate spans loaded may produce the maximum flexural moments. Furthermore, since the Code²⁵ allows the use of 75% live load for pattern loading, an additional loading pattern with full unreduced live load on all spans may be considered.

For vertical loads, the program selects the design requirements for flexure and shear based on the following loading patterns (Figure 2-14):

- Loading Pattern No. 1: Two adjacent spans loaded with 75% live load.
- Loading Pattern No. 2: Alternate (odd) spans loaded with 75% live load.
- Loading Pattern No. 3: Alternate (even) spans loaded with 75% live load.
- Loading Pattern No. 4: All spans loaded with 100% live load.

When moments due to lateral loads are entered as input, or when concentrated lateral loads are specified, the following additional loading patterns are considered:

- Loading Pattern No. 5: All spans loaded with dead load and the effect of positive lateral loads.²⁶
- Loading Pattern No. 6: All spans loaded with dead load and the effect of negative lateral loads.
- Loading Pattern No. 7: All spans loaded with dead and live load and the effect of positive lateral loads.
- Loading Pattern No. 8: All spans loaded with dead and live load and the effect of negative lateral loads.

²⁵ ACI 318-95, 13.7.6.3; CSA-A23.3-94, 13.9.4

²⁶ A positive lateral load is defined as a horizontal force acting from left to right on the equivalent frame at the left end of the frame. A negative lateral load is defined as a horizontal force acting from right to left on the equivalent frame at the left end of the frame (see Figure 2-13).

LOADING PATTERN NO. 4

Note: 75% of live load is used in Patterns 1-3, while 100% of live load is used in Patterns 4 and 7-8. No live load exists for Patterns 5 and 6.

* Each support in Pattern 1 is loaded as shown for determination of moments and shears

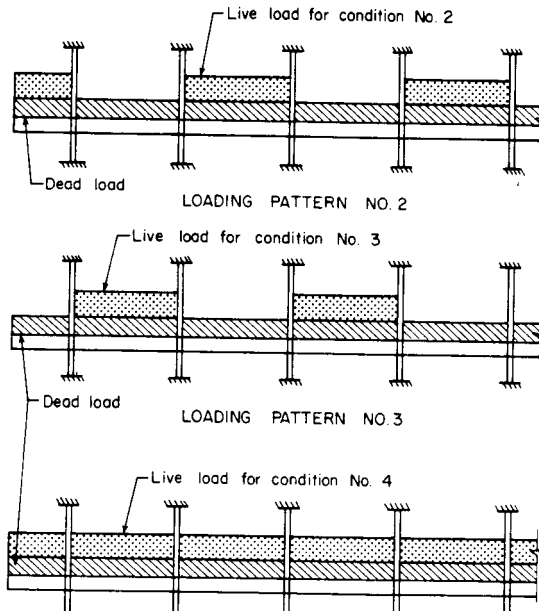


Figure 2-14 Basic pattern live loads

For the ACI 318 Code, the possible combinations of the Dead (D), Live (L), and Wind (W) loads considered by the program are:

$$U_1 = 1.4D + 1.7L \quad \text{Eq. 2-17}$$

$$U_2 = 0.75(1.4D + 1.7L + 1.7W) \quad \text{Eq. 2-18}$$

$$U_3 = 0.75(1.4D + 1.7L - 1.7W) \quad \text{Eq. 2-19}$$

$$U_4 = 0.9D + 1.3W \quad \text{Eq. 2-20}$$

$$U_5 = 0.9D - 1.3W \quad \text{Eq. 2-21}$$

The Eqs. 2-16 through 2-21 show the default values built into the program.

For the CSA-A23.3 Code, the possible combinations of the Dead (D), Live (L), and Wind (W) loads considered by the program are:

$$U_1 = 1.25D + 1.5L \quad \text{Eq. 2-22}$$

$$U_2 = 1.25D + .7(1.5L + 1.5W) \quad \text{Eq. 2-23}$$

$$U_3 = 1.25D + .7(1.5L - 1.5W) \quad \text{Eq. 2-24}$$

$$U_4 = .85D + 1.5W \quad \text{Eq. 2-25}$$

$$U_5 = .85D - 1.5W \quad \text{Eq. 2-26}$$

The Eqs. 2-22 through 2-26 show the default values built into the program.

27 Article 4.1.3.2 of the National Building Code of Canada (NBCC), requires that wind and seismic forces be multiplied by load factors of 1.5 and 1.0, respectively. ADOSS makes no such distinction when dealing with these loads. Unless modified by the user, ADOSS uses the default live load factor of 1.5 for all designs involving the CSA A23.3 Standard. Where these designs include the consideration of seismic loads, a modified seismic load factor of 1.0 should be input by the user.

Column and Middle Strip Widths

The Code²⁸ defines the width of the column strip on each side of the column centerline as being one-fourth of the smaller of either the transverse or the longitudinal span. These widths are printed as part of the echo of input data.

The strip widths at a *support* are computed by (see Figure 2-15)

Column strip

$$W_{cs} = \min \left[\min \left\{ \frac{\ell_{2,1}}{2}, \frac{\ell_1}{4} \right\}_i + \min \left\{ \frac{\ell_{2,r}}{2}, \frac{\ell_1}{4} \right\}_i, \right. \\ \left. \min \left\{ \frac{\ell_{2,1}}{2}, \frac{\ell_1}{4} \right\}_{i+1} + \min \left\{ \frac{\ell_{2,r}}{2}, \frac{\ell_1}{4} \right\}_{i+1} \right] \quad \text{Eq. 2-27}$$

Middle strip

$$W_{ms} = \min \{ \ell_{2,i}, \ell_{2,i+1} \} - W_{cs} \quad \text{Eq. 2-28}$$

The strip widths in the *span* are defined as:

Column strip

$$W_{cs} = \min \left\{ \frac{\ell_{2,1}}{2}, \frac{\ell_1}{4} \right\} + \min \left\{ \frac{\ell_{2,r}}{2}, \frac{\ell_1}{4} \right\} \quad \text{Eq. 2-29}$$

Middle strip

$$W_{ms} = \ell_2 - W_{cs} \quad \text{Eq. 2-30}$$

where

$$\min(a,b) = \text{the smaller of "a" and "b".}$$

28 ACI 318-95, 13.2.1; CSA-A23.3-94, 13.1

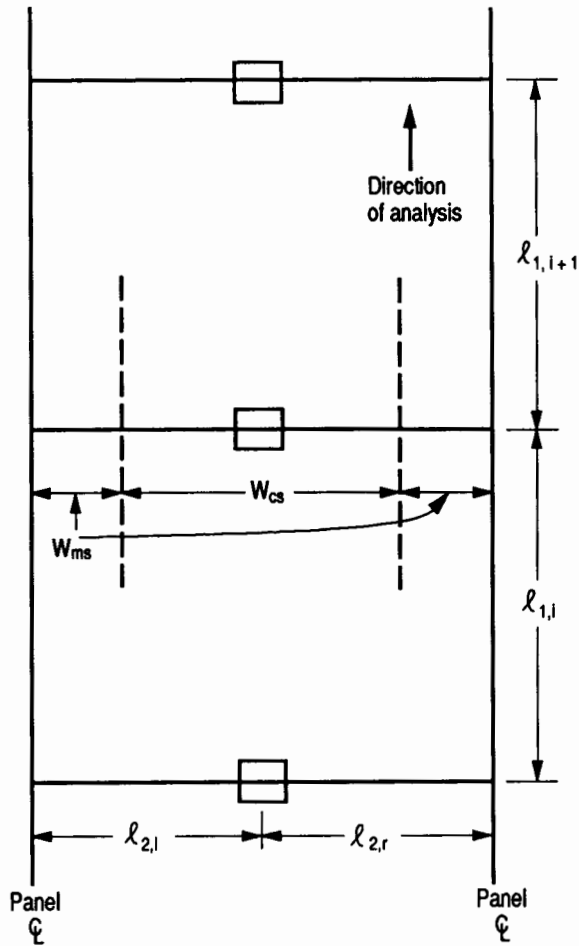


Figure 2-15 Strips widths at support

- l_1 = span length in the direction of analysis.
- l_2 = the total input transverse strip width.
- $l_{2,1}, l_{2,r}$ = the input transverse strip widths on the left and right of column dimension, c_2 , respectively.

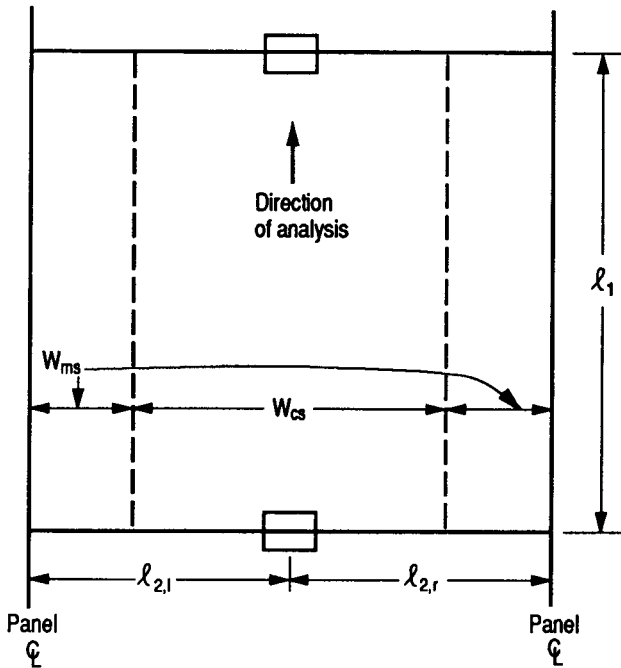


Figure 2-16 Strips widths in span

$i, i+1 =$ span left and right of column dimension c_1 , respectively.

If a beam exists

$$W_{cs} = W_{cs} - W_b \quad \text{Eq. 2-31}$$

If the beam width is greater than the column strip width, W_{cs} , then

$$W_{ms} = W_{ms} - (W_b - W_{cs}) \quad \text{Eq. 2-32}$$

$$W_{cs} = 0 \quad \text{Eq. 2-33}$$

where

$$W_b = \text{beam width.}$$

For exterior frames, the edge width should be specified to the edge of the slab from the column centerline.

Design Moments

ADOSS considers negative moments for design purposes as those producing tension at the top of the slab. The negative design moment is taken at a section located at the face of the column, or column capital, but in no case is it considered at a location greater than 0.175 of the longitudinal span length, l_1 , away from the center of the column.²⁹ This absolute value is a limit on long narrow supports, in order to prevent undue reduction in the design moment. For slab systems with transverse beams, the face of a beam is not considered as the face of support. For end columns with capitals, the moments are taken at the midpoint of the capital extension.³⁰ The column and middle strip moments correspond to the moments assigned to the slab element only.

For ACI designs the column strips are proportioned to resist the portions in percent of interior negative factored moments according to Table 2.4.³¹

Table 2.4 – Column Strip Percent of Interior Negative Factored Moments at Supports

l_2/l_1	0.5	1.0	2.0
$(\alpha_1 l_2/l_1) = 0$	75	75	75
$(\alpha_1 l_2/l_1) \geq 1.0$	90	75	45

The column strips are proportioned to resist the portions in percent of exterior negative factored moments according to Table 2.5.³²

29 ACI 318-95, 13.7.7.1; CSA A23.3-94; 13.9.5.1

30 ACI 318-95, 13.7.7.2; CSA A23.3-94; 13.9.5.2

31 ACI 318-95, 13.6.4.1; CSA3-A23.3-M94, 13.12.2

32 ACI 318-95, 13.6.4.2

Table 2.5 – Column Strip Percent of Exterior Factored Moments at Support

l_2/l_1		0.5	1.0	2.0
$(\alpha_1 l_2/l_1) = 0$	$\beta_t = 0$	100	100	100
	$\beta_t \geq 2.5$	75	75	75
$(\alpha_1 l_2/l_1) \geq 1.0$	$\beta_t = 0$	100	100	100
	$\beta_t \geq 2.5$	90	75	45

The values α_1 in Tables 2.4 and 2.5 and β_t in Table 2.5 are defined as:

α_1 = ratio of flexural stiffness of the beam section to flexural stiffness of a width of slab bounded by centerlines of adjacent panels (if any) on each side of the beam in the direction of analysis. For flat plates, flat slabs, and waffle $\alpha_1 l_2/l_1 = 0$.

β_t = ratio of torsional stiffness of an edge beam section to flexural stiffness of a width of slab equal to the span length of the beam, center-to-center of supports;³³ see Eq. 2-34. When no transverse beams are present, $\beta_t = 0$.

$$\beta_t = \frac{E_{cb}C}{2E_{cs}I_s} \tag{Eq. 2-34}$$

where

E_{cb} = modulus of elasticity of beam concrete.

E_{cs} = modulus of elasticity of slab concrete.

C = cross-sectional constant to define torsional properties; see Eq. 2-14.

I_s = moment of inertia of the gross section of the slab about its centroidal axis.

³³ ACI 318-95, 13.0; CSA3-A23.3-94, 13.0

Linear interpolation is performed between values. When a column width, c_2 , is equal to or greater than 75 percent of the strip width, ℓ_2 the negative moment is uniformly distributed across ℓ_2 .³⁴

When designing by the CSA A23.3 Standard, a portion of the total positive or interior negative moment equivalent to:

$$\frac{\alpha_1}{[1 + (\ell_2/\ell_1)^2]} \qquad \text{Eq. 2-34a}$$

is resisted by the beam.

For exterior supports the beam is proportioned to resist 100% of the negative moment.

That portion of the moment not resisted by the beam is resisted by the slab. The reinforcement required to resist this moment is distributed evenly across the slab.

When lateral loads are present, ADOSS, by default, distributes the effects of lateral loads according to Tables 2.4 and 2.5. You may optionally distribute a greater percentage of the difference between the total load moment and vertical only load moment directly to the column strip. In no case, however, will ADOSS distribute less lateral load moment to the column strip than that required by Tables 2.4 and 2.5, for ACI, or equation 2-34a for CSA.

The middle strips are proportioned to resist the portion of the total factored moments that is not resisted by the column strips.

For ACI designs the longitudinal beams are proportioned to resist 85 percent of the column strip moments if $\alpha_1 \ell_2 / \ell_1$ is equal to or greater than 1.0. For values of $\alpha_1 \ell_2 / \ell_1$ between 0 and 1.0, the beam is designed to resist a proportionate percentage of the column strip moment between 0 and 85%.³⁵

For design purposes, ADOSS computes the amount of reinforcement for the moments on the left and right sides of the support. The negative design moment is thus the moment which requires the most area of reinforcement to be resisted. The location, left or right of the support, of the maximum moment may vary when systems differ on each side of the support (for example, a system with beams on one side only). ADOSS considers positive moments for design purposes as those producing tension at the bottom of the slab. The column strips are proportioned to

34 ACI 318-95, 13.6.5

35 ACI 318-95, 13.6.5

resist the portions in percent of positive factored moments according to Table 2.6.³⁶

For ACI designs the column strips are proportioned to resist the portions in percent of factored vertical load moments along the span according to Table 2.6.³⁶

Table 2.6 – Column Strip Percent of Positive Factored Moments

$\ell_2 \ell_1$	0.5	1.0	2.0
$(\alpha_1 \ell_2 \ell_1) = 0$	60	60	60
$(\alpha_1 \ell_2 \ell_1) \geq 0$	90	75	45

Linear interpolation is performed between values. Note, for flat plates, flat slabs, and waffle slabs, $\alpha_1 \ell_2 \ell_1 = 0$.

When lateral loads are present, ADOSS, by default, distributes the effects of lateral loads according to Tables 26. You may optionally distribute a greater percentage of the difference between the total load moment and vertical only load moment directly to the column strip. In no case, however, will ADOSS distribute less lateral load moment to the column strip than that required by Tables 2.6.

The middle strips are proportioned to resist the remainder of the total factored moments.

The longitudinal beams are proportioned to resist 85 percent of the column strip moments if $\alpha_1 \ell_2 \ell_1$ is equal to or greater than 1.0. For values of $\alpha_1 \ell_2 \ell_1$ between 0 and 1.0, the beam is designed to resist a proportionate percentage of the column strip moment between 0 and 85.³⁷

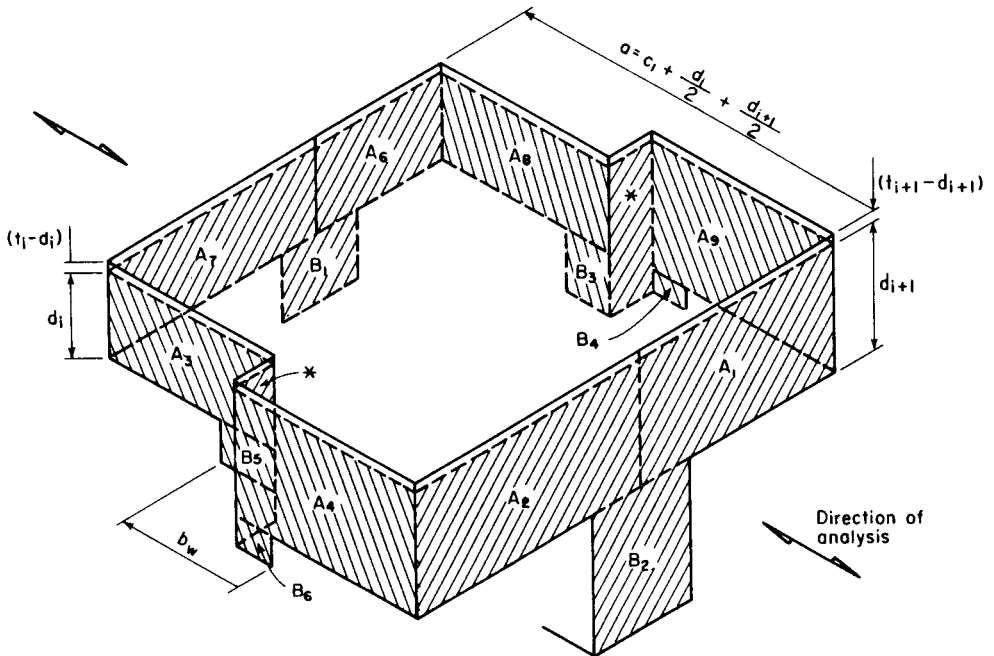
³⁶ ACI 318-95, 13.6.4.4

³⁷ ACI 318-95, 13.6.5

Shear Analysis of Slabs

Three types of shear can occur on slab systems: wide beam shear, punching shear, and moment transfer shear. Wide beam shear or one-way shear on two-way systems must be checked manually. This section discusses punching shear including moment transfer shear. It is assumed that both types of shear act on the same critical section. The critical section for shear is defined in the Code.³⁸

Figure 2-17 shows the general shear area used by ADOSS. Note that the shaded area represents the general case and is modified for special considerations as explained below.



* Not considered by program.

Figure 2-17 Critical section for shear at column

38 ACI 318-95, 11.12.1.2; CSA-A23.3-94, 13.4.3

ADOSS Graphics

Shallow beams are considered in the unbalanced moment transfer as indicated in Figure 2-17. Ordinarily, transverse beams transfer unbalanced moment to the column through torsion along the beam and not through shear between the slab and column. However, the Code leaves the transfer method to the engineer’s judgment concerning the point at which punching shear is no longer applicable and beam shear becomes the dominate element in shear transfer to the column. In the default mode of ADOSS, the program makes no such distinction and computes unbalanced moment transfer stress without regard to any beams framing into the column. It is possible, however, to have ADOSS distribute the shear and torsion among the framing beams. Although the depth of the beam is considered in the critical section surfaces, the distances to the critical section are *not* increased at the intersection with any beams. This approach is conservative. ADOSS does not compute torsional stresses or one-way beam shear stresses in the slab. If in the engineer’s judgment these may control, they must be computed manually.

For a circular column or column capital, a square shape with an equivalent area is assumed as shown in Figure 2-18. Critical section area for punching, A_c , is then multiplied by $\pi(D + d)/(2D \sqrt{\pi + 4d})$ to account for the difference between circular and square critical sections, where D is the diameter of a column or a column capital and “ d ” is the effective depth of the slab.

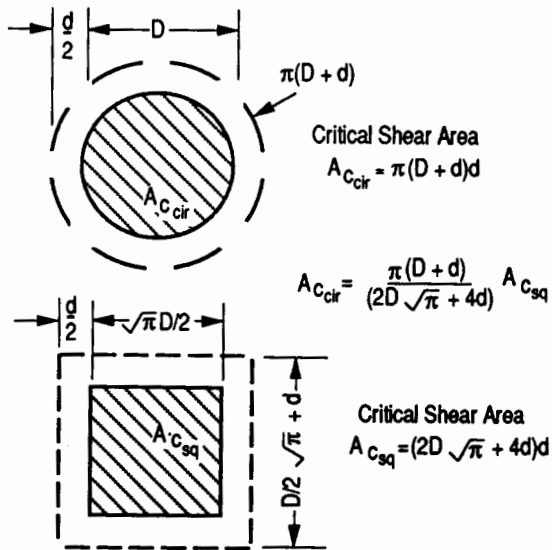


Figure 2-18 Critical section area for circular column

Critical Section for Interior Supports of Interior Frames

The critical section (Figure 2-19) consists of four vertical surfaces through the slab, located at distances of $d/2$ beyond the support faces.

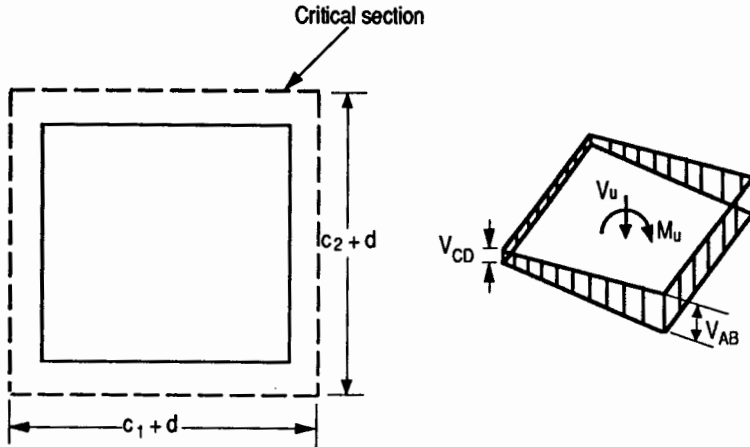


Figure 2-19 Interior supports of interior frames

For a critical section to be “closed,” the concrete slab around a column must extend to a distance greater than or equal to ten times the slab depth, $10h$, beyond the column face. Otherwise the section is considered “open.” The critical section for interior supports of interior frames is always “closed.” A “closed” section will have all its faces defined in Figure 2-15, resisting shear as indicated by Eq. 2-35.

$$A_c = \sum_{i=1}^8 A_i \tag{Eq. 2-35}$$

If beams frame³⁹ into the column, then the critical section includes the dimensions of the beams (B_1 through B_8 in Figure 2-15).

³⁹ A beam is considered as framing into the column if the beam is within a face of the column.

Critical Section for Exterior Supports of Interior Frames

The critical section for exterior supports of interior frames (Figure 2-20) will be either closed (full A_7 and A_6 for the first column or A_2 and A_3 for the last column in Figure 2-17) or open, depending upon the length of the cantilever in relation to slab thickness. The critical section will be considered "closed" when the clear cantilever span, ℓ_c , is greater than or equal to ten times the slab depth, $10h$, beyond the column face. If beams frame into the column, then the critical section includes the contributions from the beam dimensions (B_2 through B_6 in Figure 2-17).

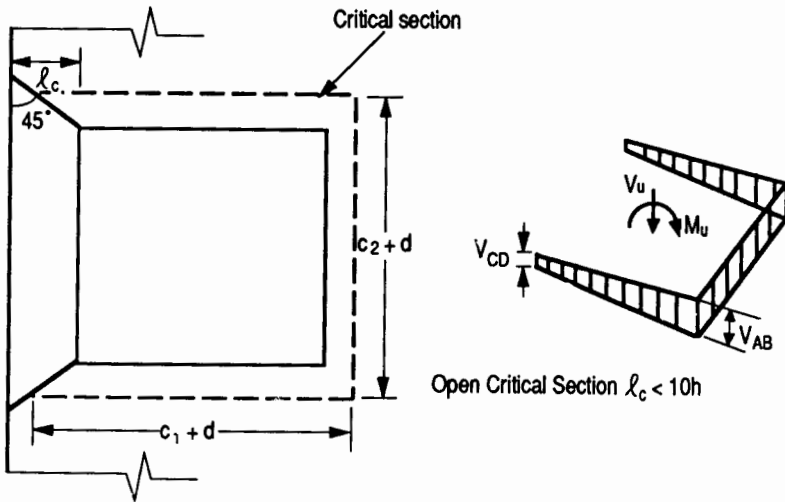


Figure 2-20 Exterior supports of interior frames

Critical Section for Interior Supports of Exterior Frames

Figure 2-21 shows the critical section for shear for an interior support of an exterior frame. Note that the section is considered as U-shaped ($A_1 = 0$, $A_8 = 0$, $B_1 = 0$, $B_8 = 0$ in Figure 2-17) and it extends up to the edge of the exterior face of the support. If beams frame into the column, then the critical section includes the contribution from the beam dimensions (B_2 through B_7 in Figure 2-17). If the exterior cantilever span is greater than or equal to ten times the slab depth, $10h$, beyond the column face, the section is treated as "closed", that is, the support is treated as an interior support of an interior frame.

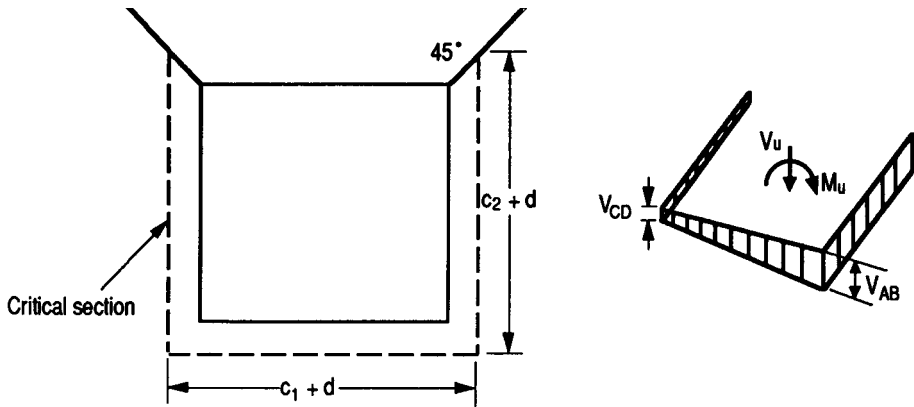


Figure 2-21 Interior supports of exterior frames

Critical Section for Exterior Supports of Exterior Frames

The critical section for an exterior support of an exterior frame (Figure 2-22) will typically be L-shaped ($A_1 = 0, A_6 = 0, A_7 = 0, A_8 = 0, B_1 = 0, B_6 = 0, B_7 = 0$ and $B_8 = 0$ in Figure 2-17).

If ℓ_c , the clear cantilever span, is greater than or equal to ten times the slab depth, $10h$, beyond the column face, then the section is treated as a U-shaped interior support. If, in addition, the exterior cantilever span is greater than or equal to ten times the slab depth beyond the column face, $10h$, the section is treated as closed. If beams frame into the column, then the critical section includes the contributions from the beam dimensions.

Computation of Allowable Shear Stress at Critical Section

Two-way shear strength of slabs is affected by concrete strength, relationship between size of loaded area and slab thickness, loaded area aspect ratio, perimeter area aspect ratio (only for ACI 318-89 code option), and shear-to-moment ratio at slab-column connections.

For the ACI 318-95 and CSA A23.3-94 Code, these variables are taken into account in the allowable shear stress Eqs. 2-36 through 2-39. The allowable shear stress used by ADOSS is computed at distances of $d/2$ around the columns and drops (if applicable) and taken as the smallest of the 3 quantities:⁴⁰

$$v_c = \left(2 + \frac{4}{\beta_c}\right) \sqrt{f'_c} \tag{Eq. 2-36}$$

$$v_c = \left(2 + \frac{\alpha_s d}{b_o}\right) \sqrt{f'_c} \tag{Eq. 2-37}$$

$$v_c = 4 \sqrt{f'_c} \tag{Eq. 2-38}$$

where

- β_c = the ratio of the long to the short side of the column.
- α_s = a constant dependent of the column location, (40 for an interior 4-sided effective critical area, 30 for an exterior 3-sided critical area, 20 for a corner 2-sided effective critical area.)
- d = distance from the slab bottom to centroid of the slab reinforcement at support.
- b_o = the perimeter of the critical section.

⁴⁰ ACI 318-95, 11.12.2.1; CSA-A23.3-94, 13.4.4

The allowable shear stress around drops is computed as:

$$\text{ACI } v_c = 2\sqrt{f'_c} \quad \text{Eq. 2-39}$$

$$\text{CSA } v_c = 0.2\phi_c \sqrt{f'_c}$$

For the ACI 318 Code, if f_{ct} is specified or for concrete types other than normal weight, f_{ct} will be substituted for $\sqrt{f'_c}$ in Eqs. 2-36 through 2-39.⁴¹ For waffle slab systems with valid ribs defined earlier in this chapter, the allowable shear stress is increased by 10% for ACI designs.⁴²

For CSA designs an additional 1-way shear check is made at end supports of exterior frames in accordance with CSA A23.3-94 Clause 12.4.6.2

Computation of Factored Shear Force at Critical Section

The factored shear force on the critical section, V_u , by default is computed as the reaction at the centroid of the critical section (e.g., column centerline for interior columns) minus the self-weight and any superimposed surface dead and live load acting within the critical section. Any partial loads, (uniform, trapezoidal, concentrated), that act within the critical section are not excluded. If the section is considered "open", two 45 degree lines are drawn from the column corners to the nearest slab edge (lines AF and DE in Figure 2-21) and the self-weight and superimposed surface dead and live loads acting on the area ADEF are also omitted from V_u (Reference 32).

If beams frame into the column and proportioned moments and shears are requested, ADOSS computes the optional factored shear force, V_u , assuming that the load is distributed to the supporting beams. The factored shear force in the beams with an $\alpha_1 \ell_2 / \ell_1$ greater than 1 will be computed from the loads on their tributary areas. These areas are bounded by two 45 degree lines, originating from the centers of the columns at the two ends of the beam span, that extend to the edge of the design strip or to the center of the transverse span (see Figures 2-22 through 2-23). The shear in the beams will be reduced by $(1 - \alpha_1 \ell_2 / \ell_1)$ for beams with $\alpha_1 \ell_2 / \ell_1$ less than 1.

41 ACI 318-95, 11.2.1.1

42 ACI 318-95, 8.11.8

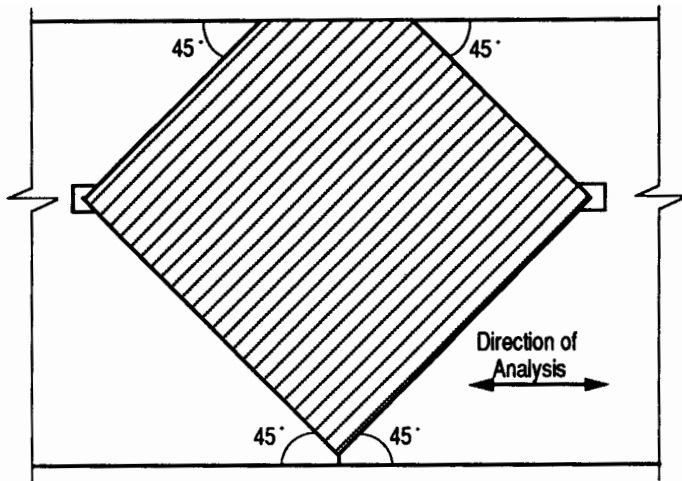


Figure 2-24 Tributary area for shear on longitudinal beams

Computation of Unbalanced Moment at Critical Section

The factored unbalanced moment used for shear transfer, M_{unbal} , by default is computed as the sum of the joint moments to the left and right, taken to the centroidal axis of the critical section.

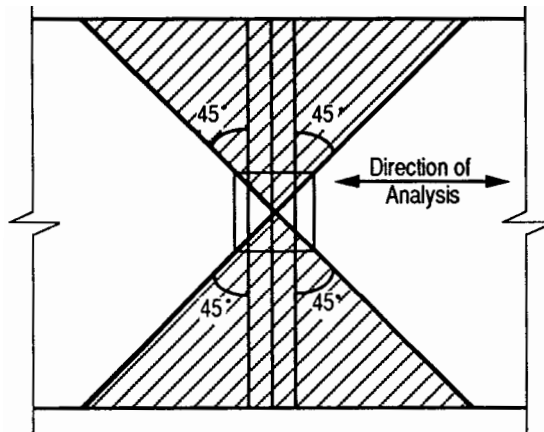


Figure 2-25 Tributary area for shear on transverse beams

If beams frame into the column and proportioned moments and shears are requested, ADOSS computes the factored unbalanced moment, assuming that the beams resist a portion of the moment. The beams in the direction of analysis are assumed to attract the percentage of unbalanced moment based on its $\alpha_1 l_2 / l_1$ value as defined in the design moments. The transverse beams are assumed to attract the remaining unbalanced moment when the β_1 value, is greater than or equal to 2.5.

The unbalanced moment applied to the transverse beams is reduced by $\beta_1 / 2.5$ for beams with β_1 less than 2.5 and the remainder of the unbalanced moment is considered directly transferred from the slab to the column.

Computation of Shear Stresses at Critical Section

The punching shear stress printed by the program is based on the following:⁴³

$$v_u = \frac{V_u}{\phi A_c} \qquad \text{Eq. 2-40}$$

where

- V_u = factored shear force on the critical section described above.
- A_c = area of concrete, including beams if any, resisting shear transfer.
- ϕ = capacity reduction factor as per the Code = .85.

Under conditions of combined shear, V_u , and an unbalanced moment, M_{unbal} , $\gamma_v M_{unbal}$ is assumed to be transferred by eccentricity of shear about the centroidal axis of the critical section. The shear stress printed by the program (Reference 2-33) for this condition corresponds to:⁴⁴

$$v_{AB} = \frac{V_u}{\phi A_c} + \frac{\gamma_v M_{unbal} c_{AB}}{\phi J_c} \qquad \text{Eq. 2-41}$$

where

- M_{unbal} = factored unbalanced moment transferred directly from slab to column, as described above.
- γ_v = is the fraction of unbalanced moment considered transferred by eccentricity of shear about the centroid of the assumed critical section.⁴⁵

$$\gamma_v = (1 - \gamma_f) \qquad \text{Eq. 2-42}$$

43 ACI 318-95, 11.12.6.2; CSA-A23.3-94, 13.4.5

44 ACI 318-95, 11.12.6.1; CSA-A23.3-94, 13.4.5.5

45 ACI 318-95, Eq. 11-42; CSA-A23.3-94, Eq. 13-8

where

$$\gamma_f = \frac{1}{1 + (2/3)\sqrt{b_1/b_2}}$$

- J_c = property of the assumed critical section analogous to polar moment inertia.
- b_1 = width of critical section in the direction of analysis.
- b_2 = width of critical section in the transverse direction.

Direct shear and moment transfer cause stresses in opposite direction on one face of the critical section. ADOSS does not check for the shear stress on this face. If the stress due to direct shear is less than 20% of the allowable shear stress, and if the unbalanced moment transferred by shear is greater than 75% of that required for moment transfer by flexure only, the engineer should manually check⁴⁶ the shear stress (v_{CD}) on this other face of the critical section.

Local effects of concentrated loads are not computed by ADOSS and must be calculated manually.

Shear Analysis of Longitudinal Beams

When longitudinal beams are present in a span, ADOSS will compute the shear reinforcement requirements for the beams. When the floor is a continuous beam system or when proportioning of the moments and shears is not selected for the beam-supported slab system, ADOSS computes V_u from the load acting over the entire width of the design strip. The program makes no distinction between shallow beams ($\alpha_1 \ell_2 \ell_1$ less than 1) and deeper beams ($\alpha_1 \ell_2 \ell_1$ greater than 1).

When proportioning of the moments and shears is selected for the beam supported slab system, the beams are proportioned to resist shear caused by loads on tributary areas. These areas are bounded by two 45 degree lines, originating from the center of the column at one end of the beam span, that extend to the edges of the design strip or meet with 45 degree lines drawn from the center of the column at the other

⁴⁶ Based on extensive investigation by N. Hawkins at the University of Washington. For detailed information, see ACI Special Publication SP-42.

end of the beam, see Figure 2-22. The shear applied to the beams will be reduced by $1 - \alpha_1 \ell_2 / \ell_1$ for beams with $\alpha_1 \ell_2 / \ell_1$ less than 1. Proportioning of moments and shears has no effect on continuous beam systems.

The “Beam Shear Requirements” table provides A_v/s values, the required area of two legs of a stirrup, divided by the stirrup spacing. The values are given at a distance “d”, the effective beam depth, away from the column face at both the left and the right ends of the beam. Values are also given at distances of .175, .375, .625, and .825 times the center-to-center span length, ℓ_1 away from the left support.

For ACI 318 Code:

In beams, the shear strength provided by concrete, V_c , is computed by:⁴⁷

$$V_c = 2\sqrt{f'_c} b_w d \quad \text{Eq. 2-43}$$

In CSA design, for beams without minimum stirrup reinforcement and greater than 300mm deep, v_c is calculated from the following equation:

$$V_c = \left(\frac{260}{1000 + d} \right) \lambda \phi_c \sqrt{f'_c} b_w d \leq 0.10 \lambda \phi_c \sqrt{f'_c} b_w d$$

When $V_u > \phi V_c / 2$, the beam must be provided with at least a minimum shear reinforcement of:⁴⁸

$$\frac{A_v}{s} = \frac{50b_w}{f_{yv}} \quad \text{Eq. 2-44}$$

47 ACI 318-95, 11.3.1.1; CSA-A23.3-94, 11.3.5.1

48 ACI 318-95, 11.5.5.2; CSA-A23.3-94, 11.2.8.4

where

- A_v = area of two legs of a stirrup,
- s = stirrup spacing,
- b_w = longitudinal beam width, and
- f_{yv} = yield strength of the shear reinforcement.

When $V_u > \phi V_c$, shear reinforcement must be provided so that:

$$\frac{A_v}{s} = \frac{V_u - \phi V_c}{\phi f_{yv} d} \quad \text{Eq. 2-45}$$

where

- V_u = factored shear force at the section being considered, and
- d = effective depth of the beam at the same location.

When V_u exceeds $\phi 5 V_c$, the beam section dimensions must be increased or a higher concrete strength must be provided.⁵⁰

When segmental moments and shears are requested, ADOSS will print stirrup spacing for four bar sizes. When $V_u \leq \phi 10 \sqrt{f'_c} b_w d$, the spacing is computed as:

$$s = \frac{1}{\frac{A_v}{s}} A_{sb} \quad \text{Eq. 2-46}$$

where

- A_{sb} = total bar area of the two legged stirrup.

The maximum stirrup spacing must not exceed $d/2$ nor 24 in. when $V_u \leq \phi 6 \sqrt{f'_c} b_w d$. When $V_u > \phi 6 \sqrt{f'_c} b_w d$, the maximum stirrup spacing must be reduced by half, to $d/4$ or 12 in.⁵¹ When $V_u > \phi 10 \sqrt{f'_c} b_w d$, the beam section dimensions must be increased or a higher concrete strength must be provided.

49 ACI 318-95, 11.5.6.1; CSA-A23.3-94, 11.3.7

50 ACI 318-95, 11.5.6.8; CSA-A23.3-94, 11.3.4

51 ACI 318-95, 11.5.4; CSA-A23.3-94, 11.2.11

Shear and Torsion Analysis of Transverse Beams

When transverse beams frame into a column, ADOSS will compute the shear and torsional requirements for each beam at a distance d , the transverse beam effective depth, away from the transverse face of the column. The distance to the top reinforcement from the tension face in the transverse beam is taken as that specified for the top reinforcement in the slabs.

The shear applied to each transverse beam, V_u , is computed from loads on the beam tributary area. These areas are bounded by two 45 degree lines, originating from the center of the column, that extend to the edge of the design strip or to the centerlines of the panels parallel to the long side (Figure 2-25).

The torque applied to each transverse beam is computed from the unbalanced moment as documented below:

$$T_{u,l} = \left(\frac{K_{t,l}}{K_{t,l} + K_{t,r}} \right) M_{\text{unbal}} \quad \text{Eq. 2-47}$$

$$T_{u,r} = \left(\frac{K_{t,r}}{K_{t,l} + K_{t,r}} \right) M_{\text{unbal}} \quad \text{Eq. 2-48}$$

where

$T_{u,l}, T_{u,r}$ = transverse beam torques on the front and back faces of the column, respectively.

$K_{t,l}, K_{t,r}$ = the torsional stiffness of the transverse beam on the front and back faces of the column, respectively.

M_{unbal} = the unbalanced moment at the center of the column.

The torque is assumed to be linearly distributed along the transverse beam. The maximum torque occurs at the joint center, and zero torque occurs at the slab panel center, transverse to the design direction (Figure 2-26).

When proportioning of the moments and shears is selected, the shear, V_u , resisted by the beams will be reduced by $1 - \alpha_1 \ell_2 \ell_1$ for beams with an $\alpha_1 \ell_2 \ell_1$ less than 1. The

torque applied to the beams will be reduced by $\beta_t/2.5$ for beams with β_t less than 2.5, see Eq. 2-34.

When the applied torque is less than $\phi \sqrt{f'_c} (A_{cp}^2/P_{cp})$ no torsional reinforcement is required and the shear reinforcement for the transverse beam is designed using the procedure for longitudinal beams.⁵³

The shear reinforcement for the transverse beam is designed using the procedure for longitudinal beams.⁵⁴

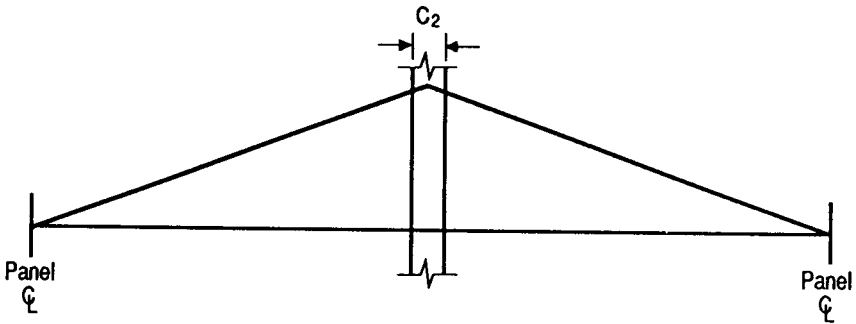


Figure 2-26 Distribution of torque in transverse beams

If the applied shear, V_u , exceeds $\phi V_c/2$, but is less than ϕV_c and/or the applied torque, T_u , is greater than $\phi \sqrt{f'_c} (A_{cp}^2/P_{cp})$ minimal shear and torsional reinforcement must be provided as follows:⁵⁵

$$\frac{A_v}{s} + \frac{2A_t}{s} \geq \frac{50B_w}{f_w} \quad \text{Eq. 2-52}$$

53 ACI 318-95, 11.3; CSA-A23.3-94, 11.3

54 ACI 318-95; 11.5; CSA-A23.3-94, 11.3.7

55 ACI 318-95; 11.6.5.2

The total stirrup reinforcement area is computed by:

$$\frac{A_{\text{tot}}}{s} = \frac{A_v}{s} + \frac{2A_t}{s} \quad \text{Eq. 2-53}$$

where

$$\begin{aligned} A_v &= \text{area of two legs of a closed stirrup,} \\ A_t &= \text{area of one leg of a closed stirrup.} \end{aligned}$$

When T_u exceeds $\phi \sqrt{f'_c} (A_{cp})^2/P_{cp}$, the required area of one leg of a closed stirrup divided by its spacing, A_t/s , is computed by:⁵⁶

$$\frac{A_t}{s} = \frac{T_n}{2A_o f_y \cot \phi} \quad \text{Eq. 2-54}$$

where

$$\cot \phi = 1$$

When the applied torque exceeds $\phi \sqrt{f'_c} (A_{cp})^2/P_{cp}$, the cross-sectional dimensions shall be such that:⁵⁷

$$\sqrt{\left(\frac{V_u}{b_w d}\right)^2 + \left(\frac{T_u P_h}{1.7 A_{oh}}\right)^2} \leq \phi \left(\frac{V_c}{b_w d} + 8 \sqrt{f'_c}\right) \quad \text{Eq. 2-55}$$

The required area for longitudinal torsion reinforcement A_s is taken as:⁵⁸

$$A_s = \frac{A_t}{s} P_h \left(\frac{f_{yv}}{f_{yt}}\right) \cot^2 \theta \quad \text{Eq. 2-56}$$

The maximum value for A_t/s in Eq. 2-56 is taken as the value obtained from Eq. 2-54.

⁵⁶ ACI 318-95, 11.6.3.6

⁵⁷ ACI 318-95, 11.6.3.1; CSA A23.3-94, 11.3.9.8

⁵⁸ ACI 318-95, 11.6.3.7; CSA A23.3-94, 11.3.9.5

Area of Reinforcement

ADOSS only computes reinforcement at the top of the floor system at the supports and at the bottom of the floor system in the span. If reinforcement is required at the bottom of the floor system at a support or at the top of the floor system in the span, it must be designed manually based on the design moments provided. The reinforcement at the support is computed from the negative design moment which requires the most area of reinforcement to be resisted. The location, left or right of the support, of the maximum moment may vary when systems differ on each side of the support for example, a system with beams on one side only).

For rectangular sections with no compression reinforcement, the design flexural strength of the column strip, middle strip or beam must equal the factored design moment:

ACI 318-95

$$M_u = \phi f_y A_s \left(d - \frac{A_s f_y}{2(0.85 f'_c) b} \right) \quad \text{Eq. 2-57}$$

The reinforcement can therefore be computed from:

$$A_s = \frac{0.85 f'_c b}{f_y} \left(d - \sqrt{d^2 - \frac{2M_u}{\phi \cdot 0.85 f'_c b}} \right) \quad \text{Eq. 2-58}$$

For CSA A23.3-94

$$M_r = \phi_s f_y A_s \left(d - \frac{\phi_s A_s f_y}{2(\alpha_1 \phi_c f'_c) b} \right)$$

The effective depth of the section is taken as the overall section depth minus the distance from the extreme tension fiber to the tension reinforcement centroid. The column strip depth may include all or part of the drop panel depth. The drop depth will not be included in the effective depth of the column strip when the drop does

not extend at least one-sixth the center-to-center span length in all directions, or when the drop depth below the slab is less than one-quarter the slab depth. If the drop does extend at least one-sixth the center-to-center span length and the drop depth is greater than one-quarter the distance from the edge of the drop panel to the face of the column or column capital, the excess depth will not be included in the column strip effective depth. If the drop width is less than the column strip width, the drop width will be used in the computation of the required reinforcement.

Continuous beam systems may be designed as T-beams and may have both tensile and compressive reinforcement.

In all cases, the steel ratio will be kept below 75 percent ρ_b , where:⁵⁹

$$\rho_b = .85\beta_1 \frac{f'_c}{f_y} \frac{87}{87 + f_y} \quad \text{Eq. 2-59}$$

where

$$\begin{aligned} \rho_b &= 0.85 \text{ for } f'_c \leq 4 \text{ ksi} \\ &= 0.65 \text{ for } f'_c \geq 8 \text{ ksi} \\ &= 1.05 - .05 \times f'_c \text{ for } 4 \text{ ksi} < f'_c < 8 \text{ ksi} \end{aligned}$$

If the slab cannot be reinforced, a “cannot design system” message will appear and the program will terminate prematurely to indicate the need for compression reinforcement. In such cases, it is recommended that the engineer review the slab thickness to ensure a more satisfactory design. For continuous beam systems, ADOSS will attempt to add compression reinforcement to the section.

The amount of reinforcement provided will not be less than the code prescribed minimum. For the ACI 318 Code, the minimum ratio of reinforcement area, to the gross sectional area of the slab strip, using Grade 60 reinforcement, is taken as .0018. When reinforcement yield strength exceeds 60 ksi, the minimum ratio is set to $.0018 \times 60/f_y$. For reinforcement with yield strength less than 60 ksi, the minimum ratio is set to 0.0020. In no case will this ratio be less than $.0014^{60}$. (See Table 2.7). The CSA Standard requires a minimum ratio of slab reinforcement area to gross sectional area of the slab strip equal to 0.002 for all grades of reinforcement.

⁵⁹ ACI 318-95, 8.4.3; CSA A23.3-94 10.5.2

⁶⁰ ACI 318-95, 7.12; CSA A23.3-94, 7.8

For beams and positive moment regions of waffle slabs, minimum reinforcement ratio, ρ , provided will not be less than:⁶¹

$$A_{s,min} = \frac{3\sqrt{f'_c}}{f_y} b_w d \quad \text{Eq. 2-60}$$

but not less than $200b_w d/f_y$

Table 2.7 – Minimum Ratios of Reinforcement to Gross Concrete Area

f_y (ksi)	A_s/A_g
< 60	0.0020
≥ 60	$\frac{0.0018 \times 60}{f_y} \geq 0.0014$

Reinforcement Selection

The ADOSS default minimum spacing of reinforcement is 6 in. (150 mm) in slabs and 1 in. (25 mm) in beams. The 6 in. minimum for slabs was chosen for practical reasons only. The minimum in no case will be less than the Code prescribed minima for both slabs and beams of one bar diameter, d_b , or 1 in.⁶²

For ACI 318 the maximum spacing of reinforcement is kept at two times the slab thickness.⁶³

For CSA A23.3 banded reinforcement is spaced at a maximum of 1.5 h_s while distributed reinforcement is spread at a maximum of 3 h_s .

An iterative process is performed to determine the number of bars and bar size. The initial number of bars is determined by dividing the total reinforcement area required, A_s , by the area of one bar, A_{sb} , of the input minimum bar size. An additional bar is

61 ACI 318-95, 10.5

62 ACI 318-95, 7.6.1

63 ACI 318-95, 13.4.2

provided if the difference between the required and the provided reinforcement areas is greater than 20% of one bar area or 0.2 in.². Next, the spacing is determined. If the minimum spacing limitations are violated, the bar size is increased and the iterative process continues until all bars sizes have been checked. If the maximum spacing limitations are not met, the number of bars required to satisfy these limitations is computed and the iteration process terminates.

For beams, layered reinforcement is provided if sufficient beam width is not available. ADOSS assumes a 1.5 in. cover to a #4 (an SI N-10) stirrup for width calculations. ADOSS also assumes that the longitudinal bar makes contact at the middle of the stirrup bend where the minimum inside diameter of the bend is $4d_b$. Therefore, an additional width is added to the cover for longitudinal bars less than #14 (an SI N-45) in size (Figure 2-25). This additional width due to the bend, w_{bend} , is equal to:

$$w_{bend} = 0.293 (1.0 - d_b/2) \quad \text{Eq. 2-61}$$

where

$$d_b = \text{bar diameter of the longitudinal bar.}$$

The clear distance between layers is assumed 1.0 in. Hooks and bends are not included in bar length figures.

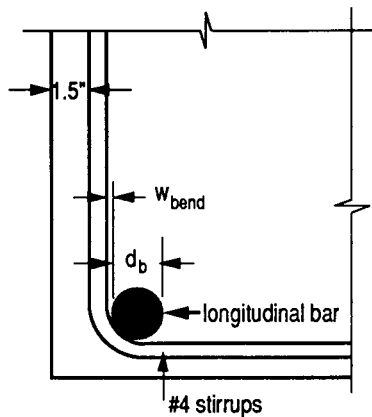


Figure 2-27 Width due to stirrup bend

64 ACI 318-95, 7.6.5

65 ACI 318-95, 7.2.2

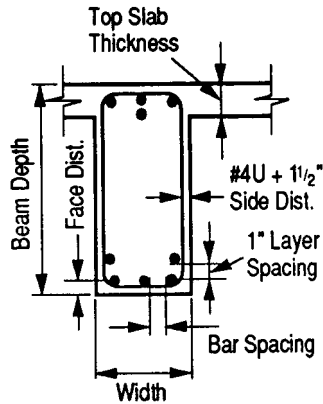


Figure 2-28 Detail of reinforcement in longitudinal beams

Bar-length computations are performed for two-way slabs only. For reinforcement at the top of the slab at the supports, the length for short bars is given by:

$$l_{\text{short}} = \text{greater of} \begin{cases} 0.5 \max (l_{ip}) + \text{greater of} \begin{cases} d \\ 12d_b \\ l_n/16 \end{cases} \\ l_{d, \text{ short}} + l_{cr} \end{cases} \quad \text{Eq. 2-62}$$

and the length for long bars is given by:

$$l_{\text{long}} = \text{greater of} \begin{cases} \max l_{ip}, + \text{greater of} \begin{cases} d \\ 12d_b \\ l_n/16 \end{cases} \\ l_{d, \text{ short}} + l_{cr} \begin{cases} 0.5 \max (l_{ip}) \\ l_{d \text{ short}} + l_{cr} \end{cases} \end{cases} \quad \text{Eq. 2-63}$$

where

$\max(\ell_{ip}) =$		maximum distance to the inflection points, where the distances are the same for both the column and middle strips,
ℓ_d	=	bar development length, ⁶⁶
d	=	effective, depth,
d_b	=	bar diameter,
ℓ_n	-	clear span length.

These bar lengths are then compared and adjusted if necessary to the minimum requirements specified by the code.⁶⁷

If the computed bar lengths overlap, it is recommended that such reinforcement be run continuously. The printed bar lengths do not include hooks or portions of bars bent down into spandrel beams or other bar-bend configurations.

The selection of bar lengths for positive reinforcement for flat plates, flat slabs, and beam-supported slabs, is based strictly on the minimum values of the Code. For waffle slabs, however, the positive reinforcement selection is based on an additional assumption of two bars per joist. For purposes or output, one bar is treated as a long bar and the other as a short bar.

The computation of tension development lengths in terms of d_b is given by the general expression:

$$\frac{\ell_d}{d_b} = \frac{3}{40} \frac{f_y}{\sqrt{f'_c}} \left[\frac{\alpha\beta\gamma\lambda}{c + k_{cr}} \right]$$

Eq. 2-64

The development length depends on some of the following factors: concrete cover, minimum transverse reinforcement, special transverse reinforcement, layer location bar size and bar clear spacing.

66 ACI 318-95, Chapter 12; CSA-A23.3-94, Clause 12.2

67 ACI 318-95; Figure 13.4.8; CSA-A23.3-94, Figure 13.1

Additional Reinforcement at Support

ADOSS computes the fraction of the unbalanced moment, $\gamma_f M_u$, that must be transferred by flexure within an effective slab width equal to the column width plus one and one-half the slab or drop panel depth (1.5h) on either side of the column where:⁶⁸

$$\gamma_f = \frac{1}{1 + \frac{2}{3} \sqrt{b_1/b_2}}$$

Eq. 2-65

The amount of reinforcement required to resist this moment is computed. The amount of reinforcement already provided for negative flexure is then computed from the bar schedule (i.e. the number of bars that fall within the effective slab width multiplied by the area of each bar). If the reinforcement area provided for flexure is greater than or equal to the reinforcement requirements to resist moment transfer by flexure, no additional reinforcement is provided, and the number of additional bars will be set to 0. If the amount of reinforcement provided for flexure is less than that required for moment transfer by flexure, additional reinforcement is required. The additional reinforcement is the difference between that required for unbalanced moment transfer by flexure and that provided for design bending moment in the slab, and it is selected based on the bar size already provided at the support.

It should be noted that the Code requires either concentration of reinforcement over column beam joint by closer spacing, or additional reinforcement, to resist the transfer moment within the effective slab width. ADOSS provides additional reinforcement, and does not concentrate existing reinforcement.

For the ACI Codes, at least two of the column strip bottom bars should be continuous or spliced at the support with Class A splices or anchored with the support.

68 ACI 318-95, 13.5.3.2; CSA-A23.3-94, 13.4.5.3

Additional Information in Span

The total static design moment for each span based on average widths and loads on the span is computed as per the Code. Partial loads are not taken into account.

Deflection Calculation

Calculation of deflections of reinforced concrete two-way slabs is complicated by a large number of significant parameters such as: the aspect ratio of the panels, the vertical and torsional deflection of supporting beams, the stiffening effect of drop panels and column capitals, cracking, and the time-dependent nature of the material response. Based on studies in References 2-22–2-24, an approximate method consistent with the equivalent frame method was developed (2-25) to estimate the column and middle strip deflections.

Under vertical loads, Reference 2-23 indicates that the midspan deflection of an equivalent frame can be considered as the sum of three parts: that of the panel assumed to be fixed at both ends of its span, and those due to the known rotation at the two support lines. The midspan “reference” deflection of a uniformly loaded equivalent frame with fixed ends is:

$$a_{f,ref} = \frac{W\ell^4}{384E_c I_{frame}} \quad \text{Eq. 2-66}$$

where

W = is the uniform load in units of weight per length squared,

E_c = modulus of elasticity of the slab concrete,

I_{frame} = moment of inertia of the full-panel width.

Calculation of the midspan deflection of the column strip or the middle strip under fixed-end conditions is based on the M/EI ratio of the strip to that of the full-width panel:

$$a_{f,strip} = a_{f,ref} \frac{M_{strip}}{M_{frame}} \frac{E_c I_{frame}}{E_c I_{strip}} \quad \text{Eq. 2-67}$$

The ratio ($M_{\text{strip}}/M_{\text{frame}}$) can be considered as a lateral distribution factor, LDF. The lateral distribution factor, LDF, at an exterior negative moment region is:

$$\text{LDF}_{\text{neg,ext}} = 100 - 10\beta_t + 12\beta_t(\alpha_1 \frac{\ell_2}{\ell_1})(1 - \frac{\ell_2}{\ell_1}) \quad \text{Eq. 2-68}$$

The LDF at an interior negative moment region is:

$$\text{LDF}_{\text{neg,int}} = 75 - 30(\alpha_1 \frac{\ell_2}{\ell_1})(1 - \frac{\ell_2}{\ell_1}) \quad \text{Eq. 2-69}$$

The LDF at a positive moment region is:

$$\text{LDF}_{\text{pos}} = 60 + 30(\alpha_1 \frac{\ell_2}{\ell_1})(1.5 - \frac{\ell_2}{\ell_1}) \quad \text{Eq. 2-70}$$

where

- α_1 = the ratio of flexural stiffness of a beam section to the flexural stiffness of a width of slab bounded laterally by centerlines of adjacent panels on either side of the beam.
- β_t = ratio of torsional stiffness of an edge beam section to the flexural stiffness of a width of slab equal to the span length of the beam, center-to-center of the supports. (See Eq. 2-34)

When $\alpha_1 \ell_2 \ell_1$ is greater than 1.0, $\alpha_1 \ell_2 \ell_1$ will be set equal to 1.0.

The column and middle strip LDF's can be computed by:

$$\text{LDF}_c = \frac{\text{LDF}_{\text{pos}} + \frac{\text{LDF}_{\text{neg,l}} + \text{LDF}_{\text{neg,r}}}{2}}{2} \quad \text{Eq. 2-71}$$

$$\text{LDF}_m = 100 - \text{LDF}_c \quad \text{Eq. 2-72}$$

where

$LDF_{neg,l}$, = LDF for the negative moment region at the left and right
 $LDF_{neg,r}$ end of the span, respectively.

If the ends of the columns at the floors above and below are assumed fixed (as is usual in an equivalent frame analysis), then the rotation of the column is equal to the net applied unbalanced moment divided by the stiffness of the equivalent column:

$$\theta = \frac{M_{unbal}}{K_{cc}} \quad \text{Eq. 2-73}$$

The midspan deflection of the panel having an end rotation of Θ radians, with the far end fixed, can be computed as:

$$a\theta = \frac{\theta \ell}{8} \quad \text{Eq. 2-74}$$

The total midspan deflection for the column or middle strip is the sum of the three parts:

$$a_{strip} = a_{f,strip} + a\theta_l + a\theta_r \quad \text{Eq. 2-75}$$

where

$a\theta_l, a\theta_r$ = midspan deflection due to rotation of left and right supports, respectively.

The deflections should be used in conjunction with the deflections obtained from an analysis in the perpendicular direction. For square panels ($\ell_1 = \ell_2$), the midpanel deflection is obtained by Eq. 2-736a as shown in Figure 2-29:

$$a = a_{cy} + a_{mx} = a_{cx} + a_{my} \quad \text{Eq. 2-76a}$$

For rectangular panels, ($\ell_1 \neq \ell_2$), the midpanel deflection is obtained by Eq. 2-76b:

$$a = [(a_{cy} + a_{mx}) + (a_{my} + a_{cx})]^2 \quad \text{Eq. 2-76b}$$

For cantilever spans, Eq. 2-66 must be modified to:

$$a_{f,\text{ref}} = \sum_{i=1}^{20} \frac{P_i x_i^2 (3\ell - x_i)}{6E_c I_{\text{frame}}} \quad \text{Eq. 2-77}$$

where

- P_i = load within segment i of the cantilever span,
- x_i = distance of load P_i from the support,
- ℓ = cantilever span.

$$a\theta = \theta\ell \quad \text{Eq. 2-78}$$

(c) Combined bending

Note: for a rectangular panel, $a = [(a_{cy} + a_{mx}) + (a_{my} + a_{cx})] / 2$

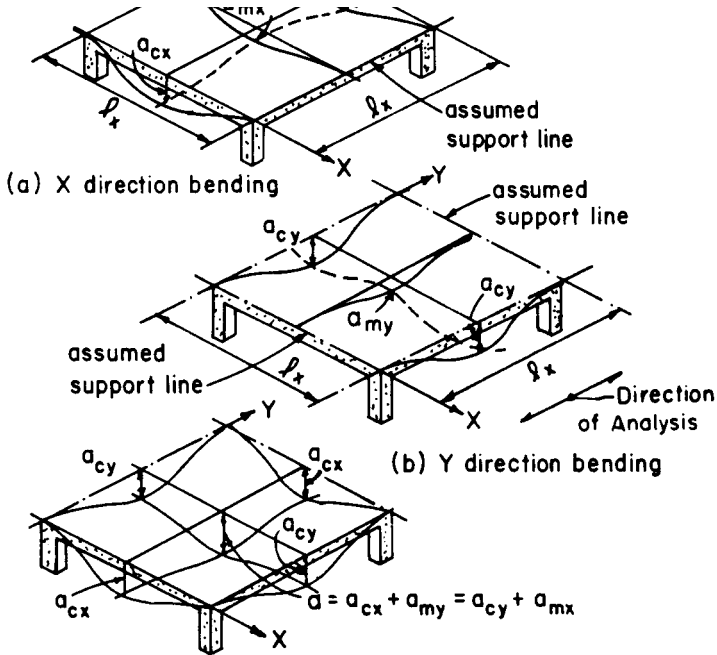


Figure 2-29 Deflection computation for a square panel

Note that the value of I_{frame} in Eq. 2-77 is taken as the effective moment of inertia at the support. Also, this method is applicable to short cantilevers and small loads. Deflections due to large loads and long cantilevers must be computed manually. Local deflections due to concentrated loads are not considered by the program and must be computed manually.

Deflections of Continuous Beams and One Way Slabs

Continuous beam and one way slab deflections are computed based on simply supported deflections less the deflection due to the end support rotations. The effects of support settlement or shear deformations are not considered. When segmental moments and shears are requested, the segmental deflections printed are at 20 equally spaced sections along the span. Concentrated loads occurring at other locations will give rise to larger deflections. These must be checked manually.

Cracking

The effect of cracking in slabs, as in beams, is a reduction in the flexural stiffness. The effect of cracking is generally an increase in deflections, although the tensile concrete between cracks continues to have a stiffening influence. A detailed review of several methods of deflection analyses can be found in Reference 2-24. To apply the Code-specified effective moment of inertia to slab systems, a value of $I_{a,frame}$ is used instead of $I_{a,frame}$ in Eqs. 2-66, 2-67, and 2-77. The value of $I_{a,frame}$ is given by:

$$I_{a,frame} = \beta I_e^+ + (1-\beta) (I_{e,l}^- + I_{e,r}^-)/2 \quad \text{Eq. 2-79a}$$

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.
- β = an empirical constant based on experimental data, whose value is less than 1.0.

For continuous beams, the appropriate value of β compatible with experimental data was found to be 0.7. For flat plates, flat slabs, and waffle slabs, the appropriate value of β appears to be 0.5. ADOSS uses a value of 0.5 for flat plates, flat slabs, and waffle slab systems and 0.7 for beam-supported slabs, continuous beams, and one way slabs.

ADOSS Graphics

The value of $I_{a,frame}$ for cantilever spans is given by:

$$I_{a,frame} = \beta I_{e^-,sup} + (1-\beta) I_{e^-,end} \quad \text{Eq. 2-79b}$$

where

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

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

For continuous beams, the appropriate value of β compatible with experimental data was found to be 0.15. For flat plates, flat slabs, and waffle slabs, the appropriate value of β appears to be 1.0. ADOSS uses a value of 1.0 for flat plates, flat slabs, waffle slabs, and beam-supported slabs, and 0.15 for continuous beams, and one-way slabs.

The value of I_e corresponds to the effective moment of inertia as developed by Bramson (2-26) and incorporated into the Code as follows:⁶⁹

$$I_c = \left\{ \frac{M_{cr}}{M_{max}} \right\}^3 I_g + \left[1 - \left\{ \frac{M_{cr}}{M_{max}} \right\}^3 \right] I_{cr} \quad \text{Eq. 2-80}$$

where

I_g = moment of inertia of the gross uncracked concrete section for the full width of the equivalent frame.

I_{cr} = moment of inertia of the cracked transformed concrete section for the full width of the equivalent frame.

M_{cr} = cracking moment for the full width of the equivalent frame.

M_{max} = maximum bending moment for the full width of the equivalent frame at the load stage at which deflection is computed.

It should be noted that the use of the full slab width in determining I_e in Eq. 2-77 is intended to average the effects of cracking of the column and middle strips.

Deflection Computations

Instantaneous Deflections. Two values of midspan deflections are computed by the program for each span. The first corresponds to dead load on all spans, while the second corresponds to dead load plus live load on all spans. Note that the live load deflection is always computed as total load deflection minus the dead load deflection. This is consistent with the I_{eff} method outlined in the Code

Long Time Deflections. The total midpanel deflection (including long-time effects) consists of the following:

$$a_{\text{total}} = a + a_{\text{creep}} + a_{\text{sh}} \quad \text{Eq. 2-81}$$

where

- a = elastic deflection,
- a_{creep} = additional deflection due to creep,
- a_{sh} = additional deflection due to shrinkage, and
- a_{total} = total deflection.

Data reported in the literature, although limited in quantity, indicate that the additional deflection due to creep and shrinkage, $a_{\text{creep}} + a_{\text{sh}}$, may be as much as two to eight times the elastic deflection, a (References 2-27–2-30). Such a factor times the dead load deflection may be used for an estimate of required slab camber.

For a crude approximation to a complex problem, additional long-time deflections may be computed as three times the elastic deflections. For a more detailed method of analysis of long-time deflection the user is referred to a study by Rangan. (Reference 2-30.)

Material Quantities

The program computes concrete, reinforcing steel, and formwork quantities. The quantity of concrete is based on an average of the slab, drop, and beam sizes. The total quantity of reinforcing steel computed by the program corresponds to the actual bar sizes and lengths required by design. No allowance is made for bar hooks, anchorage embedment, and so forth. It should be noted that the quantity of reinforcement printed by the program pertains to bending in *one direction only*. In practice, the *total* amount of reinforcement for the structure should also include the quantities obtained for the appropriate perpendicular equivalent frames.

Chapter 3

Menu Descriptions

ADOSS Graphics Menu Descriptions

ADOSS Graphics uses pull down menus to access commands that allow input of the floor geometry, manipulate ways to view the system, and ultimately design the system and review the results. This section gives a summary of each command in all the pull down menus. Use this chapter to learn what functions ADOSS commands perform.

The File Menu

The File menu is used for saving or retrieving data, printing, and exiting. The File menu contains the following commands: New, Open, Save, Save As, Export, Revert, Print Screen, Print Results, and Exit.

New

The New command clears any data input, returns to the default values, and begins the Auto Input process (see the “Auto Input” command from the Input menu later in this chapter). Thus, you are able to create a new data file. However, before you can begin a new data file, ADOSS will ask whether you want to save the current data. Answering yes will save the old data and begin a new data file. Answering no will discard any changes to the data and begin a new data file. Answering cancel will return you to ADOSS so that you can continue to work with the current data.

See “Creating a New Data File” of Chapter 4 for more information on using the New command.

Open

The Open command allows you to load an existing ADOSS data file. The dialog box that appears shows you a listing of all the files with the extension contained in the default data directory or in the current directory (if a default data directory was

not specified). This box also enables you to change the current drive and directory. If you are currently working on a data file and select the Open command, ADOSS will ask whether you want to save the current data. Answering Yes will save the old data and display the Open dialog box. Answering No will discard any changes to the data and display the Open dialog box. Answering *cancel* will return you to ADOSS so that you can continue to work with the current data.

See "Opening an Existing Data File" of Chapter 4 for more information on using the Open command.

Save

The Save command saves the changes you've made to the current data under that same filename. The new data overwrites the old data, and you *cannot* retrieve the old data. It is a good practice to periodically Save while inputting data. If a data file is untitled, the Save As dialog box will appear.

See "Saving Your Data" of Chapter 4 for more information on using the Save command.

Save As

The Save As command 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 will ask if you would like to overwrite the file.

See "Saving Your Data" of Chapter 4 for more information on using the Save command.

Export

The Export command saves the current data file into a file format other than the default format. By using the "ADOSS V4.X-V5.2X" conversion utility from the provided list, the Export command ensures compatibility between ADOSS V6.00 and previous versions by converting V6.00 data files to a format that is readable by previous versions.

See "Saving Your Data" of Chapter 4 for more information on using the Save command.

Revert

The Revert command is used to discard current changes to the data and return to the version of data saved in the original data file. This option will only be enabled if the file had been saved prior to editing.

Print Results

The Print Results command allows you to send the analysis and design results to the printer. By selecting options in the Output Options command prior to performing the design, you can print selected sections of the data instead of the entire analysis.

See “Viewing and Printing the Results” and ”Condensing the Output” of Chapter 4 for more information on using the Print Results command.

Print Screen

The Print Screen command prints the floor system geometry in the plan, elevated, and isometric views, prints the shear and moment diagrams, and prints the deflected shapes. To obtain a print you must first perform the design of the view, then view the shear and moment diagrams or the deflected shapes.

See “Viewing and Printing Shear and Moment Diagrams” and “Viewing and Printing Deflected Shapes” of Chapter 4 on using the Print Screen command.

Exit

The Exit command ends the ADOSS session and returns you to Windows. If you’ve made any changes to your data and have not saved them, ADOSS will first ask whether you want to save or abandon any changes you’ve made before you exit.

The Input Menu

The Input menu allows you to enter and modify data for the floor system.

Auto Input

The Auto Input command is designed to make the inputting process easier. By selecting Auto Input, a logical sequence of dialog boxes will automatically be displayed allowing you to enter data for your floor system. Depending on the floor system selected, the program will display only those dialog boxes required to design your floor system.

See "Creating a New Data File" of Chapter 4 for more information on using the Auto Input command.

Labels

The Labels command allows you to enter the project name, span id, engineer, date, and time. Date and time are automatically filled when a new file is created. This information will be printed with the output and is intended to help you organize your data.

See "Including Labels with the Output" of Chapter 4 for more information on using the Labels command.

General Information

The General Information command will allow you to enter the number of spans, units, code, structure type, frame location, lateral loads, loading pattern, and the design option. You must always choose the General Information command before doing any further inputting since it affects the availability of other commands in this menu.

See "Using the General Information Command" of Chapter 4 for more information on using the General Information command.

Material Properties

The Material Properties command enables you to input material property requirements for concrete and reinforcement. Concrete density, compressive strength, tensile splitting strength, modulus of rupture, as well as the longitudinal and shear reinforcement yield levels are required.

See “Defining the Material Properties” of Chapter 4 for more information on using the Material Properties command.

Reinforcement Data

The Reinforcement menu contains commands that allow you to specify the distance to reinforcement, reinforcement bar sizes, and minimum bar spacing for both slabs and beams. Depending on the type of floor system you have selected, either Slabs or Beams or both of these commands will be available.

See “Defining Reinforcement Parameters” of Chapter 4 for more information on using the Reinforcement Data menu.

Geometry

The Geometry menu contains commands that allow you to input geometric dimensions for slabs, ribs, longitudinal beams, columns/capitals, drops, and transverse beams. A cascading menu contains the member types to modify. Depending on the floor system, some menu items will be disabled.

See “Defining the Structure Geometry” of Chapter 4 for more information on using the Geometry menu.

Loads

The Loads menu contains commands that allow you to enter superimposed uniform loads, trapezoidal loads, concentrated axial loads and moments, lateral loads, and allows you to modify the load factors. Depending on the lateral load type, some menu items will be disabled.

See “Specifying Loads on the Structure” and “Using the General Information Command” of Chapter 4 for more information on using the Loads menu.

Column Fixity

The Column Fixity command allows you to modify the percentage of the actual column joint stiffness to be used in the analysis to determine the joint moments and shears.

See “Changing the Column Fixity” of Chapter 4 for more information on using the Column Fixity command.

The View Menu

The View menu commands enable you to modify the floor system's appearance on the screen to suit your viewing needs. The following is a list of what is available in this menu.

Redraw

The Redraw command will redraw the floor system in full size. If you have altered your screen view using the Zoom command, select Redraw to restore the figure's original proportions.

Scale On/Off

The Scale On/Off commands allow you to view your floor system with or without a scale in ft. (m) along the left and top edge of the view window. The left edge of the floor system and the column centerline is considered the origin.

See "Viewing the Floor System with a Scale" of Chapter 4 for more information on using the Scale On/Off commands.

Zoom

The Zoom command enables you to zoom in on any portion of your floor system for closer analysis. By selecting zoom and either using the mouse or keyboard arrow keys to specify a zooming region, the program will enlarge the portion you select.

See "Zooming in on the System" of Chapter 4 for more information on using the Zoom command.

Change Isometric Angle

The Change Isometric Angle command allows you to modify the angle at which the floor system is displayed in the Isometric View. The default angles are set at -45 about the X and 45 about the Z axis.

See "Changing the Isometric View Angle" of Chapter 4 for more information on using the Change Isometric Angle.

View Members

The View Members command allows you to view selected members of the floor system in the views windows. Clicking the left mouse button on the box next to the

items in the dialog box, or tabbing to the member type and pressing the space bar will toggle the selection. ADOSS will draw any members that contain a check in the box.

See “Viewing Specific Member Types” of Chapter 4 for more information on using the View Members command.

View Loads

The View Loads command allows you to view selected loads of the floor system in the Partial Loads window. Clicking the left mouse button on the box next to the items in the dialog box, or tabbing to the type and pressing the space bar will toggle the selection. ADOSS will draw any loads that contain a check in the box.

See “Viewing Specific Load Types” of Chapter 4 for more information on using the View Loads command.

View Results

The View Results command will execute PCA View, so you may view the design results for your floor system on the screen after a design has been performed. The PCA View window, displaying the results, will overlap the main window of the ADOSS program.

See “Viewing and Printing the Results” of Chapter 4 for more information on using the View Results command.

View Shears/Moment

The View Shears/Moments command creates a window displaying a shear and moment diagrams of a particular span and load pattern after a design of the floor system has been performed.

See “Viewing and Printing Shear and Moment Diagrams” of Chapter 4 for more information on using the View Shear/Moment command.

View Deflections

The View Deflections command creates a window displaying the deflected shape of a particular span of a continuous beam or one-way slab system after a design of the floor system has been performed.

See “Viewing and Printing the Deflected Shapes” of Chapter 4 for more information on using the View Deflections command.

The Design Menu

The Design menu contains commands that enable you to perform the analysis and design of the floor system or column section.

Condense/Expand Output

The Condense/Expand Output command allows you to suppress portions of the output. ADOSS will print only those items that have a check in the box. To take effect, the design must be performed after the Condense/Expand Output command is executed.

See “Condensing the Output” of Chapter 4 for more information on using the Condense/Expand Output command.

Design System

The Design System command executes the solver portion of ADOSS. If some data is still required when this command is executed, ADOSS will respond with “More data required” and display the dialog box that must be completed.

See “Designing the Floor System” of Chapter 4 for more information on using the Design System command.

Design Column

The Design Column command executes PCACOL if it is found on the computer system. This option will be grayed if PCACOL is not found after the design of the floor system is performed. In order for ADOSS to find PCACOL, the directory where PCACOL resides must be in the DOS PATH statement. (See your DOS manual for help on the PATH statement.)

See “Designing the Column” of Chapter 4 for more information on using the Design Column command.

The Options Menu

The Options menu allows you to change the startup options of the ADOSS program to suit your needs. These changes are saved in the PCAPROGS.INI file and they become the default settings when ADOSS is executed.

Background Color

The Background Color command allows you to change the background of the three views windows to black or white. The background color chosen will become the default background color when ADOSS is executed.

See “Changing the Background Color” of Chapter 4 for more information on using the Background Color command.

Member Colors

The Member Colors command allows you to change the colors of the individual member types of the floor system views. This option will only be available on systems that support color. These colors will become the default colors when ADOSS is executed.

See “Changing the Member Colors” and “Changing Background Colors” of Chapter 4 for more information on using the Member Colors command.

Load Colors

The Load Colors command allows you to change the colors of the individual load types of the loads window. This option will only be available on systems that support color. These colors will become the default colors when ADOSS is executed.

See “Changing the Load Colors” of Chapter 4 for more information on using the Load Colors command.

Default Units/Code

The Default Units/Code command allows you to change the default units and code used at startup that were set during the ADOSS installation. The choices set here will become the defaults when ADOSS is executed.

See “Changing the Default Units and Code” of Chapter 4 for more information on

using the Default Units/Code command.

Default Data Directory

The Default Data Directory command allows you to modify the directory where ADOSS looks for data when it is executed, changing the directory specified during the ADOSS installation.

See “Changing the Default Data Directory” of Chapter 4 for more information on using the Default Data Directory command.

Page Layout

The Page Layout command allows you to set the left, right, top, and bottom margins of the print screens.

See “Changing the Margins of a Screen Print”, “Viewing and Printing Shear and Moment Diagrams”, and “Viewing and Printing the Deflected Shapes” of Chapter 4 for more information on using the Page Layout command.

The Window Menu

Once all the data for the floor system has been input, there are three ways in which to view your model: Plan View, Elevated View, and Isometric View. This menu enables you to arrange and select different view settings for these windows along with any shear and moment windows, or deflected shape windows.

See “Manipulating Windows” of Chapter 4 for more information on using the commands in the Window menu.

Cascade

The Cascade command displays all the 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

The Tile All Windows command arranges all open windows so that no window overlaps another.

Tile Views Windows

The Tile Views Windows command arranges the open view windows on your screen without overlapping them. If all the view windows are open, the windows will be arranged so that Isometric View is a half page display and Plan View and Elevated View each share the other half. The Shear/Moments and Deflection windows (if opened) will not be displayed.

Arrange Icons

The Arrange Icons command will align all the icons present at the bottom of the screen.

Remaining Commands

The remaining menu items is a list of the windows that are available for viewing. Selecting any window from this menu will restore the window to its previous size and position from an icon.

The Help Menu

The Help menu includes commands that enable you to obtain online help for the ADOSS program. The help screens in the program are closely related with Chapters 3 and 4 of this manual.

See “Using On-line Help” of Chapter 4 for more information on using the commands in the Help menu.

Index

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

Commands

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

Procedures

The Procedures command lists several procedures on ADOSS operation, similar to Chapter 4. Click on any underlined topic and a help screen appears with information about that procedure.

Using Help

The Using Help command gives help on using the Microsoft Windows help system.

About ADOSS

The About ADOSS command displays a dialog box that gives you copyright information, the version number of the ADOSS program, your firm name, city, and state.

The Control Menu

All the view windows have one pull-down menu located in the upper left hand corner of the open window. This is the Control menu. To access the Control menu, press **Enter** + **F6** to cycle through the windows and press **Alt** + **-**, hyphen, to open the Control menu of the desired window. The following is a list and a brief description of the commands in this menu.

Note: See “Basic Skills” of Chapter 2 in the Microsoft Windows User Guide for more information on using the commands in the Control menu.

Restore

The Restore command will restore a window or an icon to its previous size and position. This menu item is available when the window is iconized or maximized.

Move

The Move command moves the window to a new location. Select **Move** and use the arrow keys to move the window in the desired direction and select **Enter** to accept the new location.

Size

The Size command resizes a window. Select **Size** and use the arrow keys to move the border of the window in the desired direction and select **Enter** to accept the

new size.

Minimize

The Minimize command reduces a window to an icon and positions it at the bottom of the screen.

Maximize

The Maximize command enlarges a window to fit your entire screen.

Close

The Close command is used to close a window and return it to an icon and the bottom of the screen.

Next

The Next command switches among open windows and icons.

Chapter 4

Operating the Program

This chapter is designed to help you work with menus and inputting data into dialog boxes. It is divided up into different sections that follow operating procedures needed to complete the input process and design a floor system. Use this chapter to guide you through inputting, for quick reference on particular sections, or to help answer any questions you may have about the program.

Main Screen Components

The following is a list of the main screen components you must know in order to understand the operating procedures of ADOSS in this chapter.

You must have a copy of Microsoft Windows 3.0 or higher already installed on your system to be able to use ADOSS. An advantage to working with Windows is that you always have your options listed in front of you. Figure 4-1 illustrates how your screen will look when you have loaded ADOSS.

Title Bar

Refer back to Figure 4-1 to locate the title bar at the top of the screen. The title bar displays the ADOSS name, and following the hyphen, displays the name of the current data file you are using. If the data you are currently working on has not been saved into a file, the word *Untitled* is displayed in the title bar.

Control Menu

Each window has a Control menu located to the left of the title bar, in the upper left corner of the window. This menu contains commands that allows you to manipulate that particular window. To access the Control menu using the mouse, click the left mouse button on the box in the upper left corner. To access the Control menu using the keyboard, press **Ctrl**+ **F6** to cycle through the windows until you reach the desired window and press **Alt** + **-**, hyphen.

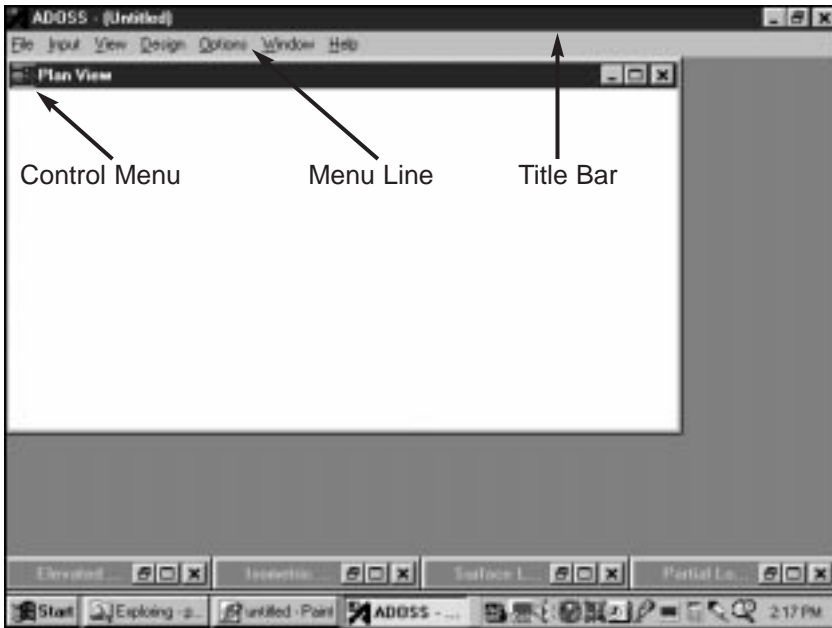


Figure 4-1 The ADOSS main window

Menu Line

Located directly below the title bar is the menu line. ADOSS commands are listed in the pop-up menus located in the menu line. These menu commands allow you to perform functions that create, view, and ultimately design the floor system.

In the ADOSS program there are seven main pop-up menus: File, Intput, View, Design Options, Windows, and Help. To access a menu item using the mouse, place the arrow cursor on the menu item you want and click the left mouse button. Each menu item can also be selected with the keyboard keys by simultaneously pressing the **Alt** key and the underlined letter of the menu you want to open. For example, to open the File menu, press **Alt** + **F**.

To close a menu without selecting a command, move the cursor to any blank area on the screen and click the left mouse button. Press **Esc** to close a menu using the keyboard keys.

To select a command from a menu with the mouse, place the arrow on the item you

ADOSS Graphics

want, and click the left mouse button. In some cases, you will be told to double click on a selection, that is, press the mouse button twice, quickly. Anytime you have to wait, for example, when loading the program or designing the system, the mouse cursor becomes an hourglass cursor. It will return to its original state when the task is completed.

To select a command from a menu using the keyboard, use the down arrow key to highlight your choice and press **Enter** or press the keyboard key of the command's underlined letter. The space bar is also equivalent to pressing the left mouse button. Special instructions for inputting with the keyboard keys are given wherever necessary.

Dialog Boxes

Dialog boxes are used to request information or provide you with information. By selecting a command from a menu, a dialog box may be displayed requesting data to complete the command. A dialog box will follow any command that contains three trailing periods, "...". After entering the required data, press the **Ok** button so that the data associated with that dialog box will be acted upon.

Edit Boxes

Edit boxes are rectangular boxes inside dialog boxes that require text to be completed. In some cases, the edit box will contain only a flashing cursor located at the left side of the box prompting you to enter text. In other cases there may already be text in the edit boxes. This text can be overwritten when you begin typing. To erase the text, use the backspace or the delete keys.

Menu and Dialog Controls

The following are descriptions of symbols and options you will encounter during the inputting process.

Enabled/Disabled Options

Some commands in the pop-up menus or input controls in the dialog boxes are black while others are grayed. Commands that are black are available for use at that time. Commands not available are gray and may become available depending on the data you enter. For example, in the Input menu, only Auto Input, Labels, and General Information are black at the start of a new file. When you have input information in the General Information dialog box you will notice that the remainder of the commands in the Input menu become black and are now ready for your input.

Radio Buttons

A radio button is used in a group of options from which you may select only one. A group of radio buttons is a type of toggle (on/off) switch that when one radio button is chosen, its state is toggled on and all other radio buttons are toggled off.

For example, in the General Information dialog box you will see radio buttons in the options you need to make a selection. When using a mouse, place the cursor over the option desired and click the left button once to toggle the switch on. When using the keyboard keys, tab to the group of options and choose one by using the up and down arrow keys to toggle the switch.

Check Boxes

A check box is a toggle used in a dialog box that enables or disables an option. To enable a check box, toggle the switch so that an X is placed in the box. Removing the X from the check box disables the command.

Click once in the check box with the mouse to enable or disable the command or use the keyboard keys to tab to the check box and press the space bar to activate the option.

Push Buttons

Push Buttons are command buttons such as **Ok** and **Cancel** that are located inside dialog boxes. These are labeled to do exactly what they say. After you have completed your input within a dialog box, click once with the left mouse button on **Ok** or press **Alt** + **O** with the keyboard keys to save the changes. If, while working in a dialog box, you want to disregard the data you have input, select **Cancel** to exit and discard any new data. When using the keyboard keys select **Alt** + **C**.

Working With Data Files

All of the commands required to save or open ADOSS data files are located in the File menu. This section will help you with the procedures of using the data files.

Creating a New Data File

When you first load ADOSS you will have a new file ready for input. The data will not have a filename associated with it, therefore, "Untitled" will appear in the title bar.

To start a new data file:

1. If you are already in the program and in an existing file, select **New** to clear your screen and return you to the default settings. If existing data has been changed prior to executing the **New** command, ADOSS will ask if you would like to save the data.
2. Selecting **New** invokes the **Auto Input** command. This command guides you through the inputting process by automatically displaying all the dialog boxes necessary to design your floor system. You may cancel the Auto Input mode by selecting **Cancel** from any dialog box.
3. After you enter data through the Input menu, use the Save As command to give the file a name.

Saving Your Data

ADOSS has three commands for saving your data. You may save your data in the ADOSS format with the same filename or a new filename, or save the data in the old ADOSS format.

In order to save the shear and moment diagrams as well as the deflected shapes of the one-way slabs and continuous beam systems, you must save your data using the Save command after you execute the Design System command.

To save your data with the same filename:

1. Select the **Save** command from the File menu. *If the data has not been modified since the last Save or Save As command was executed, this option will not be available.* The file will be updated and the Save command will be grayed.

To give your data a new filename:

1. If you have not saved your data yet, select the **Save** or **Save As** command from the File menu, these commands will have the same effect. If you would like to save a data file that currently has a filename with a new filename, select the **Save As** command. The dialog box of Figure 4-2 will appear. When no filename has been given to the current data, the default filename is UNTITLED.ADS and it is highlighted in the edit box.
2. Type a new name to overwrite the current name.
3. Press **Save** to exit the dialog box and save the data into the filename specified.

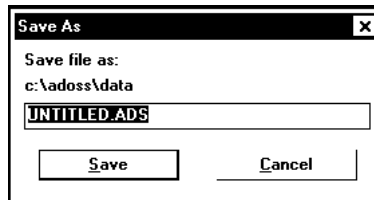


Figure 4-2 The Save As dialog box

To save your data in the old ADOSS format:

1. Select the **Export** command from the File menu. The dialog box of Figure 4-3 will appear.
2. Type a new name for the data file.
3. Press the **Export** button. At this point, if a file with the same name exists, a message box will appear requesting if you would like to overwrite that file.
4. Select the “ADOSS V4.X-V5.X Data” conversion utility from the list.
5. Press **Ok** to save the data in the old ADOSS format.

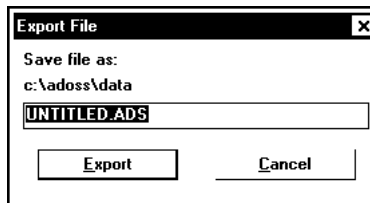


Figure 4-3 The Export dialog box

Opening an Existing Data File

ADOSS allows you to open data files that were saved at an earlier time. ADOSS also allows you to open data files from a previous version of ADOSS.

To open an existing data file:

1. Select **Open** from the File menu to bring in an existing ADOSS data file. The dialog box of Figure 4-4 will be displayed. All the files with an **.ADS** extension contained in the current drive and directory will be displayed in the list box. This dialog box also enables you to change the current drive and directory.
2. Type in the name of the file you want to open. You may also select a file from the provided list with the mouse or using the keyboard by tabbing to the list and using the up and down arrow keys.

3. Press **Open** to exit the dialog box and allow ADOSS to read the data. You may combine steps 2 and 3 of this procedure by double clicking the left mouse button over the desired file from the provided list.

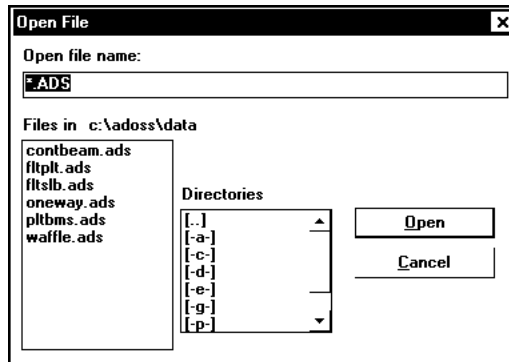


Figure 4-4 The Open dialog box

Modeling a Building Floor System

To model a building floor or a supported structural slab with ADOSS, lay out design strips in one of two orthogonal directions (or lay out strips in the direction you want to orient the flexural reinforcement). Select unique sets of identical strips such as exterior or interior strips. The dimensions of each unique strip are entered into ADOSS directly. Other items such as floor system, (flat plate, flat slab, etc.), concrete properties, and loadings must be pre-selected based on design considerations. Initial slab or beam dimensions must be based on rough hand calculations or Code determined criteria. After an initial run, optimizing the slab or beam dimensions can be accomplished by simply adjusting the dimensions and re-running the data.

Including Labels with the Output

To keep track of your output, ADOSS allows you to specify the name of the project, span id, engineer, date, and time (date and time are automatically input when a new data file is begun). These labels will be printed with your output and are intended

ADOSS Graphics

to help you organize your data.

To enter labels:

1. Select **Labels** from the Input menu. The dialog box of Figure 4-5 will appear.
2. Enter the Project ID, the identifying project title.

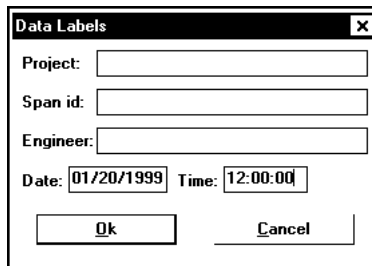


Figure 4-5 The Data Labels dialog box

3. Tab to the Span ID and enter the identifying span label.
4. Tab to the Engineer edit box and enter your name or the name of the engineer who prepared the data.
5. Specify a new date and time or keep the default values.
6. Select **OK** to exit the dialog box and allow ADOSS to save these labels.

Creating the Floor System

The floor system is created by entering data through the Input menu. You may select the Auto Input command from the Input menu to automatically take you through the required dialog boxes or you can follow the manual procedure given below.

To create the floor system:

1. Select the **General Information** command from the Input menu to define the basic information. Based on this information, the applicable remaining commands in the Input menu will become available.
2. Enter the material properties by selecting the **Material Properties**

command from the Input menu.

3. Enter the reinforcement information by selecting the **Slabs** and/or **Beams** commands from the Input/Reinforcement menu.
4. Enter the floor system geometry from the commands located in the **Geometry** menu. Some of the member types may not be available, depending on the floor system chosen in the General Information command of step 1.
5. Finally, enter the loads on the system from the commands located in the Input/**Loads** menu.

Using the General Information Command

The General Information command allows you to enter the number of spans, units, code, structure type, frame location, lateral load types, loading pattern, and the design option needed by ADOSS to proceed with the input process. You must choose this command before doing any further inputting since this command affects the availability of the commands in the Input menu.

To enter general information:

1. Select the **General Information** command from the Input menu. The dialog box of Figure 4-6 will appear.

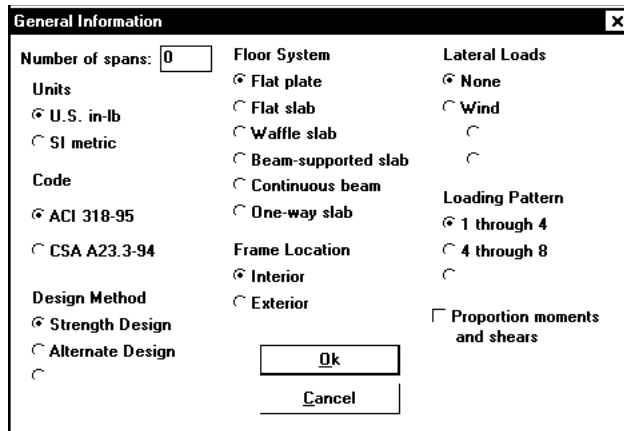


Figure 4-6 The General Information dialog box

2. Enter the **Number of spans** for your particular floor system. The minimum and maximum number of spans are 3 and 14 spans, respectively. Note that two cantilevers must be included in the number of spans. If an invalid number of spans is given, a message will inform you of such when the **Ok** button is pressed.
3. Select your unit of measurement. **US in-lb** and **SI Metric** are the two offered in ADOSS. The SI Metric option also uses metric reinforcement.
4. Select the building standard you want your floor system to be designed to: **ACI 318-95, CSA-A23.3-94**.
5. Select the **Design method**. Strength reduction factors and load factors are set by this option. The ACI code allows **Strength Design** and **Alternate States Design** whereas **Limit States** is the only option allowed with CSA Standard CSA-A23.3-94.
6. Choose the **Floor system** type, you have six floor systems to choose from. This option affects the availability of the geometry commands in the Input/Geometry menu described later in this chapter. For example, drop panel dimensions are required for flat slab and waffle slab systems, whereas beam dimensions are required for beam-supported slabs and continuous beam systems.
7. The **Frame location** sets the default left slab width to either an interior or exterior slab, depending on your selection.
8. Next, select the type of **Lateral loading** present. The default is none. When Wind lateral loads are specified, the lateral loads are entered as either concentrated **Forces** above and at the level of slab to be designed, or **Moments** entered to the left, right, above, and below the joint. This option affects the availability of the lateral loads commands in the Input/Loads menu described later in this chapter.
9. Choose the **Loading pattern** to be used by ADOSS for analysis and design. For vertical loads, the program selects the design requirements for flexure and shear based on the following loading patterns:
 - Pattern 1 Full dead and 75% live on adjacent spans.
 - Pattern 2 Full dead and 75% live on alternate odd spans.
 - Pattern 3 Full dead and 75% live on alternate even spans.
 - Pattern 4 Full dead and live on all spans.

When concentrated lateral loads are specified, or when moments due to lateral loads are specified, the following additional loading patterns are considered:

Pattern 5 Full dead and positive lateral.

Pattern 6 Full dead and negative lateral.

Pattern 7 Full dead, live, and positive lateral.

Pattern 8 Full dead, live, and negative lateral.

10. Lastly, when the box marked **Proportion moments and shears** is checked, the moments and shears at a joint will be proportioned between the slab and beams framing into the joint. This option is not available for one-way slabs or continuous beams and it has no effect if the transverse beams do not exist in the other floor systems. If you are using the keyboard, use the space bar to insert or remove a check from this box. (See “Shear Analysis of Slabs”, “Shear Analysis of Longitudinal Beams”, and “Shear and Torsion Analysis of Transverse Beams” in Chapter 2 on how this option affects the design results.)
11. Press to exit the dialog box and allow ADOSS to use the new data.

Defining the Material Properties

The **Material Properties** command from the Input menu allows you to input material properties of the concrete and the reinforcement. This command must be executed in order to perform a design of the floor system.

Use the tab key to get to each edit box then type in your values, or use your mouse and click directly on the desired box, then type in your values. Refer to Chapter 2 “Material Properties” for a detailed explanation of the default values.

To define material properties:

1. Select the **Material Properties** command from the Input menu. The dialog box of Figure 4-7 will appear.
2. Enter the concrete **Density** for the following members: Slabs, Beams, and Columns. Entering a value for slabs will automatically display a value for each of the other members. The Type of concrete is automatically determined from the values input for density.

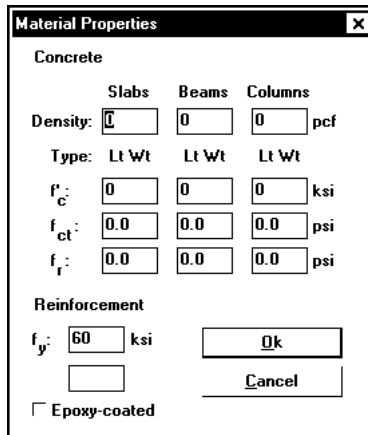


Figure 4-7 The Material Properties dialog box

3. Enter the concrete compressive strength. By entering a value for f'_c , values for f_{ct} and f_r will automatically be computed for the slabs, beams and columns.
4. If you have values for the average split tensile strength of concrete, f_{ct} , enter the values in the edit boxes for the slabs, beams, and columns. These values will be used to compute modulus of rupture and development lengths for concrete weight of densities lower than normal. Default values are computed based on f'_c .
5. If you have values for the modulus of rupture, f_r , enter the values in the edit boxes for the slabs, beams, and columns. Default values are computed based on f_{ct} . These values will be used for deflection analysis. A large value for f_r will produce a deflection analysis based on gross, non-cracked, sections. The CSA A23.3-94 Standard requires that for the calculation of slab deflections an f_r value equal to $0.6\sqrt{f'_c}/2$ be used. ADOSS defaults to this value of f_r for the slab concrete in CSA design runs.
6. Enter the yield strength, f_y , for slab and beam longitudinal reinforcement.
7. Enter the yield strength, f_{yv} , for beam shear reinforcement.
8. Select whether the main reinforcement is epoxy-coated by clicking the left mouse button on the box or tabbing to the box and pressing the space bar. This selection affects development lengths.

9. Press **Ok** to exit the dialog box so that ADOSS will use these material properties.

When Alternate Design is selected, use f_s for f_y and f_{yv} .

Defining Reinforcement Parameters

In order for ADOSS to select the reinforcement, you must define the slab and beam reinforcement, bar sizes, location, and minimum spacing dimensions. Depending on the type of floor system you specified, either Slabs, Beams, or both of these commands will be available to you. (See “Area of Reinforcement” and “Reinforcement Selection” for a discussion of the reinforcement computations.)

For both Slab and Beam dialog boxes, use the tab key to move to each input box or use your mouse to click on an input box.

To define reinforcement:

1. Select either the **Slabs** or **Beams** command from the Input/Reinforcement menu. The dialog boxes of Figure 4-8 will appear.
2. For slabs and beams, enter the distance to the centroid of the reinforcing from the tension face. For the top reinforcement, this distance is from the top of the slab to the centroid of the top bars; and for the bottom reinforcement, this distance is from the bottom of the slab to the centroid of the bottom bars, (see Figure 4-9). The default value is 1.5 in. (40mm) for both input items.
3. Enter the **Minimum bar size** to start the iteration for determining flexural reinforcement.
4. Enter the **Maximum bar size** in beam design for beam-supported slab and continuous beam systems only. This number will be used as a stop in the iteration for determining flexural bars in beams.
5. Select whether the reinforcement is placed in the outer or inner layer. This selection affects development lengths and will be grayed if any code other than ACI 318-95 was selected in the General Information dialog box.
6. Enter **Minimum bar spacing** for slab and beam flexural reinforcement. This number should be based on aggregate size or detailing

	Top	Bottom
Distance to reinforcement centroid from tension face:	<input type="text" value="1.5"/>	<input type="text" value="1.5"/> in
Minimum bar number:	<input type="text" value="4"/>	<input type="text" value="4"/>
Layer location:	<input checked="" type="radio"/> Outer <input type="radio"/> Inner	<input checked="" type="radio"/> Outer <input type="radio"/> Inner
Minimum clear bar spacing:	<input type="text" value="6"/> in	
<input type="button" value="Ok"/> <input type="button" value="Cancel"/>		

	Top	Bottom
Distance to reinforcement centroid from tension face:	<input type="text" value="1.5"/>	<input type="text" value="1.5"/> in
Minimum bar number:	<input type="text" value="4"/>	<input type="text" value="4"/>
Maximum bar number:	<input type="text" value="14"/>	<input type="text" value="14"/>
Layer location:	<input checked="" type="radio"/> Outer <input type="radio"/> Inner	
Minimum clear bar spacing:	<input type="text" value="1"/> in	
<input type="checkbox"/> Code minimum ties provided.		
<input type="button" value="Ok"/> <input type="button" value="Cancel"/>		

Figure 4-8 The Slab and Beam Reinforcement dialog boxes

considerations. Default spacing is 6 in. (150mm) for slabs, and 1 in. (25mm) for beams.

7. Select whether Code minimum stirrups are provided for beam-supported slabs and continuous beam systems. This section requires that minimum stirrups be provided over the entire length of the reinforcement development. This option affects the development length.
8. Press **Ok** to exit the dialog box and allow ADOSS to use the new data.

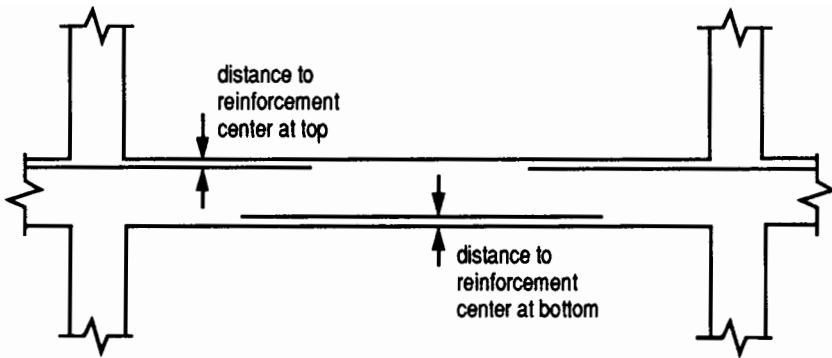


Figure 4-9 Distances to the reinforced centroid

Defining the Structure Geometry

Select **Geometry** from the Input menu to enter slab, rib, longitudinal beam, column, column capital, drop, and transverse beam geometry. Depending on the floor system chosen in the General Information box, commands in the Geometry menu may or may not be available.

The input method for these dialog boxes (except for ribs) differs from others you have already encountered.

Using the replace button:

When beginning a new file, ADOSS fills the geometry information with zeros. In order to modify the geometry, you must enter the data and replace the values.

1. Select the geometry command of the member type to modify, such as the Slabs command, from the Input/Geometry menu.
2. Enter dimensions for the desired span or column.
3. For the program to accept your input, select **Replace**. The updated data will appear in the list box to the right.
4. Press **Ok** to exit the dialog box so that ADOSS will use the new data.

Using the copy from button:

Some dimensions may be the same for a member type. Instead of typing the data

over for each line use the **Copy From** command located in the lower half of the dialog box.

1. Select the geometry command of the member type to copy, such as the Slabs command, from the Input/Geometry menu.
2. Select the members to copy to. When copying geometry, surface loads, or column fixity data, multiple spans or columns can be selected.

Mouse selection:

Single Click	Selects the item and removes the selection from all other items. This selection will also become the anchor for subsequent mouse actions.
Shift + Single Click	Selects all items between the selection anchor, and removes the selection from items not in that range.
Control + Single Click	Toggles the selection state of the selected item but does not remove the selection from the other items. This selection will also become the anchor for subsequent mouse actions.

Keyboard selection:

Shift + F8	Toggles the add mode state. The add mode is indicated by a flashing list box cursor.
Space bar	If the add mode is disabled, the space bar is the same as a single click of the mouse. If the add mode is enabled, the space bar is the same as Ctrl + a single click of the mouse.
Arrow keys	If the add mode is disabled, these keys move the list box cursor as defined by the key. This selection will also become the anchor for subsequent mouse actions. If the add mode is enabled, these keys move the list box cursor as defined by the key but does not select the item, remove the selection from other items, or move the selection anchor.

3. Select the **Copy From** button and enter the member number to copy from.

4. Press **OK** to exit the Copy From dialog box so that ADOSS will update the list box.
5. Press **Ok** to exit the dialog box so that ADOSS will use the new data.

Defining the Slab Geometry

The Slabs geometry command from the Input/Geometry menu is available for all floor systems. Span numbers are automatically sequenced from 1 to the number of spans entered in the General Information box. There are three different dialog boxes for slab dimensions, the one displayed depends on the floor system chosen (see Figure 4-10). This dialog box will not have span length information if the system is a continuous beam. This command is optional for continuous beam systems.

To input slab geometry:

1. Select the **Slabs** command from the Input/Geometry menu. One of the dialog boxes of Figure 4-10 will appear.
2. Enter the span **length** from column centerline to column centerline or edge to column centerline for the two cantilever spans (see Figure 4-11). (*Note: for continuous beam systems, the span lengths are input in the **Longitudinal Beam Geometry** box and not in the **Slab Geometry** box.*) If there is no cantilever, half the column dimension should be entered as the span length for cantilevers. If the program detects a cantilever span length less than one-half the column dimension in the direction of analysis, c_1 , the cantilever length will be increased. If a partial load is affected by the span length l_1 , a message warns the user of this condition.
3. Enter the slab thickness of the span. For continuous beam systems where the beam is a T-beam, this is the flange depth.
4. Enter the span design width in the transverse direction of analysis on the left and right side of the column (see Figure 4-11). These distances are usually one-half the distance to the next transverse column or edge of the slab for exterior spans. The left and right designations are arbitrary. Be consistent. Both interior and exterior spans may be used in a design strip. Exterior sides are designated with the letter "E" input after the slab width value, such as 2.5E. This indicates an exterior width of 2.5 ft (m). Also, an exterior width will automatically be designated by ADOSS by entering

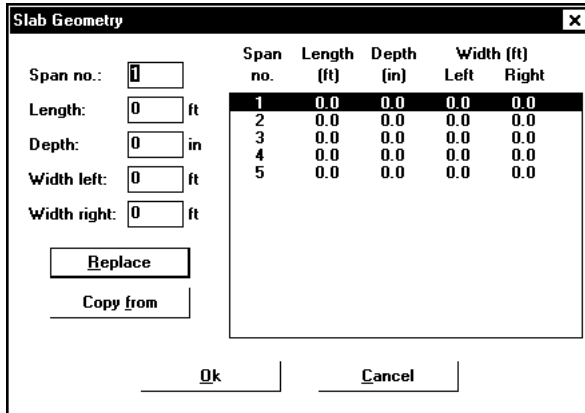


Figure 4-10a The Slab Geometry dialog box for flat plate, flat slab, beam-supported slab, and one-way slab systems

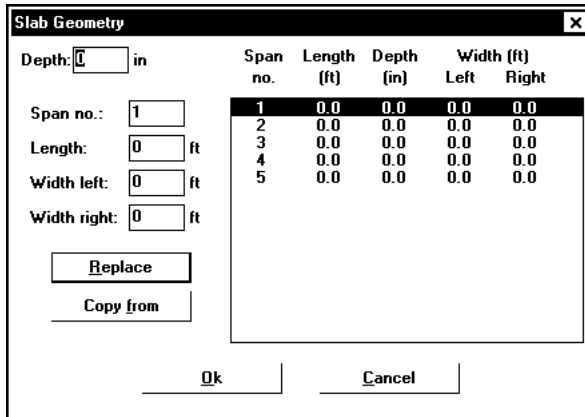


Figure 4-10b The Slab Geometry dialog box for waffle slab systems

a width value less than or equal to the transverse column dimension, c_2 . To change an exterior width to an interior width, enter an “I” after the width value. Exterior sides do not contribute to the attached torsional stiffness, although they do contribute to loading. For T-beams, the width left plus width right may be larger than the allowed flange width. ADOSS will use the total width entered for weight and superimposed loading but

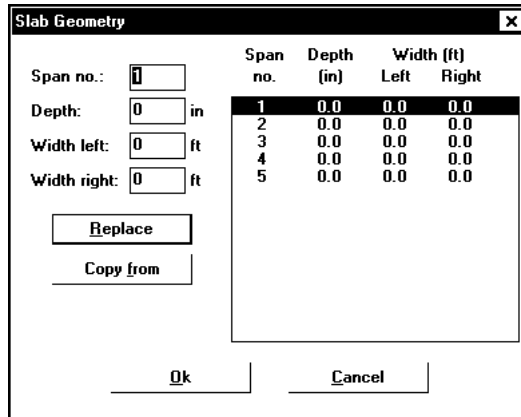


Figure 4-10c The Slab Geometry dialog box for continuous beam systems

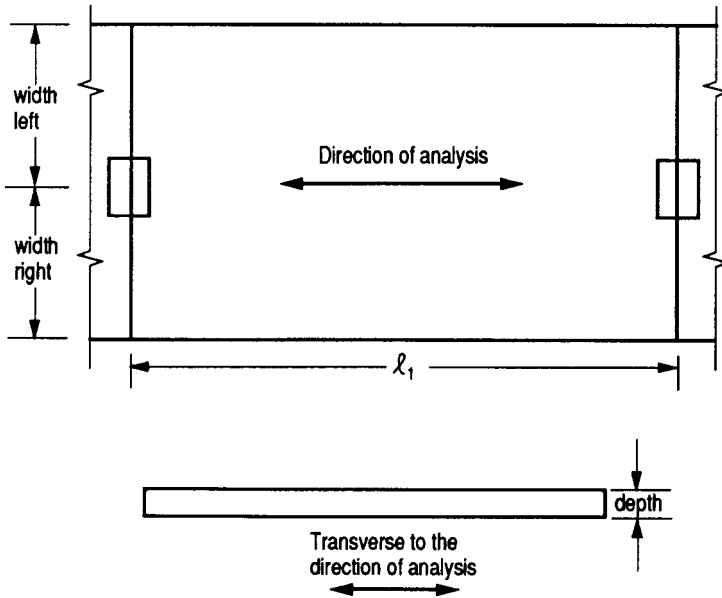


Figure 4-11 The required slab dimensions

ADOSS Graphics

will use code allowed dimensions for flange width and stiffness computations.

5. Press the **Replace** button to update the slab geometry.
6. Repeat steps 2 through 5 until all the spans have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
7. Press **Ok** to exit the dialog box and allow ADOSS to use the updated slab geometry.

Defining the Rib Geometry

For the waffle slab system, you must define the rib geometry. The ribs are assumed to be the same throughout the strip.

To enter rib geometry:

For the rib dialog box, use the tab key to move to each edit box or use your mouse to click on an edit box.

1. Select the Ribs command from the Input/Geometry menu. The dialog box of Figure 4-12 will appear.
2. Enter the spacing between ribs at the bottom for Clear rib spacing (see Figure 4-13).

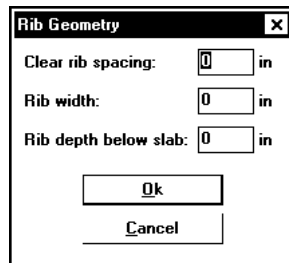


Figure 4-12 The Ribs Geometry dialog box

3. Enter the width at the bottom for Rib width (see Figure 4-13).
4. Enter the depth of the rib below the slab for Rib depth (see Figure 4-13).
5. Press **OK** to exit the dialog box so that ADOSS can use the rib geometry.

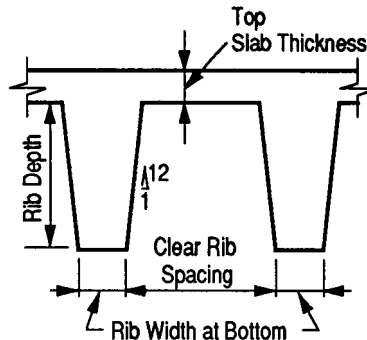


Figure 4-13 The required rib dimensions

Defining Longitudinal Beam Geometry

Longitudinal beam dimensions are required for the beam-supported slab and continuous beam systems. The span numbers are automatically sequenced from 1 to the number of spans. There are two different dialog boxes, one for beam-supported slab systems, and the other for continuous beam systems. *Note, the span length for longitudinal beams are entered here for the continuous beam system.*

To input geometry for beam-supported slab system:

1. Select the **Longitudinal Beams** command from the Input/Geometry menu. The dialog box of Figure 4-14 will appear.
2. Enter the width of the beam (see Figure 4-15).
3. Enter the depth of the beam from the top of the slab (see Figure 4-15).
4. Press the **Replace** button to update the longitudinal beam geometry.

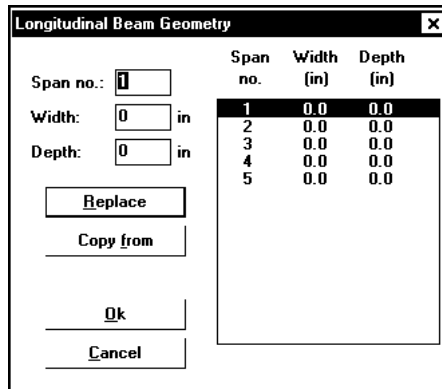


Figure 4-14 The Longitudinal Beam Geometry dialog box for beam-supported slab systems

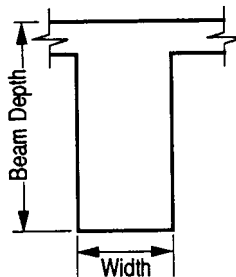


Figure 4-15 Required longitudinal beam dimensions

- Repeat steps 2 through 4 until all the beams have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
- Press **Ok** to exit the dialog box so that ADOSS will use the new beam geometry.

To input geometry for continuous beam system:

1. Select the **Longitudinal Beams** command from the Input/Geometry menu. The dialog box of Figure 4-16 will appear.
2. Enter the span length from column centerline to column centerline or edge to column centerline for the two cantilever spans. If there is no cantilever, half the column dimension should be entered as span length for cantilever. If the program detects a cantilever span length less than one-half the column dimension in the direction of analysis, c_1 , the cantilever length will be increased. If a partial load is affected by the span length, l_1 , a message warns the user of this condition.
3. Enter the width of the beam (see Figure 4-15).
4. Enter the depth of the beam from the top of the slab at the left end, center, and right end, respectively, if applicable (see Figure 4-17). For a prismatic beam, enter a value for the beam depth at the left end only.
5. Enter the length of the haunch at the left and right ends, respectively, from the face of the column, if applicable (see Figure 4-17). For a prismatic beam, leave these values blank or zero.
6. Press the **Replace** button to update the longitudinal beam geometry.
7. Repeat steps 2 through 6 until the beams have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
8. Press **OK** to exit the dialog box so that ADOSS will use the new beam geometry.

Longitudinal Beam Geometry [X]

Span no.: Depth: Left: Center: Right: in

Length: ft Haunch Length: Left: Right: ft

Width: in

Span no.	Length (ft)	Width (in)	Depth (in)			Haunch (ft)	
			Left	Center	Right	Left	Right
1	0.0	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	0.0
3	0.0	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	0.0
5	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Figure 4-16 The Longitudinal Beam Geometry dialog box for continuous beam-systems

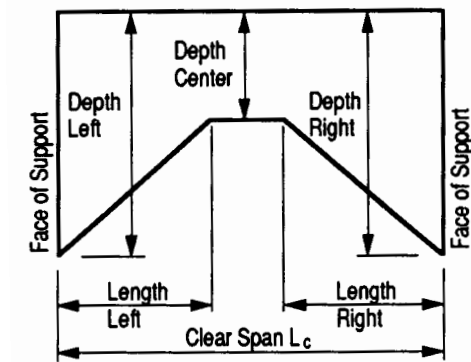


Figure 4-17 Required haunched beam dimensions

Defining Column and Capital Geometry

Column data is optional, if no column is specified at the joint, the joint is assumed hinged. Column numbers are automatically sequenced from 1 to the number of spans specified minus 1. You will be allowed to enter in column dimensions above and below.

To input column/capital geometry:

1. Select the Columns/Capitals command from the Input/Geometry menu. The dialog box of Figure 4-18 will appear.

Column no	loc	c ₁ (in)	c ₂ (in)	Height (ft)	Ext. (in)	Depth (in)
1	Abv	0.0	0.0	0.0		
	Blw	0.0	0.0	0.0	0.0	0.0
2	Abv	0.0	0.0	0.0		
	Blw	0.0	0.0	0.0	0.0	0.0
3	Abv	0.0	0.0	0.0		
	Blw	0.0	0.0	0.0	0.0	0.0
4	Abv	0.0	0.0	0.0		
	Blw	0.0	0.0	0.0	0.0	0.0

Figure 4-18 The Column/Capital Geometry dialog box

2. Enter a value for c_1 , the column dimension in the direction of analysis (see Figure 4-19).
3. Enter a value for c_2 , the column dimension perpendicular to the direction of analysis (see Figure 4-19). Round columns are specified with a zero input for c_1 , c_1 is then taken as the diameter. Zero values for both column dimensions will create a pin condition.

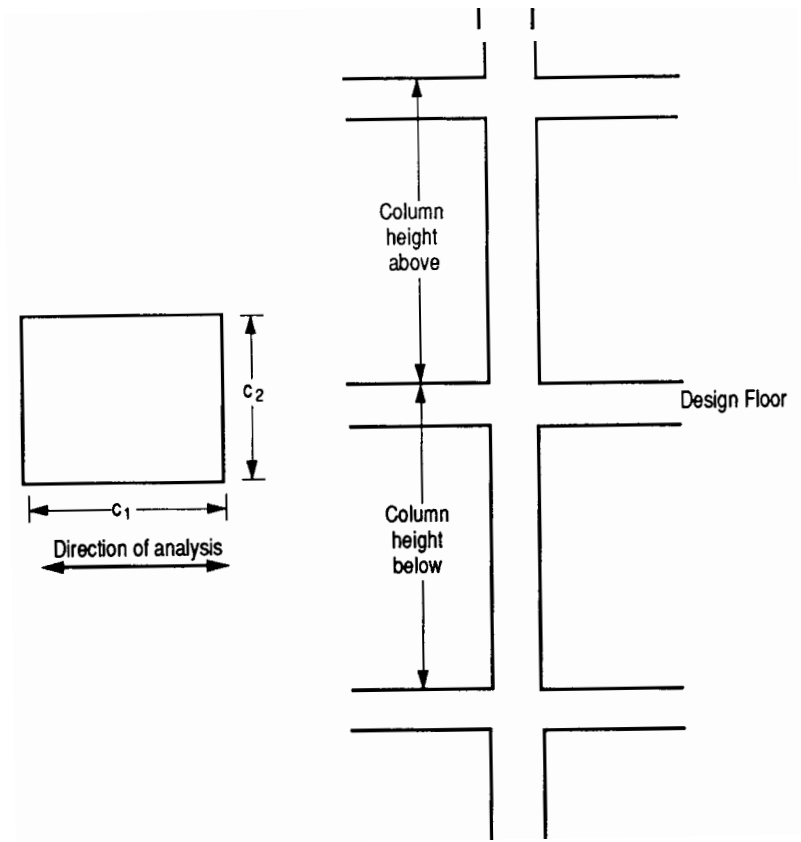


Figure 4-19 Required column dimensions

4. Enter the column height above, the distance from the top of the design floor to the top of the floor above (see Figure 4-19). ADOSS obtains the clear column height above by subtracting the average slab depth from the height given. Only the slab is considered for the floor system above. A zero dimension for the column heights above and below will create a >pin condition.
5. Enter the column height below, the distance from the design floor to the top of the floor below (see Figure 4-19). To obtain a clear column height

below, the slab/drop/beam depth is subtracted from the height given. Only the slab is considered for the floor system below. A zero dimension for the column heights above and below will create a pin condition.

6. Enter the capital extension which is the distance from the edge of the column to the end of the capital (see Figure 4-20).
7. Enter the capital depth which is the distance from the bottom of the soffit (slab, drop, or beam), to the bottom of the capital. The extension/depth ratio must be less than or equal to 1.0 (see Figure 4-20).

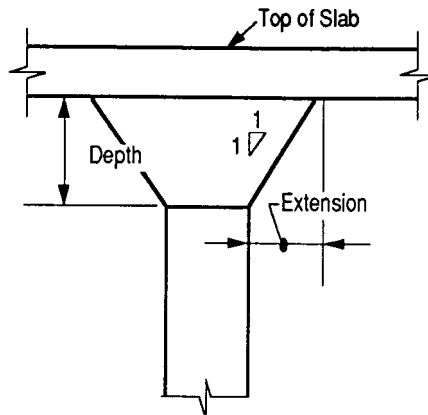


Figure 4-20 Required capital dimensions

8. Press the **Replace** button to update the slab geometry.
9. Repeat steps 2 through 8 until all the columns and capitals have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
10. Press **Ok** to exit the dialog box so that ADOSS will use the new data.

Defining Drop Geometry

Drops are only available for the flat slab or waffle slab systems. The drop numbers are automatically sequenced from 1 to the number of columns. The drop length and width dimensions are computed by ADOSS, based on slab span dimensions, when the check box marked “Standard Drop” is checked.

To input drop geometry

1. Select the **Drops** command from the Input/Geometry menu. The dialog box of Figure 4-21 will appear.
2. Select whether ADOSS should compute the drop dimensions or the dimensions will be user specified. If ADOSS is to compute the dimensions, the “Standard drop” option should be checked and then only the drop depth will be available. When the “Standard drop” option is selected ADOSS will calculate drop panel dimensions in accordance with ACI 318-95 Clause 13.3.7. Similar requirements contained in previous editions of the CSA A23.3 Standard have been removed from the 1994 edition. As a result the ACI minimum specifications for drop panels are also used in CSA A23.3 runs when the “Standard Drops” option is selected. If you would like to specify drop dimensions other than those computed by ADOSS, you must remove the check from the “Standard Drop” option. Using the mouse, click the left mouse button once on

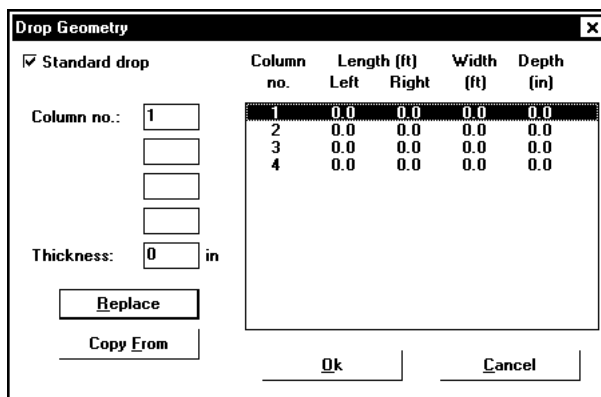


Figure 4-21 The Drop Geometry dialog box

Standard drop and the dimensions will become available. Using the keyboard, tab to Standard drop and press the space bar to remove the check and the dimensions will become available.

3. Enter the dimension in the direction of analysis from the column centerline to the edge of the drop left of the column (see Figure 4-22). If this is a standard drop, this dimension will not be available and the length left is set equal to the slab span length left/6 for interior columns or the left cantilever length for the first column.
4. Enter the dimension in the direction of analysis from the column centerline to the edge of the drop right of the column (see Figure 4-22). If this is a standard drop, this dimension will not be available and the length right is set equal to the slab span length right/6 for interior spans or the right cantilever length for the last column.

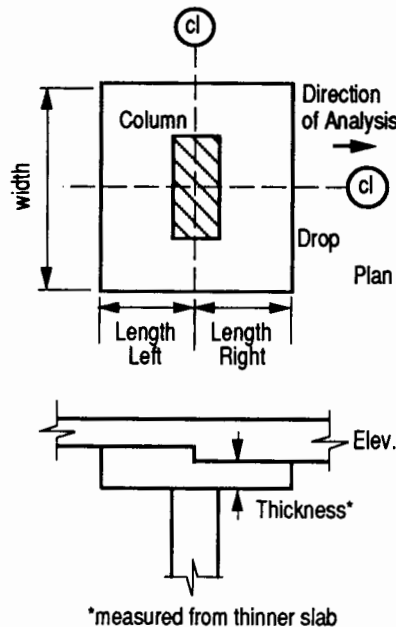


Figure 4-22 The required drop dimensions

ADOSS Graphics

5. Enter the width dimension in the transverse direction (see Figure 4-22). If this is a standard drop, this dimension will not be available and the width is set equal to slab width/3.
6. In order for ADOSS to recognize drops, drop depths are required for the flat slab systems even if Standard Drop is checked. Enter the depth of the drop from the span with the smaller slab depth (see Figure 4-22). For waffle slab systems, the depth is automatically assumed to be equal to the rib depth below the slab and is not displayed. A value entered will be considered to exist below the rib depth.
7. Press the **Replace** button to update the drop geometry.
8. Repeat steps 2 through 7 until all the drop dimensions have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
9. Press **Ok** to exit the dialog box so that ADOSS will use the new drop geometry.

Defining Transverse Beam Geometry

The Transverse Beam command allows you to input the width, depth, and eccentricity of transverse beams at each column. The beams are automatically sequenced from 1 to the number of columns. This command is not available for the continuous beam and one-way slab systems and it is optional for the remaining systems.

To input transverse beam geometry:

1. Select **Transverse Beams** from the Input/Geometry menu. The dialog box of Figure 4-23 will appear.
2. Enter the width of the transverse beam (see Figure 2-24).
3. Enter the depth of the transverse beam which is taken from the top of the slab to the bottom of the beam (see Figure 2-24).
4. Enter the eccentricity, which is measured from the joint centerline, positive to the right, and negative to the left of the joint (see Figure 2-24).
5. Press the **Replace** button to update the transverse beam dimensions.

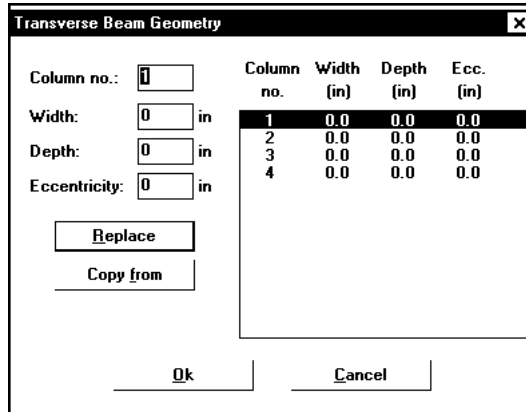


Figure 4-23 The Transverse Beam Geometry dialog box

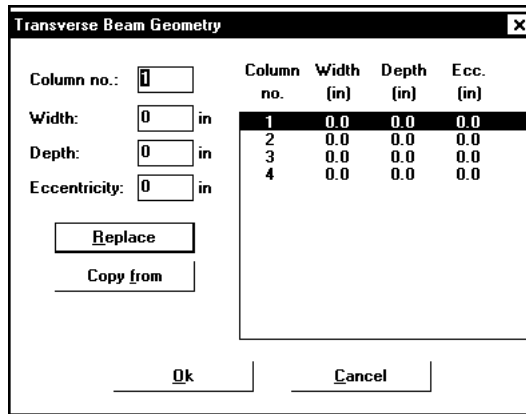


Figure 4-24 Required transverse beam dimensions

- Repeat steps 2 through 5 until all the beams have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
- Press **Ok** to exit the dialog box so that ADOSS will use the new beam geometry.

Changing the Column Fixity

You can change the percent of the actual column stiffness used in the analysis at all the joints. When the fixity lies between zero and 100%, the joint stiffness contribution by the column is multiplied by that percentage. Zero fixity indicates a pin support. The shear stresses are computed using the actual dimensions. For end columns, a fixity greater than 100% sets the column stiffness equal to a large number (E20), in effect creating a fully fixed joint. This applies to end columns only. The default fixity used by ADOSS is 100%, (i.e., the actual column stiffness).

To change column fixity:

1. Select **Column Fixity** under the Input menu. The dialog box of Figure 4-25 will appear.
2. Select the column whose fixity you would like to modify by clicking the left mouse button on the column in the list box to the right, or enter the column number in the “Column no.” edit box.
3. Press the **Replace** button to update the column fixity.
4. Repeat steps 2 through 3 until all the column fixity percentages have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
5. Press **Ok** to exit the dialog box so that ADOSS can use the new data.

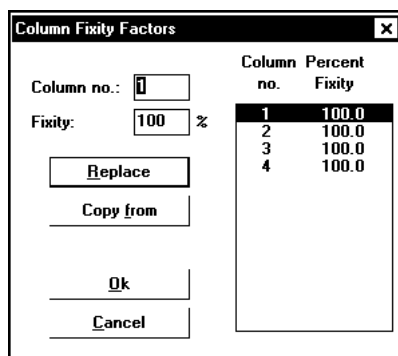


Figure 4-25 The Column Fixity Factors dialog box

Specifying Loads on the Floor System

To complete your analysis of a floor system, you must specify the applied loads. ADOSS computes the self weight of the floor system. There are several types of applied loads that may be entered, they are found in the Input/Loads menu. Surface loads are placed over the entire strip. Partial loads consist of uniform or trapezoidal loads, concentrated loads, and concentrated moments that may exist anywhere within the span length. Refer to Chapter 2 “Loading” for a detailed explanation of the superimposed loading on the floor system.

Specifying Uniform Surface Loads

Surface loads are uniform loads acting over the entire strip. These loads have units of psf(kN-m²). The method for inputting surface loads is the same as that used in entering the geometry.

To input surface loads:

1. Select the **Surface Loads** command from the Input/Loads menu. The dialog box of Figure 4-26 will appear.

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

Figure 4-26 The Surface Loads dialog box

2. Enter the unfactored superimposed dead and live loads acting over the entire area of the strip, positive surface loads act downward. ADOSS automatically computes the structure's self-weight.
3. Press the **Replace** button to update the surface loads.

4. Repeat steps 2 and 3 until the loads have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From** button).
5. Press **Ok** to exit the dialog box so that ADOSS will use the new data.

Specifying Partial Surface, Concentrated, and Moment Loads

You may enter uniform or trapezoidal loads that do not span from column centerline to column centerline in the direction of analysis. You may also specify concentrated vertical loads and moments. These loads are called partial loads and are input through the Partial Dead and Partial Live loads command of the Loads menu. Partial loads are assumed to act over the entire strip width.

To input partial dead and live loads:

1. Select the **Partial Dead** or **Partial Live** load command from the Input/Loads menu. The dialog box of Figure 4-27 will appear.
2. Enter the span number of the span that you would like to input the loads.
3. Choose either uniform, trapezoidal, concentrated loads, or moments from the radio buttons by clicking the left mouse button on the choice or tabbing to the buttons and using the up and down arrow keys. Up to 100 dead and 100 live loads may be entered.
4. Tab through the edit boxes and define unfactored dead or live load values and their locations. Depending on the type of loading, some or all of the edit boxes will be available. Figure 4-28 shows the input required and the following is a description of that input.

W_a For uniform loads, W_a is the uniform load in units of lbs/ft (KN/m), positive W_a is downward. For trapezoidal loads, W_a is the weight at the left end in units of lbs/ft (KN/m), positive W_a is downward. For concentrated loads, W_a is the force in units of kips (KN), positive W_a is downward, and for moments, W_a is the moment in units of ft-kips (KN-m), positive W_a is clockwise.

W_b For trapezoidal loads, W_b is the weight at the right end in units of lbs/ft (KN/m), positive W_b is downward. For all the other partial load types, W_b is not available.

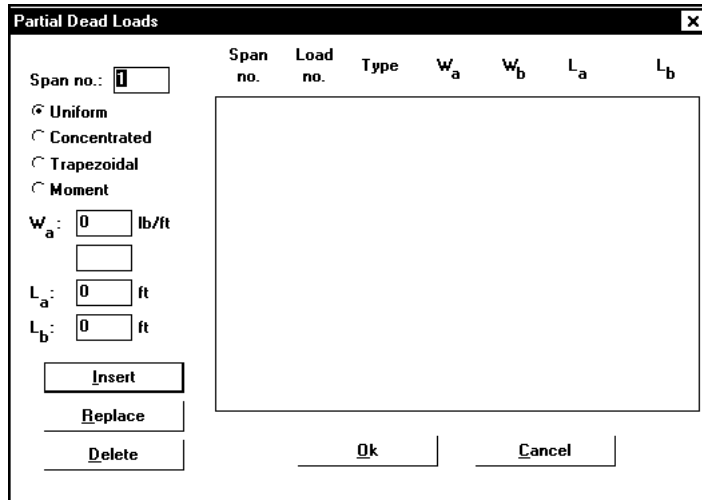


Figure 4-27 The Partial Dead Loads dialog box

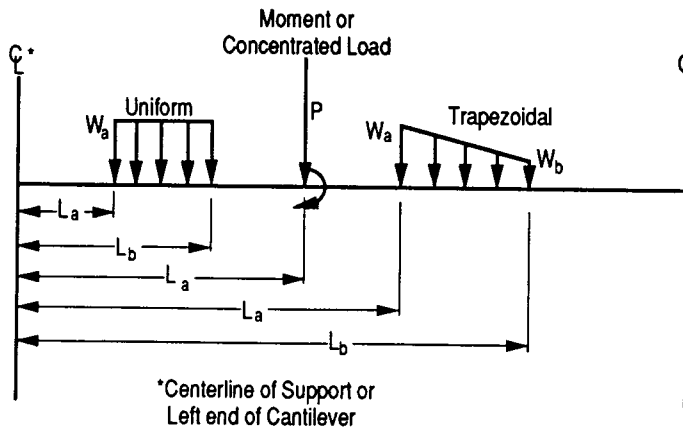


Figure 4-28 The required partial load data

- L_a For uniform and trapezoidal loads, L_a is the distance where the load begins from the centerline of the column at the left of the span, in units of ft (m). For concentrated loads and moments, L_a is the distance where the load exists from the centerline of the column at the left of the span in units of ft (m).
- L_b For uniform and trapezoidal loads, L_b is the distance where the load ends from the centerline of the column at the left of the span, in units of ft (m). For concentrated loads and moments, L_b is unavailable.

Although your particular loading may not actually act over the entire transverse width, all partial loading is converted internally by ADOSS to act over the full width of the slab. In the design direction, partial loads given as acting over less than 1/20 of the span length will be averaged over 1/20 by the program.

5. Select **Insert** to define the partial load.
6. Repeat steps 2 through 5 until all the partial loads have been entered then press **Ok** to exit the dialog box so that ADOSS will use the new loads.

To change partial dead and live loads data:

1. Select the partial load you want to change from the list box to the right by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Make your changes to the load by modifying the load type, the load magnitude, and/or location.
3. Select **Replace** to replace the old data with the new data.

To delete partial dead and live loads data:

1. Select the partial load you want to delete from the list box to the right by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Press the **Delete** button.

Specifying Lateral Loads

ADOSS will combine the gravity load analysis with a lateral load analysis. Lateral loads may be entered as forces at the design floor and the floor above, or entered as joint moments depending on the type of lateral load selected in the General Information dialog box. When the lateral loads are input as forces, ADOSS will perform a portal load analysis to obtain the joint moments. This method is an approximate method and uses the entire strip width and uncracked sections in the analysis which is not a recommended approach. A more accurate approach is to enter the lateral analysis through joint moments. These moments must be obtained by a frame analysis program such as PCA-FRAME program. Whichever method is used, the joint moments are combined with the gravity load moments to produce load patterns 5 through 8.

To input lateral load forces:

1. After the geometry has been entered, select the General Information command from the Input menu.
2. Select **Wind** from the Lateral Loads options.
3. Select **Forces** from the types of wind load options.
4. Select **OK**.
5. Select the **Lateral Forces** command from the Input/Loads menu. The dialog box of Figure 4-29 will appear.

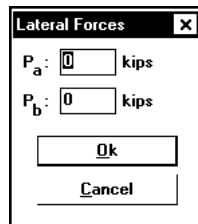


Figure 4-29 The Lateral Forces dialog box

6. Enter the total of the lateral loads of the floors above the design floor in the P_a edit box.
7. Enter the lateral load at the design floor in the P_b edit box.
8. Select **OK** to exit the dialog box so that ADOSS may use these lateral forces.

To input lateral load moments:

1. After the geometry has been entered, select the General Information command from the Input menu.
2. Select **Wind** from the Lateral Loads option.
3. Select **Moments** from the types of wind loads options.
4. Select **OK**.
5. Select the **Lateral Moments** command from the Input/Loads menu. The dialog box of Figure 4-30 will appear. The joint numbers are automatically sequenced from 1 to the number of columns.
6. Enter the moments at the joint left, right, top, and bottom; positive moments are clockwise.
7. Press the **Replace** button to update the joint moments.
8. Repeat steps 6 and 7 until all the joints have been updated. You can use the **Copy From** button as a short cut (see “Entering the Structure Geometry” earlier in this chapter for help on the **Copy From**).
9. Press **OK** to exit the dialog box and allow ADOSS to use the updated joint moments.

The dialog box titled "Lateral Moments" contains the following elements:

- Joint no.:
- Left: ft-k
- Right: ft-k
- Top: ft-k
- Bottom: ft-k
- Buttons: **Replace**, **Copy from**, **Ok**, **Cancel**
- Table of Moments (ft-kips):

Joint no.	Left	Right	Top	Bottom
1	0.0	0.0	0.0	0.0
2	0.0	0.0	0.0	0.0
3	0.0	0.0	0.0	0.0
4	0.0	0.0	0.0	0.0

Figure 4-30 The Lateral Moment dialog box

Changing the Live Load Reduction Factor

ADOSS uses 75% of the live load by default when considering patterned loading of load patterns 1 through 3. You may change this percentage.

To change the live load reduction factor:

1. Select the **Factors** command from the Input/Loads menu. The dialog box of Figure 4-31 will appear if the ACI code was selected in the General Information dialog box.
2. Enter the percentage of the live load to be used in pattern loading.
3. Press **Ok** to exit the dialog box so that ADOSS may use the new reduction factor.

Load Factors

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

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

$U_2 = 0.75 * ([] * \text{Dead} + [] * \text{Live} + 1.7 * \text{Latl})$

$U_3 = 0.9 * \text{Dead} + 1.3 * \text{Latl}$

Proportion lateral forces

Using code vertical distribution factors

Assigning percentage to column strip: 100 %

Ok Cancel

Figure 4-31 The Load Factors dialog box

Changing the Load Factors

ADOSS allows you to change the magnification factors applied to the loads. The default values depend on the code selected with the General Information command.

The default load factors for the ACI codes are:

$$U1 = 1.4D + 1.7L$$

$$U2 = .75(1.4D + 1.7L \pm 1.7W)$$

$$U3 = .90D \pm 1.3W$$

The default load factors for the CSA code are:

$$U1 = 1.25D + 1.5L$$

$$U2 = 1.25D + .7(1.5L \pm 1.5W)$$

$$U3 = .85D \pm 1.5W$$

To change load factors:

1. Select Factors from the Input/Load menu. The dialog box of Figure 4-31 will appear if the ACI code was selected in the General Information dialog box.
2. Use the tab key to jump from one load factor to the next and type in the desired load factor. *Note that the dead and live load factors of the first load case are linked to the dead and live load factors of the second load case; therefore, different factors for the dead and live loads cannot be input for the two cases.*
3. Select **OK** when all the desired load factors have been modified to exit the dialog box and allow ADOSS to use the new data.

Applying More Lateral Load to the Column Strip

If lateral loads are applied to the strip of a two-way floor system, ADOSS, by default, distributes the lateral forces to the column and middle strip using the same factors computed for vertical loads. ADOSS allows you to specify a portion of that lateral load directly to the column strip.

To apply more lateral load to the column strip:

1. Select **Factors** from the Input/Loads menu. The dialog box of Figure 4-31 will appear if the ACI code was selected in the General Information dialog box. *Note that this option will only be available for flat plate, flat slab, waffle slab, and beam-supported slab systems.*
2. Select the option “to proportion lateral forces by assigning a percentage to the column strip” from the radio buttons by clicking the left mouse button over the option or tabbing to the buttons and using the up and down arrow keys.
3. Enter a percentage of the lateral load to be applied directly to the column strip.
4. Press **OK** to exit the dialog box so that ADOSS may use the new percentage.

To use code vertical distribution factors:

1. Select **Factors** from the Input/Loads menu. The dialog box of Figure 4-31 will appear if the ACI code was selected in the General Information dialog box. *Note that this option will only be available for flat plate, flat slab, waffle slab, and beam-supported slab systems.*
2. Select the option “to proportion lateral forces by using the code vertical distribution factors” from the radio buttons by clicking the left mouse button over the option or tabbing to the buttons and using the up and down arrow keys.
3. Press **OK** to exit the dialog box so that ADOSS may use the code distribution factors.

Viewing the Input Data

ADOSS allows you to modify the geometry and load windows so that you can analyze your input data more closely. This section provides procedures on performing these functions.

Viewing the Floor System with the Scale

You may view the floor system with a scale in ft (m) along the left and top edge of the view windows. This allows you to check the structure dimensions or help in determining the location of a zoom.

To turn the scale on:

1. Select the **Scale On** command from the View menu. This command will redraw the floor system in full size, place the scale on the top and left of the window, and change the Scale On command to Scale Off.

To turn the scale off:

1. Select the **Scale Off** command from the View menu. This command will redraw the floor system in full size, remove the scale from the top and left of the window, and change the Scale Off command to Scale On.

Zooming in on the System

In order to view the floor system in greater detail, ADOSS allows you to magnify a portion of the floor system for closer analysis.

Using zoom with the mouse:

1. Select Zoom from the View menu. *Note that the cursor has changed to a cross.*
2. Move the cross cursor to the upper left corner of the portion of the system you want to enlarge.
3. Press and hold down the left mouse button while dragging the cursor to the lower right portion of the system enclosing the desired area within the dashed box. *Note the cursor has changed to an arrow.*
4. Release the mouse to enlarge that portion.

Using zoom with the keyboard:

1. Select the **Zoom** command from the View menu by pressing **[Alt] + [V]** , then type **[Z]** . *Note that the cursor has changed to a cross.*
2. Using the arrow keys, move the cross cursor to the upper left corner of the portion of the system you want to enlarge.
3. Press the space bar once and use the arrow keys to enclose the desired area within the dashed box. *Note the cursor has changed to an arrow.*
4. Press the space bar once again to enlarge that portion.

If you have altered your screen view using the Zoom command, select Redraw from the View menu to restore the figure to its original proportions.

Changing the Isometric View Angle

You can modify the X and Z angles at which the floor system is displayed in the Isometric View window. By default, the floor system is viewed at -45° about the X axis and 45° about the Z. The right hand rule is used to determine the angle of rotation about the X and Z axes and the X axis is always rotated first.

To change angles:

1. Select Change Isometric Angle from the View menu. The dialog box of Figure 4-32 appears displaying the currently set angles.
2. Change the angles of rotation about the X and Z axes. The X axis is rotated first then the Z axis is rotated. The floor system is rotated about the axis using the right hand rule.
3. Select **[OK]** to accept the new angles and to modify the isometric view.

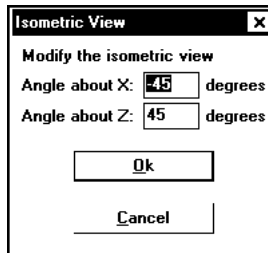


Figure 4-32 The Isometric View dialog box

Viewing Specific Member Types

In order to see the floor system members in greater detail, you can select specific member types to view by enabling and disabling them.

To select member types using the mouse:

1. Select the **View Members** command from the View menu. The dialog box of Figure 4-33 will appear showing the currently displayed member types. Depending on the floor system, some of the member types will be grayed and unavailable.
2. To hide a member type from a view of the floor system you must remove the X from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden member type, click the left mouse button on the check box to add the X to the check box.
3. Select **OK** to view only the selected member types.

To select member types using the keyboard:

1. Select the **View Members** command from the View menu. The dialog box of Figure 4-33 will appear showing the currently displayed member types. Depending on the floor system, some of the member types will be grayed and unavailable.

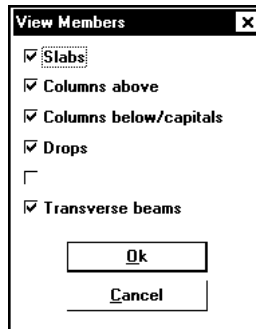


Figure 4-33 The View Members dialog box

2. To hide a member type from a view of the floor system you must remove the X from the check box. Tab to the member type you would like to hide and press the space bar to remove the X. To view a previously hidden member type, tab to the member type and press the space bar to add the X to the check box.
3. Select **OK** to view only the selected member types.

Viewing Specific Load Types

In order to see the partial loads in greater detail, you can select specific load types to view by enabling and disabling them.

To select load types using the mouse:

1. Select the **View Loads** command from the View menu. The dialog box of Figure 4-34 will appear showing the currently displayed load types.
2. To hide a load type from the partial loads window you must remove the X from the check box. Click the left mouse button on the check box of the load type you would like to hide. To view a previously hidden load type, click the left mouse button on the check box to add the X to the check box.
3. Select **OK** to view only the selected load types.

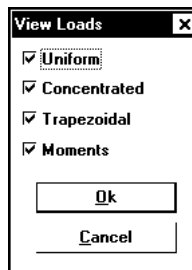


Figure 4-34 The View Loads dialog box

ADOSS Graphics

To select load types using the keyboard:

1. Select the View Loads command from the View menu. The dialog box of Figure 4-34 will appear showing the currently displayed load types.
2. To hide a load type from the partial loads window you must remove the X from the check box. Tab to the load type you would like to hide and press the space bar to remove the X. To view a previously hidden load type, tab to the member type and press the space bar to add the X to the check box.
3. Select **OK** to view only the selected load types.

Designing

Once all the geometry, material, reinforcement, and load data have been input, you can begin the design. The commands in the Design menu are used for this purpose.

Condensing the Output

The Condense/Expand Output command under the Input menu allows suppression of parts of the ADOSS output. ADOSS will print those items that have a check in the box. You must perform the design after executing the Condense/Expand Output command in order for ADOSS to print only the requested information.

To include or remove the desired output:

1. Select **Condense/Expand Output** from the Design menu. The dialog box of Figure 4-35 will appear. Figure 4-35 shows the output sections that will be printed by default.
2. To remove or add a check, click the left mouse button on the box or tab to the output item and press the space bar. The following is a list of the output options.

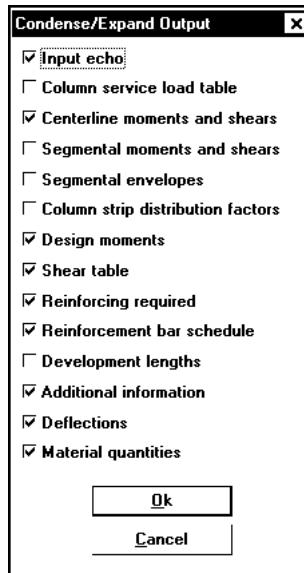


Figure 4-35 The Condense/Expand Output dialog box

Input Echo	An echo of the floor system input. The first page of the input, however, will always be printed whether this option is selected or not.
Column Service Load Table	The column service level moments and axial force table. This table is used for output to the PCACOL column design program. The axial loads represent the load of a single story only. A column can still be designed with ADOSS if this option is turned off.
Centerline Moments and Shears	Factored moments and shears at the left, right, top, and bottom of the centerline of the joint for each load case.
Segmental Moments of the and Shears	Factored moments and shears at the left and right ends of the span and at the center of 20 equal segments within. Shear and moment diagrams can still be viewed if this option is turned off.

ADOSS Graphics

Segmental Envelopes	The maximum factored top and bottom moments and shears as well as the governing pattern at the left and right ends of the span and at the center of 20 equal segments within. Shear and moment envelope diagrams can still be viewed if this option is turned off.
Column Strip Distribution Factors	Column strip lateral distribution factors used to compute the column strip, middle strip, and beam moments.
Design Moments	Factored design moments at the column face and in the span.
Shear Table	The punching shear calculations around the column and drop (if applicable) due to direct shear and with moment transfer.
Reinforcing Required	Required reinforcement area based on the moments at the design location.
Reinforcement Bar Schedule	Number of bars, bar number, and bar lengths required.
Development Lengths	The basic development lengths including all the factors used to determine the final development length.
Additional Information	Provided and required reinforcement, any additional reinforcement required at joint due to unbalanced moment and the total static design moments for each span.
Deflections	I-effective for the dead load and service load short-time deflection of the column and middle strips for the dead, live, and total loads. The live load deflection is computed as the difference between the total and dead load deflections.
Material Quantities	Table of the approximate total and unit quantities of concrete, formwork and reinforcement. The

includes allowance for vertical sides of drops, beams, and so forth. The reinforcement estimate is for one direction only and ignores items such as hooks, bends, and waste.

3. Press **OK** to exit the dialog box and allow ADOSS to print only the output portions requested.

Designing the Floor System

The **Design System** command starts the design portion of ADOSS after you have finished inputting all the data. If the PCACOL column design program is on the system, a design of the concrete columns will be allowed.

To design the system:

1. Select the **Design System** command from the Design menu. If any data required to analyze and design the system has not been input prior to executing this command, ADOSS will display a “Cannot design system, need more data” message. The dialog box of the required data will appear. Complete the data and select **OK**.

In order to save the shear and moment diagrams or the deflected shapes with the data file, you must save the data using the Save command after executing the Design System command.

Designing the Column

Once the floor system has been designed, you can design the columns. When ADOSS is started, it searches the current directory, then all the directories of the PATH for the PCACOL program. If it is found, ADOSS makes this command available.

The dimensions are taken from the specified column above or below. The moments at the column top are the same moments obtained from the ADOSS analysis joint bottom moments. The moments at the column bottom are the same moments obtained from the ADOSS analysis joint top. The moments passed to the PCACOL program are therefore centerline moments and the axial force is the sum of the shears left and right of the joint. Therefore, PCACOL will design the column section which frames into the joint at the specified column.

To design the column using the mouse:

1. Bring the elevated view window to the top of the desktop by selecting Elevated from the Window menu.
2. Double click the left mouse button on the desired column. ADOSS creates a PCACOL compatible data file and executes the PCACOL program.
3. Once in the PCACOL program, enter any additional loads on the column, then design the column using the methods found in the PCACOL manual.

To design the column using the keyboard:

1. Select the **Design Column** command from the Design menu. The dialog box of Figure 4-36 will appear.
2. Select the column number of the column to design.

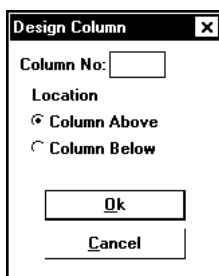


Figure 4-36 The Design Column dialog box

3. Select the location of the column, above or below the design slab, to be designed from the radio buttons by tabbing to the locations and using the up and down arrow keys to select.
4. Press **Ok** to exit the dialog box and to execute the PCACOL program.
5. Once in the PCACOL program, enter any additional loads on the column, then design the column using the methods found in the PCACOL manual.

Viewing and Printing Design Results

Once the analysis and design is performed, you can view and print the results, shear and moment diagrams, and deflected shapes (deflected shapes of only the continuous beam and one-way slab may be viewed). This section provides procedures on performing these functions.

Viewing and Printing the Results

After performing the design of the floor system, you are able to view and print the design results. ADOSS saves the design results into a file with the same name (i.e., the first part of a filename with a maximum length of 8 characters) of the ADOSS data file and an extension of .OUT. If the data has not been given a filename, the design results filename will be UNTITLED.OUT.

Selecting options in the Condense/Expand Output command from the Design menu allows you to select portions of the results to be printed. The Condense/Expand Output command must be executed prior to the design.

To view the design results:

1. Select the **View Results** command from the View menu. The PCA View utility will be used to display the design results. The PCA View window will overlap the main window of the ADOSS program, displaying the results file.
2. You can use the mouse or the cursor keys to scroll through the results.
3. Select the Exit command or press **[Alt] + [E]** to return to the ADOSS program.

To print the analysis:

1. Select the **Print Results** command from the File menu to send the design results to the printer. A message box will be displayed informing you that the file is being printed. Press the **[Cancel]** button only if you would like to terminate the print job.

Printing the Floor System Geometry

You may print the floor system geometry at any time. The plan, elevated, and isometric views will always be printed regardless of the number of view windows opened.

To print the floor system geometry:

1. Select the **Print Screen** command from the File menu. The dialog box of Figure 4-37 will appear. The view windows will be the default window type to print.
2. Press **Ok** to print the floor geometry.

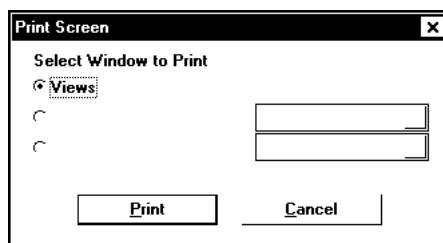


Figure 4-37 The Print Screen dialog box

Viewing and Printing Shear and Moment Diagrams

Once the design has been performed, you may view and print the shear and moment diagrams for any span at any available loading pattern. The shear and moment window will be split in half horizontally; the shear diagram will occupy the upper half and the moment diagram will occupy the lower half where each diagram will be scaled to fill the entire half of the window. In order to save the shear and moment diagram information with the data file, you must save the data using the Save command after executing the Design System command.

To view shears and moments:

1. Select the **View Shears/Moments** command from the View menu. The dialog box of Figure 4-38 will appear.
2. Enter the span number in the dialog box displayed for which you would

like to see the shear and moment diagrams. If an invalid number is specified an error message will inform you of it.

3. Select from the radio buttons whether a specific load pattern or the shear and moment envelopes should be displayed.
4. If the load pattern is to be displayed, enter the pattern number for which you would like to see the shear and moment diagrams. If an invalid number is specified, an error message will inform you of it.
5. Press **Ok**. A new window will display the specified shear and moment diagram. This new window's title is now displayed as a command in the Window menu. Selecting the title will bring this shear and moment diagram window to the front and make it the active window.

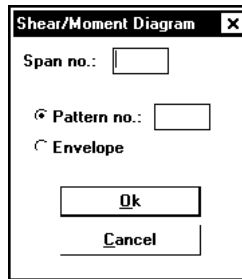


Figure 4-38 The Shear/Moment Diagram dialog box

To print shears and moments:

1. In order to print a shear and moment diagram, you must first view it using the procedure above.
2. Select the **Print Screen** command from the File menu. The dialog box of Figure 4-37 will appear.
3. Select the Shear/Moment Diagrams option from the radio buttons. The Shear/Moment Diagrams option will be grayed if either the Design System command has not been executed or no shear and moment diagrams have been viewed.
4. Select the shear and moment diagram you would like to print from the provided list located to the right of the radio button by clicking the mouse

on the down arrow to select a window or press **Shift** + the down arrow key until the desired window appears.

5. Press **Print** to send the diagram to the printer. You may change the margins of the screen print by selecting the Page Layout command (see “Changing the Margins of a Screen Print” later in this chapter).

Viewing and Printing the Deflected Shapes

Once the design has been performed, you may view the deflected shape of the one-way slab and continuous beam systems. *This command will not be available for the other floors systems.* The dead, live, and total load deflections will be superimposed in one graph. The diagram will be scaled to fill the entire window. In order to save the deflected shapes information with the data file, you must save the data using the Save command after executing the Design System command.

To view deflected shapes:

1. Select the View Deflections command from the View menu. The dialog box of Figure 4-39 will appear.
2. Enter a span number in the dialog box displayed for which you would like to see the deflected shape. If an invalid span number is specified an error message will inform you of it.
3. Press **Ok** . A new window will display the deflected shapes of the specified span. This new window’s title is now displayed as a command in the Window menu. Selecting the title from the Windows menu will bring this deflection window to the front and make it the active window.

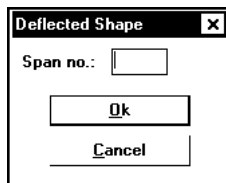


Figure 4-39 The Deflected Shape dialog box

To print deflected shapes:

1. In order to print a deflected shape, you must first view it using the procedure above.
2. Select the **Print Screen** command from the File menu. The dialog box of Figure 4-37 will appear.
3. Select the Deflection Diagrams option from the radio buttons. The Deflection Diagrams option will be grayed if either the Design System command has not been executed, no deflection diagram has been viewed, or the system is not a one-way slab or continuous beam.
4. Select the deflection diagram you would like to print from the provided list located to the right of the radio button by clicking the mouse on the down arrow to select a window or press **[Shift]** + the down arrow key until the desired window appears.
5. Press **Print** to send the diagram to the printer. You may change the margins of the screen print by selecting the Page Layout command (see “Changing the Margins of a Screen Print” later in this chapter).

Customizing ADOSS

You can change the colors used by ADOSS as well as default units, code, data directory, and margins used at startup. These commands are located in the options menu. This section provides procedures on customizing ADOSS.

Changing the Background Color

You can change the background color of the view windows to either black or white. The member colors will be reset to the default value for that particular background color. These changes are saved in the PCAPROGS.INI file and they become the default settings when ADOSS is executed.

To change the background color:

1. Select the **Background Color** command from the Options menu. The dialog box for Figure 4-40 will appear.

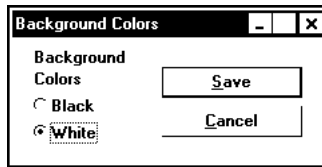


Figure 4-40 The Background Colors dialog box

2. Choose either black or white from the radio buttons by clicking the left mouse button over the desired color or using the up and down arrow keys to select a color.
3. Press **Save** to store the color and change the view windows' background.

Changing the Member Colors

You can change the colors of the different member types. *You may only change the member colors on a computer that supports color.* There are eight colors to choose from. When the background color is changed, the member colors will be reset to the default values for that particular background. You may want to change the member color after the background has been changed. These changes are saved in the PCAPROGS.INI file and they become the default settings when ADOSS is executed.

To change member colors using the mouse:

1. Select the **Member Colors** command from the Options menu. The dialog box of Figure 4-41 will appear.
2. Select the colors from the box to the left of the member type by clicking the left mouse button on the down arrow to scroll through the selections, then click on the desired color.
3. Press **Save** to store the member colors and redraw the floor system with the new colors.

To change member colors using the keyboard:

1. Select the **Member Colors** command from the Options menu. The dialog box of Figure 4-41 will appear.
2. Press **Shift** + the down arrow key until the desired color appears.
3. Select **Save** to store the member colors and redraw the floor system with the new colors.

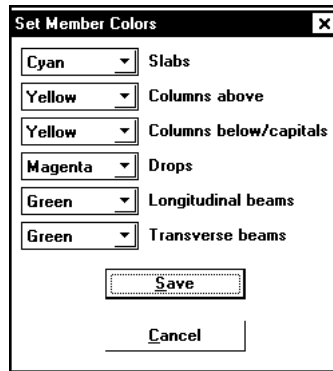


Figure 4-41 The Member Colors dialog box

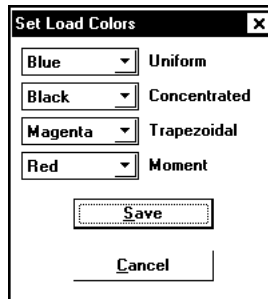


Figure 4-42 The Set Load Colors dialog box

Changing the Load Colors

You can change the colors of the different load types. <MI> You may only change the load colors on a computer that supports color.<D> There are eight colors to choose from. These changes are saved in the PCAPROGS.INI file and they become the default settings when ADOSS is executed.

To change load colors using the mouse:

1. Select the **Load Colors** command from the Options menu. The dialog box of Figure 4-42 will appear.

ADOSS Graphics

2. Select the colors from the box to the left of the load type by clicking the left mouse button on the down arrow to scroll through the selections, then click on the desired color.
3. Press **Save** to store the load colors and redraw the partial loads with the new colors.

To change load colors using the keyboard:

1. Select the Load Colors command from the Options menu. The dialog box of Figure 4-42 will appear.
2. Press **Shift** + the down arrow key until the desired color appears.
3. Select **Save** to store the load colors and redraw the partial loads with the new colors.

Changing the Default Units and Code

You can change the default units and code that ADOSS uses when it starts a new file. The default settings were set during the installation process. These changes are saved in the PCAPROGS.INI file and they become the default settings when ADOSS is executed.

To change the default units and code settings:

1. Select the Default Units/Code command from the Options menu. The dialog box of Figure 4-43 will appear.
2. Select the units and code that ADOSS will use as defaults from the radio buttons. These new values will be stored in the PCAPROGS.INI file.
3. Select **Save** to store the new settings.

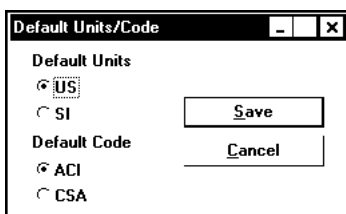


Figure 4-43 The Default Units/Code dialog box

Changing the Default Data Directory

When ADOSS is executed, the files of the default data directory are displayed in the Open dialog box when the Open command is executed. You may change the default data directory that ADOSS searches. This directory is saved in the PCAPROGS.INI file and it becomes the default setting when ADOSS is executed.

To change the default data directory:

1. Select the **Default Data Directory** command from the Options menu. The dialog box of Figure 4-44 will appear.
2. Enter a new data directory in the edit box that is displayed. Not entering a directory will cause ADOSS to search the current directory on startup.
3. Select **Save** to store the values in the PCAPROGS.INI file.

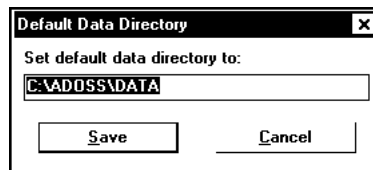


Figure 4-44 The Default Data Directory dialog box

Changing the Margins of a Screen Print

You may set the margins of output obtained using the Print Screen command. The margins you select will be saved in the PCAPROGS.INI file and become the default settings. These changes are saved in the PCAPROGS.INI file and they become the default settings when ADOSS is executed.

To set margins:

1. Select the **Page Layout** command from the Options menu. The dialog box of Figure 4-45 will appear.
2. Tab to each edit box and enter in new margins. The left margin is the distance from the left side of the page to where the drawing will begin. The right margin is the distance from the right side of the page to where

the drawing will end. The top margin is the distance from the page top to where the drawing will begin. The bottom margin is the distance to the page bottom to where the diagram will end. These margins are always entered in units of inches, despite the units setting in the General Information command.

3. Select **Save** to store the new margins in the PCAPROG.INI file.

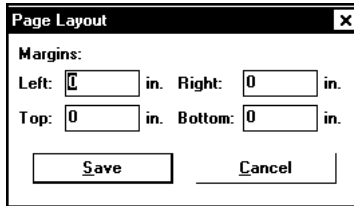


Figure 4-45 The Page Layout dialog box

Manipulating the Windows

After inputting the data for the floor system, you have three different ways to view the model and two windows for displaying loads. The Window menu enables you to arrange all the windows and select different view settings for them.

You may view your floor system in a Plan View, Elevated View, and Isometric View. The default view displayed at startup is the Plan View. The other two views are displayed as icons in the lower left hand corner of your screen.

The Plan View is a direct view of your system from above. The X and Y angles are 0°. The Elevated View displays your system from one side. This view is equivalent to an X axis angle of -90° and a Z axis angle of 0°. The Isometric View is a three dimensional view of your model where the default angle is -45° about the X axis and 45° about the Z axis. You may change this viewing angle by selecting Change Isometric Angle in the View Menu.

The Surface Loads window gives a graphical representation of the surface loads entered on all the spans and the Partial Loads window gives a graphical representation of the partial loads on all the spans.

There may be more windows displayed for a particular floor system. This depends on the number of shear and moment diagrams or deflected shape windows created. The following instructions apply to all the windows.

To open a window using the mouse:

1. Select **Restore** from the window's Control menu, or double click on the icon of the window you would like to view.

To open a window using the keyboard:

1. Press the **[Alt]** + **[W]** to open the Window menu.
2. Press the down arrow key to highlight the window you want to open and press **[Enter]** or press the number of the window you want to open. If there are more than nine windows open, the last command in the Window menu will be **More Windows**. When this command is selected, the window must be chosen from a list box that will appear.

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

1. Select **Cascade** so the windows become the same size and layer on top of each other with the title bar of each visible.
2. Select **Tile All Windows** so that the screen divides into equal parts and displays all the windows you have open.
3. Select **Tile View Windows** so that the open view windows are arranged on your screen without overlapping. If all three view windows are open, they will be arranged so that the Isometric View is a half page display and Plan View and Elevated View each share the other half.

Working with Windows

See "Working with Windows" in the Microsoft Windows manual for help on resizing, moving, shrinking, enlarging, and closing windows.

Getting On-Line Help

The Help menu includes commands that enable you to obtain help for the ADOSS program.

To obtain a list of all the help topics:

1. Select **Index** to get a list of all the available help topics. The Index list is a combination of the **Commands** and **Procedures** help screens.

To obtain a list of the commands:

1. Select **Commands** to obtain a list of all the menu items.

To obtain a list of the procedures:

1. Select the **Procedures** command to obtain a list of important topics in designing a floor system.

To obtain help within the help system using the mouse:

1. Click once on an underlined topic to obtain information about it. Whenever available, these screens will advise you on where to find additional information on a topic.

To obtain help within the help system using the keyboard:

1. Use the tab key to select an underlined topic. Use the down arrow key to scroll through all the commands and make them accessible.
2. Press **Enter** on the underlined topic to obtain more information about it.

To obtain definitions using the mouse:

1. You may obtain a definition of a word that is underlined with a dash by holding down the left mouse button on the word. A definition of the word will be displayed in a pop-up box.
2. Release the mouse button and the pop-up box will disappear.

To obtain definitions using the keyboard:

1. Use the tab key to highlight the dashed underlined words.
2. Once the word is highlighted, press and hold down the **Enter** key. A

definition of the word will be displayed in a pop-up box.

3. Release the **Enter** key and the pop-up box will disappear.

Additional help features are available in the boxes located under the menu line. Those available in the ADOSS program are: Index, Back, and Search.

To access the help index within the help system:

1. Click on the **Index** box to get to the Index help screen.

To jump back to the previous help screen:

1. Click on the **Back** box to return to the previous help screen.

To find help on a specific word or phrase:

1. Click on the **Search** box to help you locate information in two ways: either scroll through a list of topics or type text in the Search For text box.

For additional instruction on getting on-line help, refer to the Using Help command in the Help menu.

Obtaining the ADOSS Version Number

The About box contains the copyright information, the version number of the ADOSS program, your firm name, city, and state.

To obtain the version number:

1. Select the **About ADOSS** command from the Help menu. The dialog box of Figure 4-46 will appear.
2. Press **Ok** to exit the dialog box.

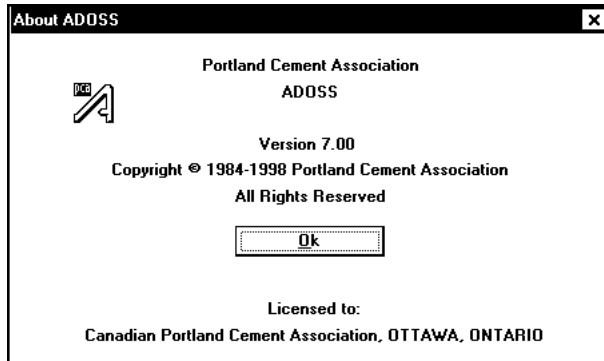


Figure 4-46 The About ADOSS dialog box

Chapter 5

Output Description

Output Elements

Depending on the floor system and the output options selected, the printed output will contain a selection of the following items (see the illustrated examples in Appendix C):

Title Sheet

General Information

Span and Column Data

Lateral Load and Output Information

Service Loads for PCACOL Slenderness Design

Statics Printout for Gravity Load Analysis

Statics Printout for Lateral Load Analysis

Distance to Inflection Points from Supports

Segmental Moments and Shears

Segmental Moment and Shear Envelopes

Design Moment Envelopes at Supports

Design Moment Envelopes in Span

Column Strip Lateral Distribution Factors at Supports

Column Strip Lateral Distribution Factors in Spans

Distribution of Design Moments at Supports

Distribution of Design Moments in Span

Shear Analysis

Beam Shear Requirements

Transverse Beam – Shear and Torsion Requirements

Negative Reinforcement

Positive Reinforcement

Development Length in Tension at Supports

Development Length in Tension in Spans

Reinforcement Schedule

Additional Reinforcement Requirements at Supports

Additional Information for In-Span Conditions

Integrity Reinforcement (CSA Standard Only)

Deflection Analysis

Quantity Estimates

Most of the output prints in tabular form with resulting data grouped at columns or spans. The next section discusses output in further detail.

Program Version

The program version number appears at the top of each output page along with the date, run time, and licensee's name.

Description of Input Echo

The input echo refers to all the data used in the analysis. ADOSS defaults common data; all other data must be input. Carefully check the data echo and compare it with the intended design system.

The Title Sheet displays details of the program's originating source and disclaimer.

General Information lists details of concrete, structure type, control parameters, number of spans, slab thickness, and so forth.

ADOSS Graphics

Span and Column Data lists the span lengths, applied loads, column sizes, drop dimensions, slab widths, design strips, column strips, and middle strips used by the program.

Note that the information printed out at this stage corresponds exclusively to the user's specified data.

Lateral Load and Output Information lists the output parameters and either lateral loads or joint moments.

CODE Checks

Prior to analysis, ADOSS tests the input for the following additional Code requirements:

- required slab thickness
- drop panel dimensions
- column capital dimensions

ADOSS prints a message when an item fails to meet Code requirements or when an item's value has been adjusted by the program. ADOSS does not check cantilever thickness; these calculations must be done manually.

Analysis Elements

Service Loads for PCACOL Slenderness Design. This table displays service load moments and shears due to Dead, Live, and Lateral (Latr.) for each load case. The moments at the column top are the service moments from the "Joint Moment Table" at the joint bottom. The moments at the column bottom are the service moments from the "Joint Moment Table" at the joint top. Therefore, these moments are applied to the column sections that frame into the joints. Note, these moments are at the joint centerlines. The axial loads 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.

Cancel

Statics Printout for Gravity Load Analysis. This table lists the centerline moments and shears corresponding to one or more gravity patterns 1 through 4. The

sign convention used in this printout is shown in Figure 5-1. Take notice that the printout corresponding to pattern 1 is based on the condition of live load acting on adjacent spans. Moments and shears for column 1 are based on the assumption of live loads acting on spans 1 and 2; moments and shears for column 2 are based on the assumption of live loads acting on spans 2 and 3, and so on.

Statics Printout for Gravity and Lateral Load Analysis. This output lists the centerline moments and shears corresponding to gravity and lateral loads – patterns 5 through 8. Note that lateral load effects are combined with vertical loads in accordance with the patterns described in Chapter 2.

Distance to Inflection Points from Supports. This chart specifies the inflection points from the center of the support on both sides of each support for all the loading patterns.

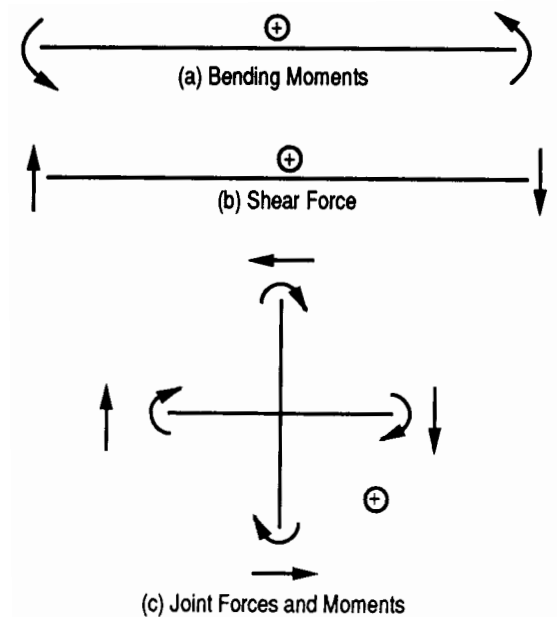


Figure 5-1 Sign convention adopted by ADOSS

Segmental Output. This table displays moments and shears at 22 locations along each span for each loading condition. There are 20 values at the center of each segment and one for each end of the span. Positive moments produce tension at the bottom and positive shears act upward.

Segmental Envelopes. Segmental moment and shear envelopes are computed both from vertical only loading patterns 1 through 4 and also from total loading patterns 1 through 8 when lateral loads are applied. This allows you to determine the effects produced by vertical and lateral forces as compared to those produced by vertical loads only. Positive moments produce tension at the bottom and positive shears act upward.

Design Moment Envelopes at Supports. This table displays the design moment envelopes, the distances to the critical sections from the centerline of the support, and the governing patterns, as well as the maximum inflection points from the centerline of the support and the governing patterns for the full equivalent frame. The total design moment envelopes are the moments produced by patterns 1 through 4 when no lateral loads are present, or patterns 1 through 8 when lateral loads are present. The vertical design moments will only be printed if lateral loads exist and they are the moments produced by patterns 1 through 4. The design moment envelopes, producing tension at the top and bottom of the floor system, are given at the left and right of the column.

Design Moment Envelopes in Spans. This table displays the design moment envelopes, the distances to the critical sections from the left support, and the governing patterns, as well as the maximum inflection points left and right of the critical sections and the governing patterns for the full equivalent frame. The total design moment envelopes are the moments produced by patterns 1 through 4 when no lateral loads are present, or patterns 1 through 8 when lateral loads are present. The vertical design moments will only be printed if lateral loads exist and they are the moments produced by patterns 1 through 4. The design moment envelopes, producing tension at the top and bottom of the floor system are given.

Column Strip Lateral Distribution Factors at Supports. This table displays the l_2/l_1 , α_1 , $\alpha l_2/l_1$, and β_t values and ultimately the percentage of the design moment at the supports to be distributed to the column strip, and the percentage of the column strip design moment to be assigned to the beam, if applicable. By default, this table is not printed. For a detailed discussion, see Chapter 2, "Design Moments".

Column Strip Lateral Distribution Factors in Spans. This table displays the l_2/l_1 , α_1 , and $\alpha l_2/l_1$, values and ultimately the percentage of the design moment in the spans to be distributed to the column strip, and the percentage of the column strip design moment to be assigned to the beam, if applicable. By default, this table is not printed. For a detailed discussion, see Chapter 2, “Design Moments”.

Distribution of Design Moments at Supports. This table displays the full equivalent frame strip width design moments as well as those distributed to the column strip, middle strip and the beam, if applicable, producing both tension at the bottom and the top. The total design moment is printed (vertical plus lateral) along with the difference between the total and vertical only load moments. This difference is that due to the effects of lateral loads. The difference will be zero if either no lateral loads exist on the floor system or the vertical load only case governs the design at the critical section. A percentage of the design moment to total moment is supplied after each design moment. As a default, design moments are distributed according to the code specified factors. Optionally, only moments obtained from vertical loading may be distributed according to these code factors while a greater percentage of the moment difference due to lateral forces may be assigned to the column strip and longitudinal beam, if present. Note that the percentages printed may not reflect those lateral distribution factors computed using the code. For a detailed discussion, see Chapter 2, “Design Moments”.

Distribution of Design Moments in Spans. This table displays the full equivalent frame strip width design moments as well as those distributed to the column strip, middle strip, and the beam, if applicable, producing both tension at the bottom and the top. The total design moment is printed (vertical plus lateral) along with the difference between the total and vertical only load moments. This difference is that due to the effects of lateral loads. The difference will be zero if either no lateral loads exist on the floor system or the vertical load only case governs the design at the critical section. A percentage of the design moment to total moment is supplied after each design moment. As a default, design moments are distributed according to the code specified factors. Optionally, only moments obtained from vertical loading may be distributed according to these code factors while a greater percentage of the moment difference due to lateral forces may be assigned to the column strip and longitudinal beam, if present. *Note that these percentages may not reflect those lateral distribution factors computed using the code.* For a detailed discussion, see Chapter 2, “Design Moments”.

Shear Analysis. The shear “around column” table displays the allowable shear stress, reactions, punching shear stress around the columns or column capitals, and lists two sets of punching shear calculations – direct shear alone and direct shear

ADOSS Graphics

with moment transfer. The calculation for moment transfer adjusts the unbalanced moment to the centroid of the critical section. The “shear transfer” is the unbalanced moment multiplied by γ_v . The shear “around drop/solid head” table displays the reactions and the punching shear stress around the drop. For a detailed discussion, see Chapter 2, “Shear Analysis of Slabs”.

Note: The Code¹ reconstructs the design equations for shear in terms of shear forces rather than shear stresses; however, ADOSS does not accommodate this change and computes shear strength in units of stress, psi or MPa. When shear stress is exceeded, ADOSS will warn: *SHEAR STRESS EXCEEDED.*

Design Elements

Beam Shear Requirements. This output contains two forms of tables. A short form displays the span length area to stirrup space value at d from the column face at both beam ends and at $0.175l_1$, $0.375l_2$, $0.675l_1$, and $0.825l_1$. A longer form (given when segmental output is selected) lists spacings for stirrup sizes at every 20th of the span, including at d . The program calculates shear using each side of the beam for each load pattern. Only the load patterns that have the largest shear are used for stirrup design. For a detailed discussion, see Chapter 2, “Shear Analysis of Longitudinal Beams”.

The beam-supported slab or continuous beam systems use the following types of shear computations: 1) total shear at beam ends with full design width, but lacking proportion, used for tributary area, and 2) proportioned shear based on slab beam stiffnesses used with the 45° tributary method. Chapter 2 explains the two computation methods in further detail. Both methods are user selectable.

Transverse Beam – Shear and Torsion Requirements. This two-table output contains the Shear and Torsion requirements for each transverse beam (the beams on the left and right transverse sides of each column). The design tables include design loads and capacities at d from the column transverse face, and the required stirrup area to spacing for shear (A_v/s), torsion (A_t/s), total (A_{tot}/s), and additional longitudinal bar area (A_l). The longitudinal area *must be* combined with the transverse beams’ flexural area. These tables serve as a design guide. The notes at the bottom of the tables describe further limitations. For a detailed discussion, see Chapter 2, “Shear and Torsion Analysis of Transverse Beams”.

¹ ACI 318-95, Chapter 11; CSA-A23.3-94, Clause 11

Negative Reinforcement. This table lists the corresponding design widths and reinforcement based on the moments printed in the Distribution of Design Moments at Supports table. The table indicates the side which requires the maximum area of reinforcement and prints either an “L” or an “R” to indicate which side of the support contains the design moment. The program checks the selected amount of reinforcement against the minimum requirements of the Code.² If a slab section cannot be singly reinforced, the program will respond with “*****”. In such cases, the engineer should consider revising the slab thickness. Only continuous beam systems will be designed for compression reinforcement. *Note that if computations indicate that a positive moment occurred at the support, a message will warn of this condition, and the section must be designed manually.* For a detailed discussion, see Chapter 2, “Area of Reinforcement”.

Positive Reinforcement. This table lists the corresponding design widths and reinforcement based on the moments printed in the Distribution of Design Moments in Spans table. The program checks the selected amount of reinforcement against the minimum requirement of the Code.² If a slab section cannot be singly reinforced, the program will prompt with “*****”. In such cases, the engineer should consider revising the slab thickness. Only continuous beam systems will be designed for compression reinforcement. *Note that if computations indicate that a negative moment occurred within the span, a message will warn of this condition, and the section must be designed manually.* For a detailed discussion, see Chapter 2, “Area of Reinforcement”.

Reinforcement Schedule. This schedule is intended as a guide for proper bar placement. ADOSS selects the bar size according to the slab thickness and maximum spacing requirements of the Code.³ The program selects the bar lengths based on the computed inflection points and the recommended minimums of the Code.⁴ For continuous beam systems, bars are placed in single a layer (see Figure 2-27), provided there is sufficient beam width. Hooks and bends are not included in bar length figures. For a detailed discussion, see Chapter 2, “Reinforcement Selection”.

Development Lengths in Tension at Supports. Development lengths are only computed for the top reinforcement of the column strips, middle strips and beams, if applicable, at supports. General factors such as: layer location, lateral support provided to beam reinforcement, light-weight concrete, yield strength, and epoxy-coated

2 ACI 318-95, 7.12 and 10.5; CSA A23.3-94, 7.8 and 10.5

3 ACI 318-95, 3; CSA A23.3-94 13.11.3

4 ACI 318-95, 13.4.8; CSA A23.3-94, 13.12.5

covering are printed at the top of the table. Specific values such as basic development length (LDB), concrete cover (CON/COV), bar clear spacing (CLEAR/SPAC), and code case factor (CODE/CASE), as well as the remaining three modifying factors of concrete cover and spacing (COV/SPA), top reinforcement (TOP/RFT) and excess of reinforcement (EXC/RFT) are printed for long (LG) and short (SH) bars. *Note that the last three factors correspond to factors 1, 4, and 6 as defined in Table 2.9.* For a detailed discussion, see Chapter 2, “Reinforcement Selection”.

Development Lengths in Tension in Spans. Development lengths are only computed for the bottom reinforcement of the column strips, middle strips and beams, if applicable, in the spans. General factors such as: layer location, lateral support provided to beam reinforcement, light-weight concrete, yield strength, and epoxy-coated covering are printed at the top of the table. Specific values such as basic development length (LDB), concrete cover (CON/COV), bar clear spacing (CLEAR/SPAC), and code case factor (CODE/CASE), as well as the remaining three modifying factors of concrete cover and spacing (COV/SPA), top reinforcement (TOP/RFT), and excess of reinforcement (EXC/RFT) are printed for long (LG) and short (SH) bars. *Note that the last three factors correspond to factors 1, 4, and 6 as defined in Table 2.9.* For a detailed discussion, see Chapter 2, “Reinforcement Selection”.

Additional Information at Supports. This table lists the maximum unbalanced moment – fraction of unbalanced moment transferred by flexure, γ_f , the resulting moment transferred by flexure, the corresponding pattern number, and the critical selection width as per the Code.⁵ When additional reinforcement is required, it is selected based on the bars size at the support. For a detailed discussion, see Chapter 2, “Area of Reinforcement” and “Additional Reinforcement at Support”.

The first part of the output table compares the required reinforcement with the provided reinforcement. Note that the required amount of reinforcement is extracted from the table “Negative Reinforcement” and does not include the required area due to unbalanced moments.

Additional Information for In-Span Conditions lists the total static design moment for each span based on the average widths and loads on the span as per the Code.⁶ Partial loads are not taken into account. This table also includes a comparison of the required reinforcement and provided reinforcement for positive reinforcement.

5 ACI 318-95, 13.3.3.2; CSA-A23.3-94, 13.4.5.6

6 ACI 318-95, Eq. 13-3; CSA-A23.3-94 Eq. 13-16

Deflection Analysis lists the I-effective due to dead load and service load short-time deflection of the column and middle strips for dead loads, live loads, and summation. The loaded span values determine the dead load deflections, while the difference between the total deflection and full dead deflections equals the full live load deflections. Note that the deflection table should be used in conjunction with the results obtained from the analysis of the perpendicular direction. For a detailed discussion, see Chapter 2, “Deflection Analysis”.

Quantity Estimates lists the approximate total and unit quantities of concrete, formwork, and reinforcement. The formwork includes allowance for vertical sides of drops, beams, and so forth. Note that the reinforcement estimate is for one direction only and ignores items such as hooks, bends, and waste. For a detailed discussion, see Chapter 2, “Material Quantities”.

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

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

Appendix B

References

1. Kripanarayanan, K. M., *Analysis and Design of Slab Systems*, Computer Program User's Manual, Portland Cement Association, SR184D, 1979.
2. Lee, B. H., *Analysis and Design of Flat Plates, Flat Slabs, Waffle Slabs, and Continuous Frames*, Portland Cement Association, 1972 (discontinued).
3. CRSI Handbook Based Upon the 1977 ACI Building Code and the 1980 Supplement, Concrete Reinforcing Steel Institute, Schaumburg, Illinois, 60195, 1982.
4. Sozen, M. A., and Siess, C. P., *Reinforced Concrete Floor Slabs – Design Methods: Their Evolution and Comparison*, Journal of the American Concrete Institute, Proceedings, V. 60, No. 8, August 1963, pp. 999-1028.
5. Hatcher, D. S., Sozen, M. A., and Siess, C. P., *Test of a Reinforced Concrete Flat Plate*, Proceedings of the American Society of Civil Engineers, V. 91, No. ST5, October 1965, pp. 205-231.
6. Guralnick, S. A., and LaFraugh, R. W., *Laboratory Study of a Forty-Five-Foot Square Flat Plate Structure*, Journal of the American Concrete Institute, Proceedings, V. 60, No. 9, September 1963, pp. 1107-1185.
7. Hatcher, D. S., Sozen, M. A., and Siess, C. P., *Test of a Reinforced Concrete Flat Slab*, Proceedings of the American Society of Civil Engineers, V. 95, No. St6, June 1969, pp. 1051-1072.
8. Jirsa, J. O., Sozen, M. A., and Siess, C. P., *Test of a Flat Slab Reinforced with Welded Wire Fabric*, Proceedings of the American Society of Civil Engineers, V. 92, No. ST3, June 1966, pp. 199-224.
9. Gamble, W. L., Sozen, M. A., and Siess, C. P., *Test of a Two-Way Reinforced Floor Slab*, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June, 1969, pp. 1073-1096.
10. Vanderbilt, M. D., Sozen, M. A., and Siess, C. P., *Test of a Modified Reinforced Concrete Two-Way Slab*, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June, 1969, pp. 1097-1116.

11. Shewmaker, R. E., Xanthakis, M., and Sozen, M. A., *Very Small Scale Reinforced Concrete Multi-Panel Flat Slabs*, Civil Engineering Studies, Structural Research Series No. 265, University of Illinois, June 1963, 171 pp.
12. Xanthakis, M., and Sozen, M. A., *An Experimental Study of Limit Design in Reinforced Concrete Flat Slabs*, Civil Engineering Studies, Structural Research Series No. 277, University of Illinois, December 1963, 159 pp.
13. Nichols, J. R., *Statistical Limitations Upon the Steel Requirement in Reinforced Concrete Flat Slab Floors*, Transactions of the American Society of Civil Engineers, V. 77, 1914, pp. 1670-1726.
14. Gamble, W. L., Sozen, M. A., and Siess, C. P., *Measured and Theoretical Bending Moments in Reinforced Concrete Floor Slabs*, Civil Engineering Studies, Structural Research Series No. 246, University of Illinois, June 1962, 322 pp.
15. Jirsa, J. O., Sozen, M. A., and Siess, C. P., *Pattern Loadings on Reinforced Concrete Floor Slabs*, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June, 1969, pp. 1117-1137.
16. Corley, W. G., Sozen, M. A., and Siess, C. P., *The Equivalent Frame Analysis for Reinforced Concrete Slabs*, Civil Engineering Studies, Structural Research Series No. 218, University of Illinois, June, 1961, 166 pp.
17. Jirsa, J. O., Sozen, M. A., and Siess, C. P., *The Effects of Pattern Loadings on Reinforced Concrete Floor Slabs*, Civil Engineering Studies Structural Research Series No. 269, University of Illinois, July 1963, 145 pp.
18. Corley, W. G., and Jirsa, J. O., *Equivalent Frame Analysis for Slab Design*, Journal of the American Concrete Institute, Proceedings, V. 67, No. 11, November 1970, pp. 875-884.
19. T ABS84, Computers and Structures, Inc., Berkeley, California, 1984.
20. ICES STRUDL II, Engineering User's Manual, Vols. 1 and 2, Department of Civil Engineering, Massachusetts Institute of Technology, R68-91/92, Cambridge, Massachusetts, 1968.
21. Hawkins, N. M., *Shear Strength of Slabs with Moments Transferred to Columns*, Special Publication SP42-35, American Concrete Institute, Detroit, Michigan, 1974, pp. 817-846.

22. ACI Committee 435, Subcommittee 7, *Deflections of Continuous Concrete Beams*, Journal of the American Concrete Institute, Proceedings V. 70, No. 12, December 1973, pp. 781-787.
23. Nilson, A. H., and Walters, D. B., Jr., *Deflection of Two-Way Floor Systems by the Equivalent Frame Method*, Journal of the American Concrete Institute, Proceedings, V. 72, No. 5, May, 1975, pp. 210-218.
24. ACI Committee 435, *State-of-the-Art Report, Deflection of Two-Way Floor Systems*, Special Publication SP43-3, American Concrete Institute, Detroit, Michigan, 1974.
25. Kripanarayanan, K. M., and Branson, D. E., *Short-Time Deflections of Flat Plates, Flat Slabs and Two-Way Slabs*, Journal of the American Concrete Institute, Proceedings, V. 73, No. 12, December 1976, pp. 686-690.
26. Branson, D. E., *Instantaneous and Time-Dependent Deflections of Simple and Continuous Reinforced Concrete Beams*, HPR Report No. 7, Pt. I, Alabama Highway Department in Cooperation with U.S. Department of Commerce, Bureau of Public Roads, August 1965.
27. Blakey, F. A., *Deformations of an Experimental Lightweight Flat Plate Structure*, Commonwealth Scientific and Industrial Research Organization (CSIRO), DBR Reprint No. 185, Civil Engineering Transactions, Institute of Engineers, V. CE 3, No. 1, Australia, March 1961.
28. Taylor, P. J., *Long-Term Deflection Calculation Methods for Flat Plates*, Constructional Review, V. 43, No. 2, Sydney, Australia, May 1970, pp. 68-74.
29. Heiman, J. L., *A Comparison of Measured and Calculated Deflections of Flexural Members in Four Reinforced Concrete Buildings*, Special Publication JSP43-21, American Concrete Institute, Detroit, Michigan, 1974.
30. Rangan, B. V., *Prediction of Long-Term Deflections of Flat Plates and Slabs*, Journal of the American Concrete Institute, Proceedings, V. 73, No. 4, April 1976, pp. 223-226.
31. Notes on ACI318-83 – Building Code Requirements for Reinforced Concrete with Design Applications, Portland Cement Association, 1984.
32. Elgabry, A. and Ghali, A., *Proposed Revisions to Building Code Requirements for Reinforced Concrete (ACI 318-83) (Revised 1986)*, ACI Structural Journal, V. 86 No. 3, May-June 1989, pp. 327-328.

Appendix C

Examples

This appendix contains examples created using ADOSS V6.0. There are no instructions given with these examples, however, the dialog boxes used to create the examples are given. Refer to Chapter 4 for help on operating procedures.

Example 1 Flat Slab

Analyze and design an interior strip shown in Figure C-1.

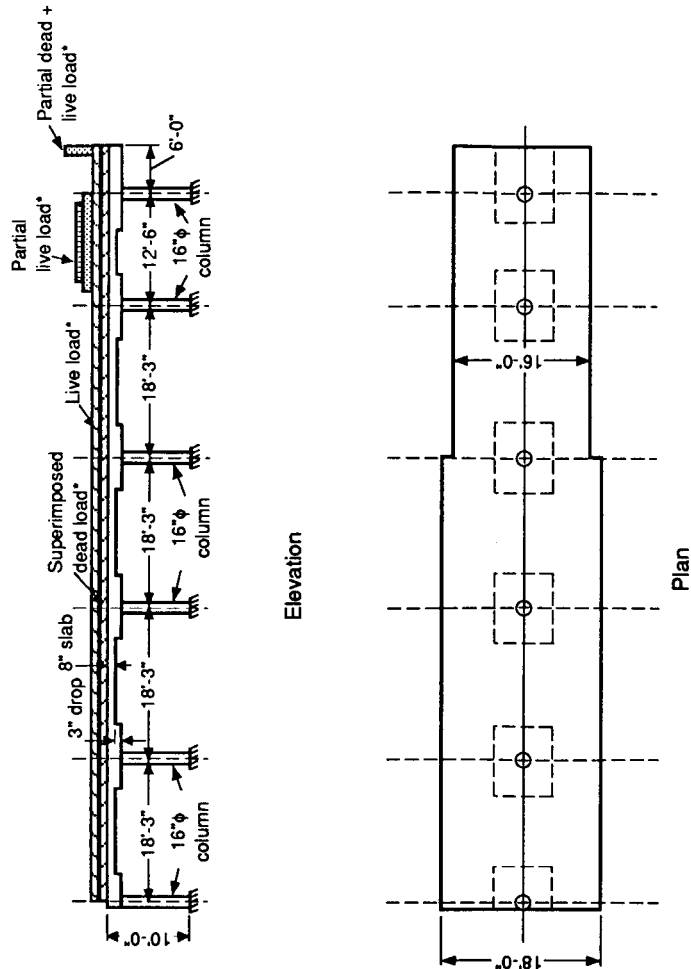


Figure C-1 Example flat slab problem

General Information

Number of spans:

Units
 U.S. in-lb
 SI metric

Code
 ACI 318-85
 CSA A23.3-94

Design Method
 Strength Design
 Alternate Design
 Limit States

Floor System
 Flat plate
 Flat slab
 Waffle slab
 Beam-supported slab
 Continuous beam
 One-way slab

Frame Location
 Interior
 Exterior

Lateral Loads
 None
 Wind
 Forces
 Moments

Loading Pattern
 1 through 4
 4 through 8
 1 through 8

Proportion moments and shear

Material Properties

Concrete

	Slabs	Beams	Columns	
Density:	<input type="text" value="150"/>	<input type="text" value="150"/>	<input type="text" value="150"/>	pcf
Type:	Normal	Normal	Normal	
f'_c :	<input type="text" value="4"/>	<input type="text" value="4"/>	<input type="text" value="4"/>	ksi
f_{ct} :	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	psi
f_s :	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	psi

Reinforcement

f_y : ksi

f_{yv} : ksi

Epoxy-coated

Slab Reinforcement Data

	Top	Bottom
Distance to reinforcement centroid from tension face:	<input type="text" value="3.0"/> in	<input type="text" value="1"/> in
Minimum bar number:	<input type="text" value="4"/>	<input type="text" value="4"/>
Layer location:	<input checked="" type="checkbox"/> Outer <input type="checkbox"/> Inner	<input checked="" type="checkbox"/> Outer <input type="checkbox"/> Inner

Minimum clear bar spacing: in

Slab Geometry

	Span no.	Length (ft)	Depth (in)	Width (ft)	
				Left	Right
Span no.:	<input type="text" value="1"/>				
Length:	<input type="text" value="0.67"/> ft				
Depth:	<input type="text" value="8"/> in				
Width left:	<input type="text" value="9"/> ft				
Width right:	<input type="text" value="9"/> ft				

Span no.	Length (ft)	Depth (in)	Width (ft) Left	Width (ft) Right
1	0.7	8.0	9.0	9.0
2	18.3	8.0	9.0	9.0
3	18.3	8.0	9.0	9.0
4	18.3	8.0	9.0	9.0
5	18.3	8.0	8.0	8.0
6	12.5	8.0	8.0	8.0
7	6.0	8.0	8.0	8.0

Column/Capital Geometry

Column no.:

Above Below
 c_1 : in
 c_2 : in
 Height: ft

Capital Geometry

Extension: in
 Depth: in

Column no.	Loc	c_1 (in)	c_2 (in)	Height (ft)	Ext. (in)	Depth (in)
1	Abv	0.0	0.0	0.0		
	Blw	16.0	0.0	10.0	0.0	0.0
2	Abv	0.0	0.0	0.0		
	Blw	16.0	0.0	10.0	0.0	0.0
3	Abv	0.0	0.0	0.0		
	Blw	16.0	0.0	10.0	0.0	0.0
4	Abv	0.0	0.0	0.0		
	Blw	16.0	0.0	10.0	0.0	0.0
5	Abv	0.0	0.0	0.0		
	Blw	16.0	0.0	10.0	0.0	0.0
6	Abv	0.0	0.0	0.0		
	Blw	16.0	0.0	10.0	0.0	0.0

Drop Geometry

Standard drop

Column no.:

Length left: ft

Length right: ft

Width: ft

Thickness: in

Column no.	Length (ft)	Width (ft)	Depth (in)
1	0.7	3.0	3.0
2	3.0	3.0	3.0
3	3.0	3.0	3.0
4	3.0	3.0	3.0
5	3.0	2.1	3.0
6	2.1	6.0	3.0

Surface Loads

Span no.:

Dead: psf

Live: psf

Span no.	Loads (psf)	
	Dead	Live
1	0.0	0.0
2	10.0	50.0
3	10.0	50.0
4	10.0	50.0
5	10.0	50.0
6	10.0	100.0
7	10.0	40.0

Partial Dead Loads

Span no.:

Uniform

Concentrated

Trapezoidal

Moment

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
7	1	UNIF	500.0		5.5	6.0

Partial Live Loads

Span no.:

Uniform
 Concentrated
 Trapezoidal
 Moment

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
6	1	UNIF	800.0		2.5	10.0
7	1	UNIF	50.0		5.5	6.0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 1
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

```

pppppp  ccccc  aaaaa
p   p  c   c  a   a
p   p  c   c   a
p   p  c   c  aaaaa
p   p  c   c  a   a
p   p  c   c  a   a
pppppp  ccccc  aaaaa
P
P
    
```

```

AAA  DDDDD  OOO  SSSSS  SSSSS
A  A  D  D  O  O  S  S  S  S
A  A  D  D  O  O  S  S
AAAAAAA  D  D  O  O  SSSSS  SSSSS
A  A  D  D  O  O  S  S  S  S  ( ttttt mm mm )
A  A  D  D  O  O  S  S  S  S  ( t m m m m )
A  A  DDDDD  OOO  SSSSS  SSSSS  ( t m m m )
    
```

```

*****
Computer program for ANALYSIS AND DESIGN OF SLAB SYSTEMS
*****
    
```

Licensee stated above acknowledges that Portland Cement Association (PCA) is not and cannot be responsible for either the accuracy or adequacy of the material supplied as input for processing by the ADOSS(tm) computer program. Furthermore, PCA neither makes any warranty expressed nor implied with respect to the correctness of the output prepared by the ADOSS(tm) program. Although PCA has endeavored to produce ADOSS(tm) error free the program is not and cannot be certified infallible. The final and only responsibility for analysis, design and engineering documents is the licensees. Accordingly, PCA disclaims all responsibility in contract, negligence or other tort for any analysis, design or engineering documents prepared in connection with the use of the ADOSS(tm) program.

Note: Further information and technical support on the ADOSS computer program is available through the CPCA Web site at: "www.cPCA.ca".

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 2
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

FILE NAME C:\ADOSS\DATA\FLTSLB.ADS
 PROJECT ID. FLAT SLAB -INTERIOR BENT-VERTICAL
 SPAN ID. -----
 Example 1

ENGINEER

DATE 01/01/99
 TIME 12:00
 UNITS U.S. in-lb
 CODE ACI 318-95
 SLAB SYSTEM FLAT SLAB SYSTEM
 FRAME LOCATION INTERIOR
 DESIGN METHOD STRENGTH DESIGN
 MOMENTS AND SHEARS NOT PROPORTIONED

NUMBER OF SPANS 7

SOLID HEAD DIMENSIONS : AS INPUT BY ENGINEER

CONCRETE FACTORS	SLABS	BEAMS	COLUMNS
DENSITY (pcf)	150.00	150.00	150.00
TYPE	NORMAL WGT	NORMAL WGT	NORMAL WGT
f'c (ksi)	4.00	4.00	4.00
density factor	1.00	1.00	1.00
fr (psi)	474.30	474.30	474.30

REINFORCEMENT DETAILS: NON-PRESTRESSED

YIELD STRENGTH F_y = 60.00 ksi
 DISTANCE TO RF CENTER FROM TENSION FACE:
 AT SLAB TOP = 1.70 in OUTER LAYER
 AT SLAB BOTTOM = 1.00 in OUTER LAYER
 MINIMUM FLEXURAL BAR SIZE:
 AT SLAB TOP = # 4
 AT SLAB BOTTOM = # 4
 MINIMUM SPACING:
 IN SLAB = 6.00 in

Example 1, Flat Slab

ADOSS Graphics

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 3
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

SPAN/LOADING DATA

SPAN NUMBER	LENGTH Tslab		WIDTH L2***		SLAB SYSTEM	DESIGN STRIP (ft)	COLUMN STRIP** (ft)	UNIFORM LOADS	
	L1 (ft)	(in)	LEFT (ft)	RIGHT (ft)				S. DL (psf)	LIVE (psf)
1*	.7	8.0	9.0	9.0	2	18.0	.0	.0	.0
2	18.3	8.0	9.0	9.0	2	18.0	9.0	10.0	50.0
3	18.3	8.0	9.0	9.0	2	18.0	9.0	10.0	50.0
4	18.3	8.0	9.0	9.0	2	18.0	9.0	10.0	50.0
5	18.3	8.0	8.0	8.0	2	16.0	8.0	10.0	50.0
6	12.5	8.0	8.0	8.0	2	16.0	6.3	10.0	100.0
7*	6.0	8.0	8.0	8.0	2	16.0	.0	10.0	40.0

- * -Indicates cantilever span information.
- ** -Strip width used for positive flexure.
- ***-L2 widths are 1/2 dist. to transverse column.
- "E"-Indicates exterior strip.

PARTIAL LOADING DATA

SPAN No.	LOAD No.	TYPE	PARTIAL DEAD LOADS				LOAD No.	TYPE	PARTIAL LIVE LOADS			
			Wa	Wb	La	Lb			Wa	Wb	La	Lb
1*												
2												
3												
4												
5												
6												
7*	1	UNIF	500.0	.0	5.5	6.0	1	UNIF	800.0	.0	2.5	10.0
							1	UNIF	50.0	.0	5.5	6.0

- * -Indicates cantilever span information.

UNITS FOR:

- UNIFORM LOAD: Wa.....plf La & Lb... ft
- CONCENTRATED LOAD: Wa.....kips La..... ft
- TRAPEZOIDAL LOAD: Wa & Wb..plf La & Lb... ft
- MOMENT: Wa.....ft-k La..... ft

NOTE: Local effects of partial loadings are NOT considered by ADOSS, compute manually.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 4
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

COLUMN/TORSIONAL DATA

COLUMN NUMBER	COLUMN ABOVE SLAB			COLUMN BELOW SLAB			CAPITAL**		COLUMN STRIP* (ft)	MIDDLE STRIP* (ft)
	C1 (in)	C2 (in)	HGT (ft)	C1 (in)	C2 (in)	HGT (ft)	EXTEN. (in)	DEPTH (in)		
1	.0	.0	.0	16.0	.0	10.0	.0	.0	9.0	9.0
2	.0	.0	.0	16.0	.0	10.0	.0	.0	9.0	9.0
3	.0	.0	.0	16.0	.0	10.0	.0	.0	9.0	9.0
4	.0	.0	.0	16.0	.0	10.0	.0	.0	8.0	8.0
5	.0	.0	.0	16.0	.0	10.0	.0	.0	6.3	9.8
6	.0	.0	.0	16.0	.0	10.0	.0	.0	6.3	9.8

Columns with zero "C2" are round columns.
 * -Strip width used for negative flexure.
 **-Capital extension distance measured from face of column.

COLUMN NUMBER	TRANSVERSE BEAM			DROP PANEL/SOLID HEAD				SUPPORT FIXITY* %
	WIDTH (in)	DEPTH (in)	ECCEN (in)	LEFT (ft)	RIGHT (ft)	WIDTH (ft)	THICK (in)	
1	.0	.0	.0	.7	3.0	6.0	3.0	100%
2	.0	.0	.0	3.0	3.0	6.0	3.0	100%
3	.0	.0	.0	3.0	3.0	6.0	3.0	100%
4	.0	.0	.0	3.0	3.0	5.3	3.0	100%
5	.0	.0	.0	3.0	2.1	5.3	3.0	100%
6	.0	.0	.0	2.1	6.0	5.3	3.0	100%

* -Support fixity of 0% denotes pinned condition.
 Support fixity of 999% denotes fixed end condition.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 5
2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

LATERAL LOAD/OUTPUT DATA

LATERAL LOADS ARE NOT SPECIFIED

OUTPUT DATA

PATTERN LOADINGS: 1 THRU 4
PATTERN LIVE LOAD FACTOR (1-3) = 100%

LOAD FACTORS:

U = 1.40*D + 1.70*L
U = .75(1.40*D + 1.70*L + 1.70*W)
U = .90*D + 1.30*W

OUTPUT OPTION(S):

Input Echo
Centerline Moments and Shears
Column Strip Distribution Fac
Shear Table
Reinforcing Required
Bar Sizing
Additional Information
Deflections
Material Quantities

**TOTAL UNFACTORED DEAD LOAD = 181.219 kips
LIVE LOAD = 93.915 kips

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 6
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

---- STATICS PRINT-OUT FOR GRAVITY LOAD ANALYSIS ----

J O I N T M O M E N T S (f t - k i p s)

JOINT NUMBER	PATTERN-1				PATTERN-2			
	LEFT	RIGHT	TOP	BOTTOM	LEFT	RIGHT	TOP	BOTTOM
1	-.6	50.0	.0	-49.4	-.6	24.7	.0	-24.1
2	-169.8	161.0	.0	8.7	-130.1	136.1	.0	-6.0
3	-139.6	141.4	.0	-1.8	-111.1	97.9	.0	13.2
4	-141.8	140.1	.0	1.7	-99.7	114.3	.0	-14.6
5	-118.8	114.9	.0	3.9	-91.4	68.8	.0	22.6
6	-78.3	71.2	.0	7.1	-52.3	71.2	.0	-18.9
JOINT NUMBER	PATTERN-3				PATTERN-4			
	LEFT	RIGHT	TOP	BOTTOM	LEFT	RIGHT	TOP	BOTTOM
1	-.6	51.3	.0	-50.7	-.6	46.1	.0	-45.5
2	-142.2	117.6	.0	24.7	-165.2	153.8	.0	11.4
3	-95.6	114.1	.0	-18.4	-125.9	129.3	.0	-3.4
4	-117.0	99.6	.0	17.4	-129.6	127.0	.0	2.7
5	-75.9	85.7	.0	-9.8	-108.7	105.4	.0	3.4
6	-65.7	51.4	.0	14.4	-74.8	71.2	.0	3.6

J O I N T S H E A R S (k i p s)

JOINT NUMBER	PATTERN-1		PATTERN-2		PATTERN-3		PATTERN-4	
	LEFT	RIGHT	LEFT	RIGHT	LEFT	RIGHT	LEFT	RIGHT
1	-1.8	34.8	-1.8	20.4	-1.8	35.2	-1.8	33.7
2	-47.0	43.2	-31.9	41.6	-45.2	27.4	-46.7	41.7
3	-41.0	41.7	-38.8	26.1	-25.0	40.0	-38.7	40.2
4	-41.8	38.8	-26.2	37.0	-40.3	24.6	-40.1	36.7
5	-36.3	42.3	-34.5	17.2	-22.0	39.5	-34.7	40.4
6	-37.2	23.3	-14.5	23.3	-36.3	16.7	-35.5	23.3

** - Negative moment encountered in span, analyze manually.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 7
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN MOMENT ENVELOPES AT CRITICAL SECTIONS FROM SUPPORTS

COL NUM	LOAD TYPE	CROSS SECTN	DESIGN MOMENT (ft-k)	DISTANCE CR. SECTN (ft)	LOAD PTRN	MAX. I. P. DISTANCE (ft)	LOAD PTRN
1	TOTL LEFT	TOP	-.4	.117	4	.670	1
		BOT	.0	.000	0	.000	0
	RGHT	TOP	31.3	.591	3	1.830	3
		BOT	.0	.000	0	.000	0
2	TOTL LEFT	TOP	-142.8	.591	1	5.490	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	136.3	.591	1	6.405	3
		BOT	.0	.000	0	.000	0
3	TOTL LEFT	TOP	-116.1	.591	1	5.490	3
		BOT	.0	.000	0	.000	0
	RGHT	TOP	117.5	.591	1	5.490	2
		BOT	.0	.000	0	.000	0
4	TOTL LEFT	TOP	-117.9	.591	1	5.490	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	117.9	.591	1	5.490	3
		BOT	.0	.000	0	.000	0
5	TOTL LEFT	TOP	-98.1	.591	1	4.575	3
		BOT	.0	.000	0	.000	0
	RGHT	TOP	90.8	.591	1	3.125	4
		BOT	.0	.000	0	.000	0
6	TOTL LEFT	TOP	-57.3	.591	1	2.500	4
		BOT	.0	.000	0	.000	0
	RGHT	TOP	58.1	.591	4	6.000	1
		BOT	.0	.000	0	.000	0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 8
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN MOMENT ENVELOPES AT CRITICAL SECTIONS ALONG SPANS

SPAN NUM	LOAD TYPE	CRITICAL SECTION (ft)	DESIGN MOMENT (ft-k)	LOAD PTRN	MAX. I.P. DIST LEFT (ft)	LOAD PTRN	MAX. I.P. DIST RGHT (ft)	LOAD PTRN
2	TOTL	7.778 TOP	.0	0	.000	0	.000	0
		BOT	87.9	3	6.863	2	6.862	3
3	TOTL	9.608 TOP	.0	0	.000	0	.000	0
		BOT	58.5	2	5.033	2	5.032	2
4	TOTL	8.692 TOP	.0	0	.000	0	.000	0
		BOT	66.0	3	5.032	3	5.948	3
5	TOTL	9.608 TOP	.0	0	.000	0	.000	0
		BOT	59.1	2	5.948	2	5.032	2
6	TOTL	6.563 TOP	-11.4	2	.000	0	.000	0
		BOT	48.9	3	4.063	3	4.063	3

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 9
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DISTRIBUTION OF DESIGN MOMENTS AT SUPPORTS

COL NUM	CROSS SECTN	TOTAL MOMENT (ft-k)	TOTAL-VERT DIFFERENCE (ft-k) (%)	COLUMN STRIP MOMENT (ft-k) (%)	BEAM MOMENT (ft-k) (%)	MIDDLE STRIP MOMENT (ft-k) (%)
1	LEFT TOP	-.4	.0 (0)	-.4 (98)	.0 (0)	.0 (2)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	31.3	.0 (0)	30.8 (98)	.0 (0)	.5 (2)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
2	LEFT TOP	-142.8	.0 (0)	-107.1 (75)	.0 (0)	-35.7 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	136.3	.0 (0)	102.2 (75)	.0 (0)	34.1 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
3	LEFT TOP	-116.1	.0 (0)	-87.1 (75)	.0 (0)	-29.0 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	117.5	.0 (0)	88.1 (75)	.0 (0)	29.4 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
4	LEFT TOP	-117.9	.0 (0)	-88.4 (75)	.0 (0)	-29.5 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	117.9	.0 (0)	88.4 (75)	.0 (0)	29.5 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
5	LEFT TOP	-98.1	.0 (0)	-73.6 (75)	.0 (0)	-24.5 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	90.8	.0 (0)	68.1 (75)	.0 (0)	22.7 (25)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
6	LEFT TOP	-57.3	.0 (0)	-56.1 (98)	.0 (0)	-1.1 (2)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	58.1	.0 (0)	56.9 (98)	.0 (0)	1.1 (2)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 10
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DISTRIBUTION OF DESIGN MOMENTS IN SPANS

SPAN NUM	CROSS SECTN	TOTAL MOMENT (ft-k)	TOTAL-VERT DIFFERENCE (ft-k) (%)	COLUMN STRIP MOMENT (ft-k) (%)	BEAM MOMENT (ft-k) (%)	MIDDLE STRIP MOMENT (ft-k) (%)
2	7.78 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	87.9	.0 (0)	52.8 (60)	.0 (0)	35.2 (40)
3	9.61 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	58.5	.0 (0)	35.1 (60)	.0 (0)	23.4 (40)
4	8.69 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	66.0	.0 (0)	39.6 (60)	.0 (0)	26.4 (40)
5	9.61 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	59.1	.0 (0)	35.5 (60)	.0 (0)	23.6 (40)
6	6.56 TOP	-11.4	.0 (0)	-6.9 (60)	.0 (0)	-4.6 (40)
	BOT	48.9	.0 (0)	29.3 (60)	.0 (0)	19.6 (40)

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 11
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

S H E A R A N A L Y S I S

NOTE--Allowable shear stress in slabs = 252.96 psi when ratio
 of col. dim. (long/short) is less than 2.0.

--Wide beam shear (see "CODE") is not computed, check manually.

--After the column numbers, C = Corner, E = Exterior, I = Interior.

D I R E C T		S H E A R		W I T H		T R A N S F E R		O F		M O M E N T
-		-		A R O U N D		C O L U M N		-		-
COL. NO.	ALLOW. STRESS (psi)	PATT NO.	REACTION (kips)	SHEAR STRESS (psi)	PATT NO.	REACTION (kips)	UNBAL. MOMENT (ft-k)	SHEAR TRANSFR (ft-k)	SHEAR STRESS (psi)	
1E	252.96	3	36.4	86.42	3	36.4	33.7	12.8	126.35	
2I	252.96	1	89.3	142.19	1	89.3	-8.7	-3.5	149.13	
3I	252.96	1	81.8	130.27	1	81.8	1.8	.7	131.70	
4I	252.96	1	79.8	126.97	1	79.8	-1.7	-.7	128.29	
5I	252.96	1	77.6	123.56	1	77.6	-3.9	-1.6	126.68	
6E	252.96	1	49.9	105.83	1	49.9	12.6	5.0	118.67	

- - AROUND DROP/SOLID HEAD - -

COLUMN NUMBER	ALLOW. STRESS (psi)	PATT NO.	REACTION (kips)	SHEAR STRESS (psi)
1E	195.22	3	30.2	32.47
2I	176.97	1	78.1	46.23
3I	176.97	1	70.6	41.80
4I	179.67	1	69.8	43.53
5I	184.13	1	68.3	46.15
6E	170.65	1	46.1	31.84

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 12
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

NEGATIVE REINFORCEMENT

COLUMN NUMBER	PATT NO.	LOCATION @COL FACE	TOTAL DESIGN * (ft-k)	COLUMN AREA * (sq.in)	STRIP WIDTH (ft)	MIDDLE AREA * (sq.in)	STRIP WIDTH (ft)
1	3		31.3	1.94	9.0	1.56	9.0
2	1	L	-142.8	2.65	9.0	1.56	9.0
3	1		117.5	2.17	9.0	1.56	9.0
4	1	R	117.9	2.18	8.0	1.38	8.0
5	1	L	-98.1	1.81	6.3	1.68	9.8
6	4	R	58.1	1.43	6.3	1.68	9.8

POSITIVE REINFORCEMENT

SPAN NUMBER	PATT NO.	LOCATION FROM LEFT (ft)	TOTAL DESIGN * (ft-k)	COLUMN AREA * (sq.in)	STRIP WIDTH (ft)	MIDDLE AREA * (sq.in)	STRIP WIDTH (ft)
2	3	7.8	87.9	1.71	9.0	1.56	9.0
3	2	9.6	58.5	1.56	9.0	1.56	9.0
4	3	8.7	66.0	1.56	9.0	1.56	9.0
5	2	9.6	59.1	1.38	8.0	1.38	8.0
6**	3	6.6	48.9	1.08	6.3	1.68	9.8

** - Negative reinforcement required, compute manually.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 13
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN RESULTS

NOTE--The schedule given below is a guide for proper reinforcement placement and is based on reasonable engineering judgement. Unusual boundary and/or loading conditions may require modification of this schedule.

NEGATIVE REINFORCEMENT

COLUMN NUMBER	COLUMN BARS				STRIP SHORT BARS				MIDDLE STRIP LONG BARS			
	* NO	SIZE	LEFT (ft)	RIGHT (ft)	* NO	SIZE	LEFT (ft)	RIGHT (ft)	* NO	SIZE	LEFT (ft)	RIGHT (ft)
1	5	# 4	.67	6.24	5	# 4	.67	4.01	8	# 4	.67	4.36
2	7	# 4	6.56	7.47	7	# 4	4.01	4.27	8	# 4	6.56	7.47
3	6	# 4	6.56	6.56	5	# 4	4.01	4.01	8	# 4	6.56	6.56
4	6	# 4	6.56	6.56	5	# 4	4.01	4.01	7	# 4	6.56	6.56
5	5	# 4	6.24	6.24	4	# 4	4.01	4.01	9	# 4	5.64	4.36
6	4	# 4	4.33	6.00	3	# 4	2.85	6.00	9	# 4	3.21	6.00

POSITIVE REINFORCEMENT

SPAN NUMBER	COLUMN LONG BARS				STRIP SHORT BARS				MIDDLE STRIP LONG BARS				STRIP SHORT BARS			
	* NO	SIZE	LENGTH (ft)	* NO	SIZE	LENGTH (ft)	* NO	SIZE	LENGTH (ft)	* NO	SIZE	LENGTH (ft)	* NO	SIZE	LENGTH (ft)	
2	5	# 4	16.16	4	# 4	16.16	4	# 4	18.46	4	# 4	15.46	4	# 4	15.46	
3	4	# 4	14.20	4	# 4	14.20	4	# 4	18.80	4	# 4	12.81	4	# 4	12.81	
4	4	# 4	14.20	4	# 4	14.20	4	# 4	18.80	4	# 4	12.81	4	# 4	12.81	
5	4	# 4	14.20	3	# 4	14.20	4	# 4	18.80	3	# 4	12.81	4	# 4	12.81	
6**	3	# 4	11.33	3	# 4	11.33	5	# 4	12.66	4	# 4	10.53	4	# 4	10.53	

** - Negative reinforcement required, design manually.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 14
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

ADDITIONAL INFORMATION AT SUPPORTS

* REINF. SUMMARY* ADD'L R/F REQ'D DUE TO UNBALANCED (U.) MOMENT TRANSFER

COLUMN NUMBER	* W/O U. MOMENT REQ'D (sq.in)	* MAX.U. PROV'D* (sq.in)	* GAMMA (ft-k)	* FLEXURAL -f (ft-k)	* PATT* TRANSFER	* CRITICAL NO. (ft)	* SLABW (sq.in)	SECTION - AREA	R/F
1	3.50	3.60	50.7	.62	31.5	3	3.9	.76	0 # 4
2	4.21	4.40	-24.7	.60	-14.8	3	3.9	.36	0 # 4
3	3.72	3.80	18.4	.60	11.0	3	3.9	.27	0 # 4
4	3.57	3.60	-17.4	.60	-10.4	3	3.9	.25	0 # 4
5	3.49	3.60	-22.6	.60	-13.6	2	3.9	.33	0 # 4
6	3.11	3.20	18.9	.60	11.3	2	3.9	.27	0 # 4

NOTE: Zero transfer "CRITICAL SLABW" indicates no support dimensions given for transfer.
 If beam(s) are present, transfer mode may be due to beam shear and/or torsion, check manually.

ADDITIONAL INFORMATION FOR
 IN - SPAN CONDITIONS

* REINF. SUMMARY *

SPAN NUMBER	* AT MIDSPAN REQ'D. - PROV'D. (sq.in)	* TOTAL FACTORED SPAN STATIC DESIGN MOMENT (W/O PARTIAL LOADS) (ft-k)
2	3.26	3.40
3	3.11	3.20
4	3.11	3.20
5	2.76	2.80
6	2.76	3.00

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 15
 2:53:10 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DEFLECTION ANALYSIS

NOTES--The deflections below must be combined with those of the analysis in the perpendicular direction. Consult users manual for method of combination and limitations.

--Spans 1 and 7 are cantilevers.

--Time-dependent deflections are in addition to those shown and must be computed as a multiplier of the dead load(DL) deflection. See "CODE" for range of multipliers.

--Deflections due to concentrated or partial loads may be larger at the point of application than those shown at the centerline. Deflections are computed as from an average uniform loading derived from the sum of all loads applied to the span.

--Modulus of elasticity of concrete, Ec = 3834. ksi

SPAN NUMBER	* C O L U M N S T R I P *				* M I D D L E S T R I P *				
	* DEAD LOAD	* DEFLECTION DUE TO:	* DEFLECTION DUE TO:			* DEAD	* LIVE	* TOTAL	* DEFLECTION DUE TO:
* Ieff. * (in^4)	* DEAD * (in)	* LIVE * (in)	* TOTAL * (in)	* DEAD * (in)	* LIVE * (in)	* TOTAL * (in)	* DEFLECTION DUE TO:	* (in)	* (in)
1	15186.	-.005	-.002	-.007	-.005	-.002	-.007		
2	12201.	.054	.024	.078	.034	.015	.049		
3	12201.	.024	.010	.035	.009	.003	.012		
4	12051.	.031	.015	.047	.016	.008	.024		
5	10845.	.035	.009	.044	.020	.002	.022		
6	10845.	.002	.016	.018	-.005	.008	.003		
7	13499.	.034	-.008	.026	.013	-.014	-.002		

Q U A N T I T Y E S T I M A T E S

TOTAL QUANTITIES

CONCRETE	41.1	cu.yd
FORMWORK	1624.	sq.ft
REINFORCEMENT (IN THE DIRECTION OF ANALYSIS)			
(NEGATIVE)	759.	lbs
(POSITIVE)	767.	lbs

SUMMARY OF QUANTITIES

CONCRETE70	cu.ft/sq.ft
FORMWORK	1.02	sq.ft/sq.ft
REINFORCEMENT**96	lbs / sq.ft

** (IN THE DIRECTION OF ANALYSIS)

Example 2, Beam-Supported Slab

Analyze and design an exterior strip shown in Figure C-2.

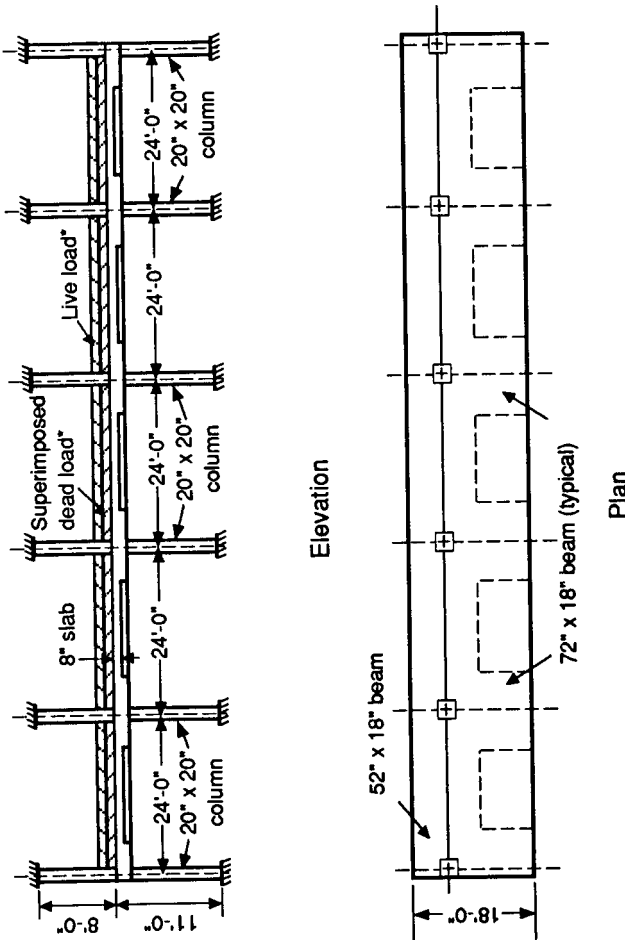


Figure C-2 Example beam-supported slab problem.

*For magnitude of loadings and drop panel dimensions, see computer output.

General Information

Number of spans:

Units

U.S. in-lb
 SI metric

Code

ACI 318-95
 CSA A23.3-94

Design Method

Strength Design
 Alternate Design
 Limit States

Floor System

Flat plate
 Flat slab
 Wall/slab
 Beam-supported slab
 Continuous beam
 One-way slab

Frame Location

Interior
 Exterior

Lateral Loads

None
 Wind
 Forces
 Moments

Loading Pattern

1 through 4
 4 through 8
 1 through 8

Proportion moments and shear

Material Properties

Concrete

	Slabs	Beams	Columns	
Density:	<input type="text" value="150"/>	<input type="text" value="150"/>	<input type="text" value="150"/>	pcf
Type:	Normal	Normal	Normal	
f'_c :	<input type="text" value="4"/>	<input type="text" value="4"/>	<input type="text" value="4"/>	kci
f_{ot} :	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	psi
f_y :	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	psi

Reinforcement

f_y : kci
 f_{yv} : kci
 Epoxy-coated

Slab Reinforcement Data

	Top	Bottom
Distance to reinforcement centroid from tension face:	<input type="text" value="3.6"/>	<input type="text" value="1.2"/> in
Minimum bar number:	<input type="text" value="4"/>	<input type="text" value="4"/>
Layer location:	<input checked="" type="checkbox"/> Outer <input type="checkbox"/> Inner	<input checked="" type="checkbox"/> Outer <input type="checkbox"/> Inner

Minimum clear bar spacing: in

Beam Reinforcement Data

	Top	Bottom
Distance to reinforcement centroid from tension face:	<input type="text" value="3.6"/>	<input type="text" value="1.2"/> in
Minimum bar number:	<input type="text" value="4"/>	<input type="text" value="4"/>
Maximum bar number:	<input type="text" value="14"/>	<input type="text" value="14"/>
Layer location:	<input checked="" type="radio"/> Outer <input type="radio"/> Inner	

Minimum clear bar spacing: in

Code minimum ties provided.

Slab Geometry

Span no.:

Length: ft

Depth: in

Width left: ft

Width right: ft

Replace

Copy from

Span no.	Length (ft)	Depth (in)	Width (ft)	
			Left	Right
1	0.8	8.0	2.2E	10.0
2	24.0	8.0	2.2E	10.0
3	24.0	8.0	2.2E	10.0
4	24.0	8.0	2.2E	10.0
5	24.0	8.0	2.2E	10.0
6	24.0	8.0	2.2E	10.0
7	0.8	8.0	2.2E	10.0

Ok Cancel

Longitudinal Beam Geometry

Span no.:

Width: in

Depth: in

Replace

Copy from

Span no.	Width (in)	Depth (in)
1	52.0	18.0
2	52.0	18.0
3	52.0	18.0
4	52.0	18.0
5	52.0	18.0
6	52.0	18.0
7	52.0	18.0

Ok Cancel

Column/Capital Geometry

Column no.:

Above Below

c_1 : in

c_2 : in

Height: ft

Capital Geometry

Extension: in

Depth: in

Column no.	Loc	c_1 (in)	c_2 (in)	Height (ft)	Ext. (in)	Depth (in)
1	Abv	20.0	20.0	8.0		
	Blw	20.0	20.0	11.0	0.0	0.0
2	Abv	20.0	20.0	8.0		
	Blw	20.0	20.0	11.0	0.0	0.0
3	Abv	20.0	20.0	8.0		
	Blw	20.0	20.0	11.0	0.0	0.0
4	Abv	20.0	20.0	8.0		
	Blw	20.0	20.0	11.0	0.0	0.0
5	Abv	20.0	20.0	8.0		
	Blw	20.0	20.0	11.0	0.0	0.0
6	Abv	20.0	20.0	8.0		
	Blw	20.0	20.0	11.0	0.0	0.0

Transverse Beam Geometry

Column no.:

Width: in

Depth: in

Eccentricity: in

Column no.	Width (in)	Depth (in)	Ecc. (in)
1	72.0	18.0	26.0
2	72.0	18.0	0.0
3	72.0	18.0	0.0
4	72.0	18.0	0.0
5	72.0	18.0	0.0
6	72.0	18.0	-26.0

Surface Loads

Span no.:

Dead: psf

Live: psf

Span no.	Loads (psf)	
	Dead	Live
1	10.0	100.0
2	10.0	100.0
3	10.0	100.0
4	10.0	100.0
5	10.0	100.0
6	10.0	100.0
7	10.0	100.0

Partial Live Loads

Span no.:

Uniform

Concentrated

Trapezoidal

Moment

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
6	1	UNIF	500.0		2.5	10.0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 1
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

```

pppppp   ccccc   aaaaa
p   p   c   c   a   a
p   p   c   c   a
p   p   c   aaaaa
p   p   c   c   a   a
p   p   c   c   a   a
pppppp   ccccc   aaaaa
p
p
    
```

```

AAA   DDDDD   OOO   SSSSS   SSSSS
A   A   D   D   O   O   S   S   S   S
A   A   D   D   O   O   S   S   S
AAAAAAA D   D   O   O   SSSSS   SSSSS
A   A   D   D   O   O   S   S   S   ( ttttt mm mm )
A   A   D   D   O   O   S   S   S   ( t m m m m )
A   A   DDDDD   OOO   SSSSS   SSSSS   ( t m m m )
    
```

```

*****
Computer program for ANALYSIS AND DESIGN OF SLAB SYSTEMS
*****
    
```

Licensee stated above acknowledges that Portland Cement Association(PCA) is not and cannot be responsible for either the accuracy or adequacy of the material supplied as input for processing by the ADOSS(tm) computer program. Furthermore, PCA neither makes any warranty expressed nor implied with respect to the correctness of the output prepared by the ADOSS(tm) program. Although PCA has endeavored to produce ADOSS(tm) error free the program is not and cannot be certified infallible. The final and only responsibility for analysis, design and engineering documents is the licensees. Accordingly, PCA disclaims all responsibility in contract, negligence or other tort for any analysis, design or engineering documents prepared in connection with the use of the ADOSS(tm) program.

Note: Further information and technical support on the ADOSS computer program is available through the CPCA Web site at: "www.cPCA.ca".

Example 2, Beam-Supported Slab

ADOSS Graphics

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 2
3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

FILE NAME C:\ADOSS\DATA\PLTBMS.ADS
PROJECT ID. FLAT PLATE WITH BEAMS

SPAN ID. Example 2

ENGINEER

DATE 01/01/99
TIME 12:00

UNITS U.S. in-lb
CODE ACI 318-95

SLAB SYSTEM BEAM-SUPPORTED SLAB
FRAME LOCATION EXTERIOR

DESIGN METHOD STRENGTH DESIGN
MOMENTS AND SHEARS NOT PROPORTIONED

NUMBER OF SPANS 7

CONCRETE FACTORS	SLABS	BEAMS	COLUMNS
DENSITY(pcf)	150.00	150.00	150.00
TYPE	NORMAL WGT	NORMAL WGT	NORMAL WGT
f'c (ksi)	4.00	4.00	4.00
density factor	1.00	1.00	1.00
fr (psi)	474.30	474.30	474.30

REINFORCEMENT DETAILS: NON-PRESTRESSED

YIELD STRENGTH (flexural) Fy = 60.00 ksi
YIELD STRENGTH (stirrups) Fyv = 60.00 ksi
DISTANCE TO RF CENTER FROM TENSION FACE:
AT SLAB TOP = 1.20 in OUTER LAYER
AT SLAB BOTTOM = 1.20 in OUTER LAYER
AT BEAM TOP = 1.20 in OUTER LAYER
AT BEAM BOTTOM = 1.20 in
FLEXURAL BAR SIZES: MINIMUM | MAXIMUM
AT SLAB TOP = # 4
AT SLAB BOTTOM = # 4
AT BEAM TOP = # 4 #14
IN BEAM BOTTOM = # 4 #14
MINIMUM SPACING:
IN SLAB = 6.00 in
IN BEAM = 1.00 in

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 3
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

SPAN/LOADING DATA

SPAN NUMBER	LENGTH Tslab		WIDTH L2***		SLAB SYSTEM	DESIGN STRIP (ft)	COLUMN STRIP** (ft)	UNIFORM LOADS	
	L1 (ft)	(in)	LEFT (ft)	RIGHT (ft)				S. DL (psf)	LIVE (psf)
1*	.8+	8.0	2.2E	10.0	4	12.2	.0	10.0	100.0
2	24.0	8.0	2.2E	10.0	4	12.2	2.8	10.0	100.0
3	24.0	8.0	2.2E	10.0	4	12.2	2.8	10.0	100.0
4	24.0	8.0	2.2E	10.0	4	12.2	2.8	10.0	100.0
5	24.0	8.0	2.2E	10.0	4	12.2	2.8	10.0	100.0
6	24.0	8.0	2.2E	10.0	4	12.2	2.8	10.0	100.0
7*	.8+	8.0	2.2E	10.0	4	12.2	.0	10.0	100.0

- * -Indicates cantilever span information.
- ** -Strip width used for positive flexure.
- ***-L2 widths are 1/2 dist. to transverse column.
- "E"-Indicates exterior strip.
- + -Indicates change in dimension due to support conditions.

PARTIAL LOADING DATA

SPAN No.	LOAD No.	TYPE	PARTIAL DEAD LOADS				LOAD No.	TYPE	PARTIAL LIVE LOADS			
			Wa	Wb	La	Lb			Wa	Wb	La	Lb
1*												
2												
3												
4												
5												
6						1	UNIF	500.0	.0	2.5	10.0	
7*												

* -Indicates cantilever span information.

UNITS FOR:

- UNIFORM LOAD: Wa.....plf La & Lb... ft
- CONCENTRATED LOAD: Wa.....kips La..... ft
- TRAPEZOIDAL LOAD: Wa & Wb..plf La & Lb... ft
- MOMENT: Wa.....ft-k La..... ft

NOTE: Local effects of partial loadings are NOT considered by ADOSS, compute manually.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 4
3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

BEAMS ALONG SPAN DATA

SPAN NUMBER	BEAM WIDTH (in)	BEAM DEPTHS			HAUNCH LENGTHS	
		LEFT (in)	CENTER (in)	RIGHT (in)	LEFT (ft)	RIGHT (ft)
1	52.0	18.0	18.0	18.0	.0	.0
2	52.0	18.0	18.0	18.0	.0	.0
3	52.0	18.0	18.0	18.0	.0	.0
4	52.0	18.0	18.0	18.0	.0	.0
5	52.0	18.0	18.0	18.0	.0	.0
6	52.0	18.0	18.0	18.0	.0	.0
7	52.0	18.0	18.0	18.0	.0	.0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 5
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

COLUMN/TORSIONAL DATA

COLUMN NUMBER	COLUMN ABOVE SLAB			COLUMN BELOW SLAB			CAPITAL**		COLUMN STRIP* (ft)	MIDDLE STRIP* (ft)
	C1 (in)	C2 (in)	HGT (ft)	C1 (in)	C2 (in)	HGT (ft)	EXTEN. (in)	DEPTH (in)		
1	20.0	20.0	8.0	20.0	20.0	11.0	.0	.0	2.8	5.0
2	20.0	20.0	8.0	20.0	20.0	11.0	.0	.0	2.8	5.0
3	20.0	20.0	8.0	20.0	20.0	11.0	.0	.0	2.8	5.0
4	20.0	20.0	8.0	20.0	20.0	11.0	.0	.0	2.8	5.0
5	20.0	20.0	8.0	20.0	20.0	11.0	.0	.0	2.8	5.0
6	20.0	20.0	8.0	20.0	20.0	11.0	.0	.0	2.8	5.0

Columns with zero "C2" are round columns.

* -Strip width used for negative flexure.

**-Capital extension distance measured from face of column.

COLUMN NUMBER	TRANSVERSE BEAM			DROP PANEL/SOLID HEAD				SUPPORT FIXITY* %
	WIDTH (in)	DEPTH (in)	ECCEN (in)	LEFT (ft)	RIGHT (ft)	WIDTH (ft)	THICK (in)	
1	72.0	18.0	26.0	.0	.0	.0	.0	100%
2	72.0	18.0	.0	.0	.0	.0	.0	100%
3	72.0	18.0	.0	.0	.0	.0	.0	100%
4	72.0	18.0	.0	.0	.0	.0	.0	100%
5	72.0	18.0	.0	.0	.0	.0	.0	100%
6	72.0	18.0	-26.0	.0	.0	.0	.0	100%

* -Support fixity of 0% denotes pinned condition.

Support fixity of 999% denotes fixed end condition.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 6
3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

LATERAL LOAD/OUTPUT DATA

LATERAL LOADS ARE NOT SPECIFIED

OUTPUT DATA
PATTERN LOADINGS: 1 THRU 4
PATTERN LIVE LOAD FACTOR (1-3) = 75%

LOAD FACTORS:
U = 1.40*D + 1.70*L
U = .75(1.40*D + 1.70*L + 1.70*W)
U = .90*D + 1.30*W

OUTPUT OPTION(S):
Input Echo
Centerline Moments and Shears
Column Strip Distribution Fac
Shear Table
Reinforcing Required
Bar Sizing
Additional Information
Deflections
Material Quantities

**TOTAL UNFACTORED DEAD LOAD = 274.317 kips
LIVE LOAD = 151.778 kips

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 7
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

---- STATICS PRINT-OUT FOR GRAVITY LOAD ANALYSIS ----

J O I N T M O M E N T S (ft - k i p s)

JOINT NUMBER	PATTERN-1				PATTERN-2			
	LEFT	RIGHT	TOP	BOTTOM	LEFT	RIGHT	TOP	BOTTOM
1	-1.7	169.2	-85.0	-82.4	-1.7	95.1	-47.4	-46.0
2	-279.5	258.3	10.7	10.4	-204.2	233.6	-14.9	-14.5
3	-242.6	246.2	-1.8	-1.8	-213.3	162.6	25.7	24.9
4	-246.2	242.6	1.8	1.8	-162.6	213.3	-25.7	-24.9
5	-263.3	293.7	-15.4	-15.0	-233.6	204.2	14.9	14.5
6	-175.0	1.7	88.0	85.3	-95.1	1.7	47.4	46.0

JOINT NUMBER	PATTERN-3				PATTERN-4			
	LEFT	RIGHT	TOP	BOTTOM	LEFT	RIGHT	TOP	BOTTOM
1	-1.2	170.8	-86.1	-83.5	-1.9	177.8	-89.3	-86.6
2	-250.6	174.6	38.6	37.4	-308.1	277.3	15.6	15.2
3	-159.1	219.0	-30.4	-29.5	-253.9	260.1	-3.2	-3.1
4	-217.8	156.7	31.0	30.1	-258.5	250.6	4.0	3.9
5	-179.6	264.9	-43.3	-42.0	-284.1	327.2	-21.9	-21.2
6	-176.7	1.2	89.1	86.4	-185.6	1.9	93.2	90.4

J O I N T S H E A R S (k i p s)

JOINT NUMBER	PATTERN-1		PATTERN-2		PATTERN-3		PATTERN-4	
	LEFT	RIGHT	LEFT	RIGHT	LEFT	RIGHT	LEFT	RIGHT
1	-4.2	55.8	-4.2	36.2	-2.9	56.0	-4.6	60.1
2	-61.4	57.7	-42.2	56.4	-59.6	37.5	-67.9	62.7
3	-56.5	57.1	-54.7	36.9	-36.3	55.6	-60.7	61.8
4	-57.1	56.5	-36.9	54.7	-55.5	35.9	-61.7	60.3
5	-58.0	65.2	-56.4	42.2	-37.9	63.5	-63.1	73.1
6	-56.7	4.2	-36.2	4.2	-56.9	2.9	-61.3	4.6

Example 2, Beam-Supported Slab

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 8
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN MOMENT ENVELOPES AT CRITICAL SECTIONS FROM SUPPORTS

COL NUM	LOAD TYPE	CROSS SECTN	DESIGN MOMENT (ft-k)	DISTANCE CR. SECTN (ft)	LOAD PTRN	MAX. I. P. DISTANCE (ft)	LOAD PTRN
1	TOTL LEFT	TOP	-1.3	.146	4	.833	1
		BOT	.0	.000	0	.000	0
	RGHT	TOP	129.7	.833	4	3.600	2
		BOT	.0	.000	0	.000	0
2	TOTL LEFT	TOP	-253.6	.833	4	7.200	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	227.1	.833	4	7.200	3
		BOT	.0	.000	0	.000	0
3	TOTL LEFT	TOP	-205.3	.833	4	7.200	3
		BOT	.0	.000	0	.000	0
	RGHT	TOP	210.7	.833	4	7.200	2
		BOT	.0	.000	0	.000	0
4	TOTL LEFT	TOP	-209.2	.833	4	7.200	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	202.3	.833	4	7.200	3
		BOT	.0	.000	0	.000	0
5	TOTL LEFT	TOP	-233.5	.833	4	7.200	3
		BOT	.0	.000	0	.000	0
	RGHT	TOP	268.3	.833	4	7.200	2
		BOT	.0	.000	0	.000	0
6	TOTL LEFT	TOP	-136.5	.833	4	3.600	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	1.3	.146	4	.833	1
		BOT	.0	.000	0	.000	0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 9
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN MOMENT ENVELOPES AT CRITICAL SECTIONS ALONG SPANS

SPAN NUM	LOAD TYPE	CRITICAL SECTION (ft)	DESIGN MOMENT (ft-k)	LOAD PTRN	MAX. I.P. DIST LEFT (ft)	LOAD PTRN	MAX. I.P. DIST RGHT (ft)	LOAD PTRN
2	TOTL	10.200 TOP	.0	0	.000	0	.000	0
		BOT	118.6	4	6.600	1	7.800	1
3	TOTL	12.600 TOP	.0	0	.000	0	.000	0
		BOT	87.6	2	6.600	1	6.600	2
4	TOTL	12.600 TOP	.0	0	.000	0	.000	0
		BOT	92.2	3	7.800	3	6.600	3
5	TOTL	11.400 TOP	.0	0	.000	0	.000	0
		BOT	87.6	2	6.600	2	6.600	1
6	TOTL	12.600 TOP	.0	0	.000	0	.000	0
		BOT	123.7	4	6.600	1	7.800	1

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 10
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DISTRIBUTION OF DESIGN MOMENTS AT SUPPORTS

COL NUM	CROSS SECTN	TOTAL MOMENT (ft-k)	TOTAL-VERT DIFFERENCE (ft-k) (%)	COLUMN STRIP MOMENT (ft-k) (%)	BEAM MOMENT (ft-k) (%)	MIDDLE STRIP MOMENT (ft-k) (%)
1	LEFT TOP	-1.3	.0 (0)	-.2 (13)	-1.0 (76)	-.1 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	129.7	.0 (0)	17.5 (13)	99.0 (76)	13.2 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
2	LEFT TOP	-253.6	.0 (0)	-34.2 (13)	-193.5 (76)	-25.9 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	227.1	.0 (0)	30.6 (13)	173.4 (76)	23.2 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
3	LEFT TOP	-205.3	.0 (0)	-27.7 (13)	-156.7 (76)	-21.0 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	210.7	.0 (0)	28.4 (13)	160.8 (76)	21.5 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
4	LEFT TOP	-209.2	.0 (0)	-28.2 (13)	-159.6 (76)	-21.4 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	202.3	.0 (0)	27.3 (13)	154.4 (76)	20.7 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
5	LEFT TOP	-233.5	.0 (0)	-31.5 (13)	-178.2 (76)	-23.8 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	268.3	.0 (0)	36.1 (13)	204.8 (76)	27.4 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
6	LEFT TOP	-136.5	.0 (0)	-18.4 (13)	-104.2 (76)	-13.9 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	1.3	.0 (0)	.2 (13)	1.0 (76)	.1 (10)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 11
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DISTRIBUTION OF DESIGN MOMENTS IN SPANS

SPAN NUM	CROSS SECTN	TOTAL MOMENT (ft-k)	TOTAL-VERT DIFFERENCE (ft-k) (%)	COLUMN STRIP MOMENT (ft-k) (%)	BEAM MOMENT (ft-k) (%)	MIDDLE STRIP MOMENT (ft-k) (%)
2	10.20 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	118.6	.0 (0)	16.0 (13)	90.5 (76)	12.1 (10)
3	12.60 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	87.6	.0 (0)	11.8 (13)	66.9 (76)	8.9 (10)
4	12.60 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	92.2	.0 (0)	12.4 (13)	70.3 (76)	9.4 (10)
5	11.40 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	87.6	.0 (0)	11.8 (13)	66.9 (76)	8.9 (10)
6	12.60 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	123.7	.0 (0)	16.7 (13)	94.4 (76)	12.6 (10)

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 12
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

S H E A R A N A L Y S I S

NOTE--Allowable shear stress in slabs = 252.96 psi when ratio
 of col. dim. (long/short) is less than 2.0.

--Wide beam shear (see "CODE") is not computed, check manually.

--After the column numbers, C = Corner, E = Exterior, I = Interior.

D I R E C T		S H E A R			W I T H		T R A N S F E R		O F		M O M E N T
-		-			A R O U N D		C O L U M N		-		-
COL. NO.	ALLOW. STRESS (psi)	PATT NO.	REACTION (kips)	SHEAR STRESS (psi)	PATT NO.	REACTION (kips)	UNBAL. MOMENT (ft-k)	SHEAR TRANSFR (ft-k)	SHEAR STRESS (psi)		
1C	252.96	4	62.9	87.71	4	62.9	133.0	51.0	149.53		
2E	252.96	4	128.3	111.74	4	128.3	-30.8	-12.3	122.61		
3E	252.96	4	120.2	104.69	4	120.2	6.2	2.5	106.88		
4E	252.96	4	119.6	104.20	4	119.6	-7.9	-3.2	107.00		
5E	252.96	4	133.9	116.62	4	133.9	43.1	17.2	131.81		
6C	252.96	4	64.1	89.38	4	64.1	-140.0	-53.7	154.47		

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 13
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

B E A M S H E A R R E Q U I R E M E N T S (kips, sq.in./in., ft.)

NOTE--Allowable shear stress in beams = 126.48 psi (see "CODE").

BEAM SPAN NO.	PATT. NO.	LEFT Vu@d SHEAR	SIDE Av/s @d	--FRACTIONAL DIST. ALONG SPAN--				RIGHT Av/s @d	SIDE Vu@d SHEAR	LEFT Vc/2. DIST.	
1	*	*	Span length equal to column size or zero							*	*
2	4	45.9	.000	.000	.000	.000	.000	.043*	-53.8	2.23	
3	4	48.5	.043*	.000	.000	.000	.000	.000	-46.6	3.00	
4	4	47.6	.043*	.000	.000	.000	.000	.043*	-47.5	3.00	
5	4	46.1	.000	.000	.000	.000	.000	.043*	-48.9	2.23	
6	4	58.9	.043*	.000	.000	.000	.000	.043*	-47.1	4.20	
7	*	*	Span length equal to column size or zero							*	*

- NOTES: 1.) To obtain stirrup spacing, divide stirrup area by Av/s value above.
 2.) To obtain stirrup area, multiply spacing by Av/s value.
 3.) Local effects due to loadings applied at other segments along beam span must be calculated manually.
 4.) Symbols following Av/s values:
 * - minimum shear $50 \cdot bw / Fyv$ - based on beam dimensions.
 x - V_s exceeds $2 \cdot V_c$, maximum stirrup spacing must be halved.
 + - Av/s value at segment located within effective depth.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 14
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

T R A N S V E R S E B E A M S H E A R A N D T O R S I O N
 R E Q U I R E M E N T S (kips, ft-k, SQ.in, /,in.)

----- LEFT SIDE -----										
BEAM No.	PATT. NO.	Vu@d SHEAR	Vc@d SHEAR	Tu@d TORSION	Tc@d TORSION	Av/s @d	At/s @d	Atot/s @d	Al @d	Smax
1	**			Exterior edge, no shear or torsion transferred						**
2	**			Exterior edge, no shear or torsion transferred						**
3	**			Exterior edge, no shear or torsion transferred						**
4	**			Exterior edge, no shear or torsion transferred						**
5	**			Exterior edge, no shear or torsion transferred						**
6	**			Exterior edge, no shear or torsion transferred						**

----- RIGHT SIDE -----										
BEAM No.	PATT. NO.	Vu@d SHEAR	Vc@d SHEAR	Tu@d TORSION	Tc@d TORSION	Av/s @d	At/s @d	Atot/s @d	Al @d	Smax
1	4	24.6	153.0	136.6	N/A	.000	.030*	.060*	5.	.0
2	3	35.5	153.0	59.1	N/A	.000	.030*	.060*	5.	.0
3	3	33.2	153.0	46.5	N/A	.000	.030*	.060*	5.	.0
4	3	33.2	153.0	47.5	N/A	.000	.030*	.060*	5.	.0
5	3	35.5	153.0	66.2	N/A	.000	.030*	.060*	5.	.0
6	4	26.6	153.0	142.6	N/A	.000	.030*	.060*	5.	.0

- NOTES: 1.) Deep beam analysis not considered.
 2.) Loads assumed applied from above beam.
 3.) Moment and shear at concentrated load must be checked manually if located along transverse beam.
 4.) Symbols following Av/s values:
 * - Code required minimum shear reinforcement.
 x - Vs exceeds 4*Vc, increase member section.
 5.) Symbols following At/s values:
 * - Code required minimum torsion.
 x - Section check requires that the member section be increased.
 6.) Symbols following Atot/s values:
 * - Code required minimum torsion reinforcement.
 # - Total stirrups exceed allowable.
 = - Stirrup spacing exceeds allowable.
 7.) Redistribution of torque is not considered.
 8.) Detail first stirrup @ 3 inches.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 15
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

N E G A T I V E R E I N F O R C E M E N T

COLUMN* NUMBER	PATT* NO.	LOCATION* @COL FACE	TOTAL DESIGN (ft-k)	* COLUMN AREA (sq.in)	STRIP WIDTH (ft)	* MIDDLE AREA (sq.in)	STRIP WIDTH (ft)	* COL. AREA (sq.in)	STRIP. WIDTH (in)
1	4	R	129.7	.58	2.8	.86	5.0	1.77	52.0
2	4	L	-253.6	1.17	2.8	.86	5.0	2.91	52.0
3	4	R	210.7	.96	2.8	.86	5.0	2.90	52.0
4	4	L	-209.2	.96	2.8	.86	5.0	2.88	52.0
5	4	R	268.3	1.24	2.8	.91	5.0	2.91	52.0
6	4	L	-136.5	.62	2.8	.86	5.0	1.86	52.0

P O S I T I V E R E I N F O R C E M E N T

SPAN NUMBER	*PATT* NO.	LOCATION* FROM LEFT (ft)	TOTAL DESIGN (ft-k)	* COLUMN AREA (sq.in)	STRIP WIDTH (ft)	* MIDDLE AREA (sq.in)	STRIP WIDTH (ft)	* COL. AREA (sq.in)	STRIP. WIDTH (in)
2	4	10.2	118.6	.53	2.8	.86	5.0	1.62	52.0
3	2	12.6	87.6	.49	2.8	.86	5.0	1.19	52.0
4	3	12.6	92.2	.49	2.8	.86	5.0	1.25	52.0
5	2	11.4	87.6	.49	2.8	.86	5.0	1.19	52.0
6	4	12.6	123.7	.56	2.8	.86	5.0	1.69	52.0

NOTE: Beam reinforcement areas are based on one layer.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 16
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN RESULTS

NOTE--The schedule given below is a guide for proper reinforcement placement and is based on reasonable engineering judgement. Unusual boundary and/or loading conditions may require modification of this schedule.

--The beam reinforcement detailing such as hooks and bar lengths, should be done manually to ensure proper bar detailing.

NEGATIVE REINFORCEMENT

COLUMN NUMBER	LONG BARS		C O L U M N		SHORT BARS		S T R I P		LONG BARS		M I D D L E S T R I P	
	* NO	SIZE	LEFT (ft)	RIGHT (ft)	* NO	SIZE	LEFT (ft)	RIGHT (ft)	* NO	SIZE	LEFT (ft)	RIGHT (ft)
1	2	# 4	.83	7.53	1	# 4	.83	5.30	5	# 4	.83	5.75
2	2	# 5	8.60	8.60	2	# 5	5.30	5.30	5	# 4	8.60	8.60
3	3	# 4	8.60	8.60	2	# 4	5.30	5.30	5	# 4	8.60	8.60
4	3	# 4	8.60	8.60	2	# 4	5.30	5.30	5	# 4	8.60	8.60
5	2	# 5	8.60	8.60	2	# 5	5.30	5.30	5	# 4	8.60	8.60
6	2	# 4	7.53	.83	1	# 4	5.30	.83	5	# 4	5.75	.83

* B E A M *

COLUMN NUMBER	* NO	SIZE	LAYERS
1	9	# 4	1
2	15	# 4	1
3	15	# 4	1
4	15	# 4	1
5	15	# 4	1
6	10	# 4	1

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 17
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN RESULTS

NOTE--The schedule given below is a guide for proper reinforcement placement and is based on reasonable engineering judgement. Unusual boundary and/or loading conditions may require modification of this schedule.

- Bars are distributed uniformly across column and middle strips.
- The beam reinforcement detailing such as hooks and bar lengths, should be done manually to ensure proper bar detailing.
- At least 2 bottom bars must be continuous or anchored within the column strip as required.

POSITIVE REINFORCEMENT

SPAN NUMBER	C O L U M N			S T R I P			M I D D L E			S T R I P		
	LONG BARS	SHORT BARS	LENGTH (ft)	LONG BARS	SHORT BARS	LENGTH (ft)	LONG BARS	SHORT BARS	LENGTH (ft)	LONG BARS	SHORT BARS	LENGTH (ft)
2	2	# 4	23.42	1	# 4	20.67	3	# 4	23.92	2	# 4	20.07
3	2	# 4	23.50	1	# 4	18.00	3	# 4	24.50	2	# 4	16.80
4	2	# 4	23.50	1	# 4	18.00	3	# 4	24.50	2	# 4	16.80
5	2	# 4	23.50	1	# 4	18.00	3	# 4	24.50	2	# 4	16.80
6	2	# 4	23.42	1	# 4	20.67	3	# 4	23.92	2	# 4	20.07

SPAN NUMBER	B E A M		
	NO	SIZE	LAYERS
2	8	# 4	1
3	6	# 4	1
4	7	# 4	1
5	6	# 4	1
6	9	# 4	1

Example 2, Beam-Supported Slab

ADOSS Graphics

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 18
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

 ADDITIONAL INFORMATION AT SUPPORTS

COLUMN NUMBER	* REINF. SUMMARY * * W/O U. MOMENT * * REQ'D - PROV'D * * (sq.in) (sq.in) *	ADD'L R/F	REQ'D DUE TO UNBALANCED (U.) MOMENT TRANSFER	* MAX.U. * * -f * * (ft-k) *	* GAMMA * * * * *	FLEXURAL * * TRANSFER * * (ft-k) *	* PATT * * NO. * * (ft)	CRITICAL * * SLABW - AREA - * (sq.in)	SECTION * * R/F * * #
1	3.22 3.40	175.8	.62	108.3	4	2.7	4.29	17	# 4
2	4.94 5.24	-76.0	.60	-45.6	3	2.7	1.59	0	# 5
3	4.73 5.00	59.9	.60	35.9	3	2.7	1.24	0	# 4
4	4.70 5.00	-61.1	.60	-36.7	3	2.7	1.26	0	# 4
5	5.06 5.24	85.2	.60	51.1	3	2.7	1.80	1	# 5
6	3.34 3.60	-183.7	.62	-113.2	4	2.7	4.53	17	# 4

NOTE: Zero transfer "CRITICAL SLABW" indicates no support dimensions given for transfer.
 If beam(s) are present, transfer mode may be due to beam shear and/or torsion, check manually.

 ADDITIONAL INFORMATION FOR
 IN - SPAN CONDITIONS

SPAN NUMBER	* REINF. SUMMARY * * AT MIDSPAN * * REQ'D. - PROV'D. * * (sq.in) (sq.in) *	* TOTAL FACTORED SPAN * * STATIC DESIGN MOMENT * * (W/O PARTIAL LOADS) * * (ft-k) *
2	3.01 3.20	293.1
3	2.54 2.80	293.1
4	2.61 3.00	293.1
5	2.54 2.80	293.1
6	3.11 3.40	293.1

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 19
 3:43:26 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DEFLECTION ANALYSIS

NOTES--The deflections below must be combined with those of the analysis in the perpendicular direction. Consult users manual for method of combination and limitations.

--Spans 1 and 7 are cantilevers.

--Time-dependent deflections are in addition to those shown and must be computed as a multiplier of the dead load(DL) deflection. See "CODE" for range of multipliers.

--Deflections due to concentrated or partial loads may be larger at the point of application than those shown at the centerline. Deflections are computed as from an average uniform loading derived from the sum of all loads applied to the span.

--Modulus of elasticity of concrete, Ec = 3834. ksi

SPAN NUMBER	DEAD LOAD	COLUMN STRIP DEFLECTION DUE TO:			MIDDLE STRIP DEFLECTION DUE TO:		
		I _{eff} (in ⁴)	DEAD (in)	LIVE (in)	TOTAL (in)	DEAD (in)	LIVE (in)
1	39707.	-.002	-.001	-.003	-.002	-.001	-.003
2	39707.	.035	.018	.054	.017	.009	.026
3	39707.	.024	.014	.038	.007	.004	.011
4	39707.	.026	.015	.041	.009	.005	.014
5	39707.	.024	.013	.037	.007	.003	.011
6	39707.	.035	.021	.056	.017	.011	.028
7	39707.	-.002	-.001	-.003	-.002	-.001	-.003

QUANTITY ESTIMATES

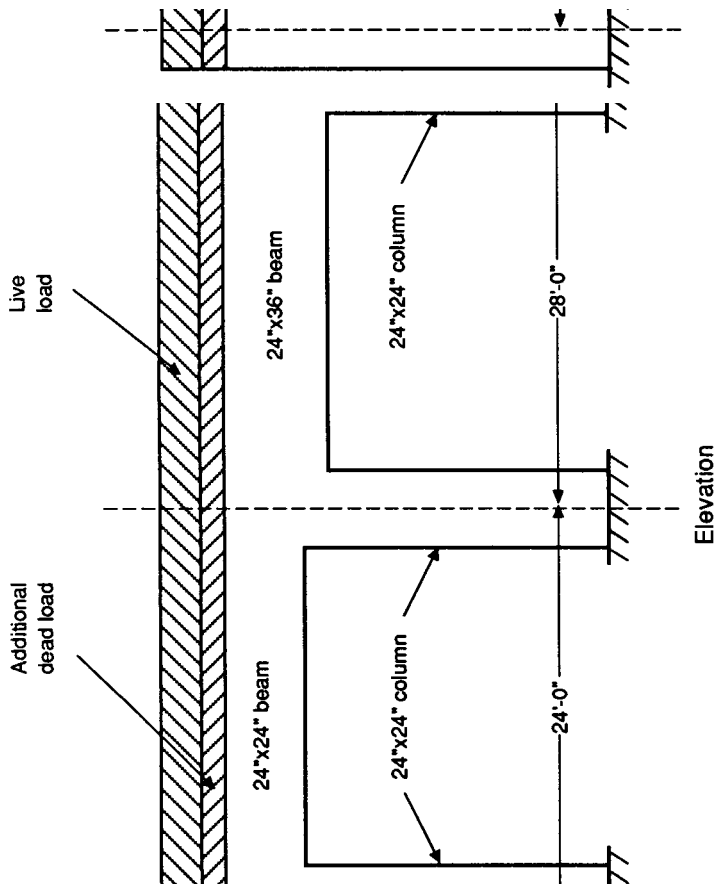
TOTAL QUANTITIES

 CONCRETE 52.8 cu.yd
 FORMWORK 2210. sq.ft
 REINFORCEMENT (COMPUTE MANUALLY)

* Program completed as requested *

Example 3, Continuous Beam

Analyze and design a continuous beam shown in Figure C-3.



*For magnitude of loadings and drop panel dimensions, see computer output.

Figure C-3 Example continuous beam problem

General Information

Number of spans:

Units:
 U.S. in-lb
 SI metric

Code:
 ACI 318-95
 CSA A23.3-94

Design Method:
 Strength Design
 Alternate Design
 Limit States

Floor System:
 Flat plate
 Flat slab
 Waffle slab
 Beam-supported slab
 Continuous beam
 One-way slab

Frame Location:
 Interior
 Exterior

Lateral Loads:
 None
 Wind
 Forces
 Moments

Loading Pattern:
 1 through 4
 4 through 8
 1 through 8

Proportion moments and shears

Material Properties

Concrete

	Slabs	Beams	Columns	
Density:	<input type="text" value="150"/>	<input type="text" value="150"/>	<input type="text" value="150"/>	pcf
Type:	Normal	Normal	Normal	
f'_c :	<input type="text" value="4"/>	<input type="text" value="4"/>	<input type="text" value="4"/>	ksi
f_{ct} :	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	<input type="text" value="423.7"/>	psi
f_r :	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	<input type="text" value="474.3"/>	psi

Reinforcement:
 f_y : ksi
 f_{yv} : ksi
 Epoxy-coated

Beam Reinforcement Data

	Top	Bottom
Distance to reinforcement centroid from tension face:	1.75 in	1.75 in
Minimum bar number:	4	4
Maximum bar number:	14	14
Layer location:	<input checked="" type="checkbox"/> Outer <input type="checkbox"/> Inner	

Minimum clear bar spacing: 1 in

Code minimum ties provided.

Ok Cancel

Longitudinal Beam Geometry

Span no.: 1

Length: 1 ft

Width: 24 in

Depth: Left 24 in, Center 0 in, Right 0 in

Haunch Length: 0 ft

Replace Copy from

Span no.	Length (ft)	Width (in)	Depth (in)			Haunch (ft)	
			Left	Center	Right	Left	Right
1	1.0	24.0	24.0	0.0	0.0	0.0	0.0
2	24.0	24.0	24.0	0.0	0.0	0.0	0.0
3	28.0	24.0	36.0	0.0	0.0	0.0	0.0
4	1.0	24.0	36.0	0.0	0.0	0.0	0.0

Ok Cancel

Column/Capital Geometry

Column no.:

Above Below

c_1 : in

c_2 : in

Height: ft

Capital Geometry

Extension: in

Depth: in

Column no	loc	c_1 (in)	c_2 (in)	Height (ft)	Ext. (in)	Depth (in)
1	Abr	0.0	0.0	0.0		
	Blw	24.0	24.0	8.0	0.0	0.0
2	Abr	0.0	0.0	0.0		
	Blw	24.0	24.0	8.0	0.0	0.0
3	Abr	0.0	0.0	0.0		
	Blw	24.0	24.0	8.0	0.0	0.0

Partial Dead Loads

Span no.:

Uniform

Concentrated

Trapezoidal

Moment

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
1	1	UNIF	2000.0	0.0	1.0	
2	1	UNIF	2000.0	0.0	0.0	24.0
3	1	UNIF	2000.0	0.0	0.0	28.0
4	1	UNIF	2000.0	0.0	0.0	1.0

Partial Live Loads

Span no.:

Uniform
 Concentrated
 Trapezoidal
 Moment

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
1	1	UNIF	1200.0	0.0	0.0	1.0
2	1	UNIF	1200.0	0.0	0.0	24.0
3	1	UNIF	1200.0	0.0	0.0	28.0
4	1	UNIF	1200.0	0.0	0.0	1.0

Drop Geometry

Standard drop

Column no.:
 Length left: ft
 Length right: ft
 Width: ft
 Thickness: in

Column no.	Length (ft) Left	Length (ft) Right	Width (ft)	Depth (in)
1	0.7	3.0	6.0	3.0
2	3.0	3.0	6.0	3.0
3	3.0	3.0	6.0	3.0
4	3.0	3.0	5.3	3.0
5	3.0	2.1	5.3	3.0
6	2.1	6.0	5.3	3.0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 1
 3:18:00 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

```

pppppp   ccccc   aaaaa
p   p   c   c   a   a
p   p   c   c   a
p   p   c   aaaaa
p   p   c   c   a   a
p   p   c   c   a   a
pppppp   ccccc   aaaaa
p
p
    
```

```

AAA      DDDDD      OOO      SSSSS      SSSSS
A   A   D   D   O   O   S   S   S   S
A   A   D   D   O   O   S           S
AAAAAAA D   D   O   O   SSSSS      SSSSS
A   A   D   D   O   O           S           S   ( ttttt mm   mm )
A   A   D   D   O   O   S   S   S   S   S   (   t   m   m   m )
A   A   DDDDD      OOO      SSSSS      SSSSS   (   t   m   m   m )
    
```

```

*****
Computer program for ANALYSIS AND DESIGN OF SLAB SYSTEMS
*****
    
```

Licensee stated above acknowledges that Portland Cement Association(PCA) is not and cannot be responsible for either the accuracy or adequacy of the material supplied as input for processing by the ADOSS(tm) computer program. Furthermore, PCA neither makes any warranty expressed nor implied with respect to the correctness of the output prepared by the ADOSS(tm) program. Although PCA has endeavored to produce ADOSS(tm) error free the program is not and cannot be certified infallible. The final and only responsibility for analysis, design and engineering documents is the licensees. Accordingly, PCA disclaims all responsibility in contract, negligence or other tort for any analysis, design or engineering documents prepared in connection with the use of the ADOSS(tm) program.

Note: Further information and technical support on the ADOSS computer program is available through the CPCA Web site at: "www.cPCA.ca".

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 2
 3:18:00 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

FILE NAME C:\ADOSS\DATA\CONTBEAM.ADS
 PROJECT ID. CONTINUOUS BEAM -VERTICAL LOADS
 SPAN ID. Example 3

ENGINEER

DATE 01/01/99
 TIME 12:00

UNITS U.S. in-lb
 CODE ACI 318-95

SLAB SYSTEM CONTINUOUS BEAM
 FRAME LOCATION INTERIOR

DESIGN METHOD STRENGTH DESIGN

NUMBER OF SPANS 4

CONCRETE FACTORS	SLABS	BEAMS	COLUMNS
DENSITY(pcf)	150.00	150.00	150.00
TYPE	NORMAL WGT	NORMAL WGT	NORMAL WGT
f'c (ksi)	4.00	4.00	4.00
density factor	1.00	1.00	1.00
fr (psi)	474.30	474.30	474.30

REINFORCEMENT DETAILS: NON-PRESTRESSED

YIELD STRENGTH (flexural) $F_y = 60.00$ ksi

YIELD STRENGTH (stirrups) $F_{yv} = 60.00$ ksi

DISTANCE TO RF CENTER FROM TENSION FACE:

AT BEAM TOP = 1.75 in OUTER LAYER

AT BEAM BOTTOM = 1.75 in

FLEXURAL BAR SIZES: MINIMUM | MAXIMUM

AT BEAM TOP = # 4 #14

AT BEAM BOTTOM = # 4 #14

MINIMUM SPACING:

IN BEAM = 1.00 in

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 3
 3:18:00 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

SPAN/LOADING DATA

SPAN NUMBER	Tslab		WIDTH		SLAB SYSTEM	DESIGN STRIP (ft)	COLUMN STRIP** (ft)	UNIFORM LOADS	
	L1 (ft)	(in)	LEFT (ft)	RIGHT (ft)				S. DL (psf)	LIVE (psf)
1*	1.0	.0	1.0	1.0	5	2.0	2.0	.0	.0
2	24.0	.0	1.0	1.0	5	2.0	2.0	.0	.0
3	28.0	.0	1.0	1.0	5	2.0	2.0	.0	.0
4*	1.0	.0	1.0	1.0	5	2.0	2.0	.0	.0

* -Indicates cantilever span information.
 ** -Strip width used for positive flexure.
 ***-L2 widths are 1/2 dist. to transverse column.
 "E"-Indicates exterior strip.

PARTIAL LOADING DATA

SPAN No.	LOAD No.	TYPE	PARTIAL DEAD LOADS				LOAD No.	TYPE	PARTIAL LIVE LOADS			
			Wa	Wb	La	Lb			Wa	Wb	La	Lb
1*	1	UNIF	2800.0	.0	.0	1.0	1	UNIF	1200.0	.0	.0	1.0
2	1	UNIF	2800.0	.0	.0	24.0	1	UNIF	1200.0	.0	.0	24.0
3	1	UNIF	2800.0	.0	.0	28.0	1	UNIF	1200.0	.0	.0	28.0
4*	1	UNIF	2800.0	.0	.0	1.0	1	UNIF	1200.0	.0	.0	1.0

* -Indicates cantilever span information.

UNITS FOR:

UNIFORM LOAD: Wa.....plf La & Lb... ft
 CONCENTRATED LOAD: Wa.....kips La..... ft
 TRAPEZOIDAL LOAD: Wa & Wb..plf La & Lb... ft
 MOMENT: Wa.....ft-k La..... ft

NOTE: Local effects of partial loadings are NOT considered by ADOSS, compute manually.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 4
3:18:00 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

BEAMS ALONG SPAN DATA

SPAN NUMBER	BEAM WIDTH (in)	BEAM DEPTHS			HAUNCH LENGTHS	
		LEFT (in)	CENTER (in)	RIGHT (in)	LEFT (ft)	RIGHT (ft)
1	24.0	24.0	24.0	24.0	.0	.0
2	24.0	24.0	24.0	24.0	.0	.0
3	24.0	36.0	36.0	36.0	.0	.0
4	24.0	36.0	36.0	36.0	.0	.0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 5
 3:18:00 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

COLUMN/TORSIONAL DATA

COLUMN NUMBER	COLUMN ABOVE SLAB			COLUMN BELOW SLAB			CAPITAL**		COLUMN STRIP* (ft)	MIDDLE STRIP* (ft)
	C1 (in)	C2 (in)	HGT (ft)	C1 (in)	C2 (in)	HGT (ft)	EXTEN. (in)	DEPTH (in)		
1	.0	.0	.0	24.0	24.0	8.0	.0	.0	2.0	.0
2	.0	.0	.0	24.0	24.0	8.0	.0	.0	2.0	.0
3	.0	.0	.0	24.0	24.0	8.0	.0	.0	2.0	.0

Columns with zero "C2" are round columns.
 * -Strip width used for negative flexure.
 **-Capital extension distance measured from face of column.

COLUMN NUMBER	SUPPORT FIXITY* %
1	100%
2	100%
3	100%

* -Support fixity of 0% denotes pinned condition.
 Support fixity of 99% denotes fixed end condition.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 6
3:18:00 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

LATERAL LOAD/OUTPUT DATA

LATERAL LOADS ARE NOT SPECIFIED

OUTPUT DATA
PATTERN LOADINGS: 1 THRU 4
PATTERN LIVE LOAD FACTOR (1-3) = 75%

LOAD FACTORS:
U = 1.40*D + 1.70*L
U = .75(1.40*D + 1.70*L + 1.70*W)
U = .90*D + 1.00*W

OUTPUT OPTION(S):
Input Echo
Centerline Moments and Shears
Column Strip Distribution Fac
Shear Table
Reinforcing Required
Bar Sizing
Additional Information
Deflections
Material Quantities

**TOTAL UNFACTORED DEAD LOAD = 187.800 kips
LIVE LOAD = 64.800 kips

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 7
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

---- STATICS PRINT-OUT FOR GRAVITY LOAD ANALYSIS ----

J O I N T M O M E N T S (ft - kips)

JOINT NUMBER	PATTERN-1				PATTERN-2			
	LEFT	RIGHT	TOP	BOTTOM	LEFT	RIGHT	TOP	BOTTOM
1	-2.7	224.6	.0	-221.9	-2.7	151.1	.0	-148.3
2	-428.6	529.9	.0	-101.3	-349.0	489.7	.0	-140.6
3	-259.6	2.7	.0	256.8	-259.2	2.0	.0	257.2

JOINT NUMBER	PATTERN-3				PATTERN-4			
	LEFT	RIGHT	TOP	BOTTOM	LEFT	RIGHT	TOP	BOTTOM
1	-2.0	224.4	.0	-222.4	-3.0	231.8	.0	-228.8
2	-405.1	446.4	.0	-41.3	-462.7	570.9	.0	-108.2
3	-179.2	2.7	.0	176.5	-265.6	3.0	.0	262.6

J O I N T S H E A R S (kips)

JOINT NUMBER	PATTERN-1		PATTERN-2		PATTERN-3		PATTERN-4	
	LEFT	RIGHT	LEFT	RIGHT	LEFT	RIGHT	LEFT	RIGHT
1	-5.5	67.1	-5.5	48.0	-3.9	67.1	-6.0	71.1
2	-83.6	102.8	-64.5	100.9	-82.2	80.8	-90.4	110.7
3	-84.5	5.5	-84.4	3.9	-61.7	5.5	-88.9	6.0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 8
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN MOMENT ENVELOPES AT CRITICAL SECTIONS FROM SUPPORTS

COL NUM	LOAD TYPE	CROSS SECTN	DESIGN MOMENT (ft-k)	DISTANCE CR. SECTN (ft)	LOAD PTRN	MAX. I. P. DISTANCE (ft)	LOAD PTRN
1	TOTL LEFT	TOP	-2.0	.175	4	1.000	1
		BOT	.0	.000	0	.000	0
	RGHT	TOP	163.7	1.000	4	3.600	2
		BOT	.0	.000	0	.000	0
2	TOTL LEFT	TOP	-375.4	1.000	4	7.200	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	463.3	1.000	4	7.000	3
		BOT	.0	.000	0	.000	0
3	TOTL LEFT	TOP	-179.9	1.000	4	2.800	2
		BOT	.0	.000	0	.000	0
	RGHT	TOP	2.0	.175	4	1.000	1
		BOT	.0	.000	0	.000	0

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 9
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DESIGN MOMENT ENVELOPES AT CRITICAL SECTIONS ALONG SPANS

SPAN NUM	LOAD TYPE	CRITICAL SECTION (ft)	DESIGN MOMENT (ft-k)	LOAD PTRN	MAX. I.P. DIST LEFT (ft)	LOAD PTRN	MAX. I.P. DIST RGHT (ft)	LOAD PTRN
2	TOTL	10.200 TOP	.0	0	.000	0	.000	0
		BOT	149.4	4	6.600	1	7.800	3
3	TOTL	16.100 TOP	.0	0	.000	0	.000	0
		BOT	297.2	4	10.500	2	9.100	1

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 10
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DISTRIBUTION OF DESIGN MOMENTS AT SUPPORTS

COL NUM	CROSS SECTN	TOTAL MOMENT (ft-k)	TOTAL-VERT DIFFERENCE (ft-k) (%)	COLUMN STRIP MOMENT (ft-k) (%)	BEAM MOMENT (ft-k) (%)	MIDDLE STRIP MOMENT (ft-k) (%)
1	LEFT TOP	-2.0	.0 (0)	.0 (0)	-2.0 (100)	.0 (0)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	163.7	.0 (0)	.0 (0)	163.7 (100)	.0 (0)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
2	LEFT TOP	-375.4	.0 (0)	.0 (0)	-375.4 (100)	.0 (0)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	463.3	.0 (0)	.0 (0)	463.3 (100)	.0 (0)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
3	LEFT TOP	-179.9	.0 (0)	.0 (0)	-179.9 (100)	.0 (0)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	RGHT TOP	2.0	.0 (0)	.0 (0)	2.0 (100)	.0 (0)
	BOT	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)

DISTRIBUTION OF DESIGN MOMENTS IN SPANS

SPAN NUM	CROSS SECTN	TOTAL MOMENT (ft-k)	TOTAL-VERT DIFFERENCE (ft-k) (%)	COLUMN STRIP MOMENT (ft-k) (%)	BEAM MOMENT (ft-k) (%)	MIDDLE STRIP MOMENT (ft-k) (%)
2	10.20 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	149.4	.0 (0)	.0 (0)	149.4 (100)	.0 (0)
3	16.10 TOP	.0	.0 (0)	.0 (0)	.0 (0)	.0 (0)
	BOT	297.2	.0 (0)	.0 (0)	297.2 (100)	.0 (0)

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 11
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

B E A M S H E A R R E Q U I R E M E N T S (kips, sq.in./in., ft.)

NOTE--Allowable shear stress in beams = 126.48 psi (see "CODE").

BEAM SPAN NO.	LEFT PATT. NO.	LEFT Vu@d SHEAR	SIDE Av/s @d	--FRACTIONAL DIST. ALONG SPAN--				RIGHT Av/s @d	SIDE Vu@d SHEAR	LEFT Vc/2. DIST.	
				.175	.375	.625	.825				
1	*	*	Span length equal to column size or zero							*	*
2	4	52.6	.020*	.020*	.000	.020*	.020*	.020*	-71.8	6.60	
3	4	84.2	.020*	.020*	.000	.000	.020*	.020*	-62.4	10.50	
4	*	*	Span length equal to column size or zero							*	*

- NOTES: 1.) To obtain stirrup spacing, divide stirrup area by Av/s value above.
 2.) To obtain stirrup area, multiply spacing by Av/s value.
 3.) Local effects due to loadings applied at other segments along beam span must be calculated manually.
 4.) Symbols following Av/s values:
 * - minimum shear $50 \cdot bw / Fyv$ - based on beam dimensions.
 x - Vs exceeds $2 \cdot Vc$, maximum stirrup spacing must be halved.
 + - Av/s value at segment located within effective depth.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 12
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

N E G A T I V E R E I N F O R C E M E N T

COLUMN NUMBER	*PATT NO.	*LOCATION *@COL	*FACE *	*DESIGN * (ft-k)	* B E A M		
					* TOTAL	* TOP STEEL	* BOTTOM STEEL
					* AREA	* AREA	* AREA
1	4		R	163.7	1.78	24.0	*
2	4		R	463.3	2.74	24.0	*
3	4	L		-179.9	1.58	24.0	*

* - No doubly reinforced section is required.

P O S I T I V E R E I N F O R C E M E N T

SPAN NUMBER	*PATT NO.	*LOCATION *FROM LEFT	*DESIGN * (ft)	* TOTAL * (ft-k)	* B E A M		
					* BOTTOM STEEL	* TOP STEEL	
					* AREA	* AREA	* AREA
2	4	10.2		149.4	1.78	24.0	*
3	4	16.1		297.2	2.63	24.0	*

* - No doubly reinforced section is required.

NOTE: Beam reinforcement areas are based on one layer.

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 13
 3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

D E S I G N R E S U L T S

NOTE--The schedule given below is a guide for proper reinforcement placement and is based on reasonable engineering judgement. Unusual boundary and/or loading conditions may require modification of this schedule.

--The beam reinforcement detailing such as hooks and bar lengths, should be done manually to ensure proper bar detailing.

N E G A T I V E R E I N F O R C E M E N T

		B E A M					
		* TOP BARS		* BOTTOM BARS			
COLUMN	*	* ----- B A R -----		* ----- B A R -----			
NUMBER	*	NO	SIZE	LAYERS	*	NO	SIZE
1		9	# 4	1		No double reinf req'd	
2		9	# 5	1		No double reinf req'd	
3		8	# 4	1		No double reinf req'd	

P O S I T I V E R E I N F O R C E M E N T

		B E A M					
		* BOTTOM BARS		* TOP BARS			
SPAN	*	* ----- B A R -----		* ----- B A R -----			
NUMBER	*	NO	SIZE	LAYERS	*	NO	SIZE
2		9	# 4	1		No double reinf req'd	
3		13	# 4	1		No double reinf req'd	

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 14
3:18:01 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

A D D I T I O N A L I N F O R M A T I O N F O R
I N - S P A N C O N D I T I O N S

SPAN NUMBER*	REINF. REQ'D. (sq.in)	SUMMARY PROV'D. (sq.in)	TOTAL FACTORED SPAN STATIC DESIGN MOMENT (W/O PARTIAL LOADS) (ft-k)
2	1.78	1.80	N/A
3	2.63*	2.60	N/A

* - Indicates REQ'D reinforcement is greater than PROV'D (check bar selection)

07-23-1999 ADOSS(tm) 7.00 Proprietary Software of PORTLAND CEMENT ASSN. Page 15
 3:18:02 PM Licensed to: Canadian Portland Cement Association, OTTAWA, ONTARIO

DEFLECTION ANALYSIS

NOTES--Spans 1 and 4 are cantilevers.

--Time-dependent deflections are in addition to those shown and must be computed as a multiplier of the dead load(DL) deflection. See "CODE" for range of multipliers.

--Deflections due to concentrated or partial loads may be larger at the point of application than those shown at the centerline.

--Modulus of elasticity of concrete, Ec = 3834. ksi

SPAN NUMBER	* C O L U M N S T R I P *				* M I D D L E S T R I P *			
	DEAD LOAD	DEFLECTION DUE TO:			DEFLECTION DUE TO:			
	I _{eff.} (in ⁴)	DEAD (in)	LIVE (in)	TOTAL (in)	DEAD (in)	LIVE (in)	TOTAL (in)	
1	27648.	-.004	-.001	-.005	Not	a p p l i c a b l e		
2	23036.	.042	.021	.062	Not	a p p l i c a b l e		
3	86005.	.045	.016	.061	Not	a p p l i c a b l e		
4	93312.	-.004	-.001	-.006	Not	a p p l i c a b l e		

Q U A N T I T Y E S T I M A T E S

TOTAL QUANTITIES

CONCRETE	10.1 cu.yd
FORMWORK	490. sq.ft
REINFORCEMENT	(COMPUTE MANUALLY)

* Program completed as requested *

Index

.ADS, 3-1
.OUT, 4-54

A

About ADOSS, 3-12, 4-65
 A_c , 2-10, 2-36 - 2-37, 2-44
ACI, 4-12
add mode, 4-17
ADOSS icon, 1-10
 a_f , 2-60
 A_s , 2-51
alternate design, 4-14
angles, 4-44
Arrange Icons, 3-11
 A_s , 2-52, 2-54
ASSIGN, 1-8
 a_{strip} , 2-62
 A_t , 2-50
 a_{\emptyset} , 2-62
 A_{tot} , 2-22, 2-50
Auto Input, 3-1, 3-4, 4-5
 A_v , 2-46, 2-50
 α_1 , 2-32, 2-41, 2-43, 2-45, 2-48, 2-61
 α_s , 2-40

B

b_1 , 2-45
 b_2 , 2-45
Back, 4-65
Background Color, 3-9, 4-56
backup copy, 1-5
bar size
 maximum, 4-14

metric, A-3
minimum, 2-54, 4-14
U.S., A-2
bar spacing
 maximum, 2-54
 minimum, 2-53 - 2-54, 4-14
 b_c , 2-40
Beam Reinforcement, 4-10, 4-14
beam supported slab, 2-7, 2-45, 2-65
beams
 depth, 2-36
 edge, 2-7
 flange, 2-4
 shallow, 2-35
 T-beam, 2-2, 2-4, 2-23, 2-52 -
 2-53
 transverse, 2-31
 b_o , 2-41
box
 check box, 4-4, 4-45, 4-46
 dialog, 4-3
 edit, 4-3
building code, 4-11
building standard, 4-11
buttons
 Copy From, 4-16
 Delete, 4-39
 Insert, 4-38
 push, 4-5
 radio, 4-4, 4-56 - 4-59
 Replace, 4-17
 b_w , 2-46
 β_1 , 2-52
 β_2 , 2-32, 2-43, 2-48, 2-61

C

- C, 2-18, 2-21, 2-32
- c₁, 2-1, 4-28
- c₂, 2-1, 2-21, 2-33, 4-28
- cantilever, 2-1, 2-62, 4-19, 4-25
- capitals, 2-1 - 2-2, 2-13, 2-31, 2-52, 2-59
 - defining, 4-26
 - depth, 4-27
 - extension, 4-27
- Cascade, 3-10, 4-65
- Change Isometric Angle, 3-6, 4-44, 4-61
- check box, 4-4, 4-45, 4-47
- checks
 - code, 2-2 - 2-7
 - geometric, 2-1
- CHKDSK, 1-5
- clear rib spacing, 4-21
- Close, 3-13
- code, 4-11
 - checks, 2-2 - 2-7, 5-3
- colors, 4-56 - 4-57
 - background, 3-9, 4-56 - 4-58
 - loads, 3-9, 4-59
 - members, 3-9, 4-59
- Column Fixity, 3-5, 4-33
 - changing, 4-33
- columns
 - c₁, 4-26
 - c₂, 4-26
 - defining, 4-26
 - designing, 4-50, 5-3
 - equivalent, 2-15
 - fixed, 4-33
 - fixity, 4-17
 - height, 4-27
 - pinned, 4-27 - 4-28, 4-32
 - round, 4-26
 - service loads, 4-50
 - strip, 2-28, 2-31, 2-33 - 2-34, 2-51
- Columns/Capitals, 4-27
- commands, 3-11, 4-66
 - About ADOSS, 3-12, 4-68
 - Arrange Icons, 3-11
 - Auto Input, 3-1, 3-4, 4-6, 4-10
 - Background Color, 3-9, 4-56
 - Beam Reinforcement, 4-10, 4-14
 - Cascade, 3-10, 4-65
 - Change Isometric Angle, 3-6, 4-46, 4-65
 - Close, 3-13
 - Column Fixity, 3-5, 4-34
 - Columns/Capitals, 4-26
 - Commands, 3-11, 4-63
 - Condense/Expand Output, 3-3, 3-8, 4-47, 4-54
 - Default Data Directory, 1-9, 3-10, 4-60
 - Default Units/Code, 1-8, 3-9, 4-56
 - Design Column, 3-8, 4-50
 - Design System, 3-8, 4-6, 4-50, 4-54 - 4-56
 - disabled, 4-4
 - Drops, 4-29
 - enabled, 4-4
 - Exit, 3-3
 - Export, 3-2, 4-7
 - Factors, 4-40, 4-41

- General Information, 3-4, 4-10-4-11, 4-16, 4-39, 4-62
- Index, 3-11, 4-63
- Labels, 3-4, 4-9
- Lateral Forces, 4-40
- Lateral Moments, 4-41
- Load Colors, 3-9, 4-57
- Longitudinal Beams, 4-20, 4-22 - 4-24
- Material Properties, 3-4, 4-10, 4-13
- Maximize, 3-13
- Member Colors, 3-9, 4-59
- Minimize, 3-13
- More Windows, 4-61
- Move, 3-12
- New, 3-1, 4-5
- Next, 3-13
- Open, 3-1, 4-8, 4-63
- Page Layout, 3-10, 4-55, 4-56, 4-60
- Partial Loads, 4-34
- Print Results, 3-3, 4-52
- Print Screen, 3-3, 4-53, 4-54
- Procedures, 3-12, 4-63
- Redraw, 3-6, 4-43
- Restore, 3-12, 4-62
- Revert, 3-3
- Ribs, 4-21
- Save, 3-2, 4-6, 4-50, 4-54 - 4-56
- Save As, 3-2, 4-5
- Scale, 3-6, 4-43
- selecting, 4-3
- Size, 3-13
- Slab Geometry, 4-18
- Slab Reinforcement, 4-14
- Surface Loads, 4-35
- Tile All Windows, 3-11, 4-62
- Tile View Windows, 3-11, 4-62
- Transverse Beams, 4-31
- Using Help, 3-12, 4-64
- View Deflections, 3-8, 4-55
- View Loads, 3-7, 4-48
- View Members, 3-7, 4-45
- View Results, 3-7, 4-52
- View Shears/Moments, 3-7, 4-53
- Zoom, 3-6, 4-45
- concrete
 - compressive strength, 2-11, 2-47, 4-13
 - density, 2-8, 2-10 - 2-11, 4-12
 - modulus of elasticity, 2-11
 - modulus of rupture, 2-12, 4-12
 - quantity, 2-67
 - split tensile strength, 2-11, 2-41, 4-12
 - type, 2-10, 2-41, 4-14
- Condense/Expand Output, 3-3, 3-8, 4-49, 4-52
- condensing output, 3-3, 4-41
- CONFIG.SYS, 1-2
- continuous beam, 2-8, 2-64 - 2-65
- Control menu, 3-12, 4-1, 4-62
 - Close, 3-13
 - Maximize, 3-13
 - Minimize, 3-13
 - Move, 3-12
 - Next, 3-13
 - Restore, 3-12
 - Size, 3-13
- controls, 4-4
 - check box, 4-4

- push button, 4-5
- radio buttons, 4-4
- conversion factors, A-1
- Copy From button, 4-16 - 4-18
- cracking, 2-59, 2-64
- creep, 2-66
- critical section
 - closed, 2-36 - 2-38
 - L-shaped, 2-39
 - open, 2-37, 2-41
 - U-shaped, 2-38
 - CSA, 4-11
 - C_t , 2-49
- customer information, 1-9
- customizing, 4-59 - 4-63

D

- data files
 - creating, 3-1, 3-4, 4-5
 - old format, 3-2, 4-7
 - opening, 3-2, 4-8
 - saving, 3-2, 4-6
 - working with, 4-5 - 4-8
- date, 4-9
- d_b , 2-53
- default
 - code, 4-62
 - data directory, 3-2, 4-60
 - units, 4-62
- Default Data Directory, 1-9, 3-10, 4-60
- Default Units/Code, 1-8, 3-9, 4-59
- definitions, 4-64
- deflections, 4-55
 - analysis, 2-12, 4-13, 5-10
 - calculations, 2-59 - 2-66
- diagrams, 4-6, 4-55
- instantaneous, 2-66
- lateral distribution factor, 2-60
- long time, 2-66
- midspan, 2-60 - 2-61
- printing, 3-3, 3-10, 4-51, 4-55
- viewing, 3-8, 4-55, 4-63

- Delete button, 4-39
- density, 2-8, 2-10 - 2-11, 4-12
- depth
 - beam, 2-36
 - capital, 4-27
 - drop, 2-52, 4-29
 - effective, 2-2, 2-36, 2-52
 - longitudinal beam, 4-22 - 4-24
 - rib, 4-21
 - slab, 2-37
 - transverse beam, 4-31
- design
 - column, 4-50
 - method, 4-12
 - performing a, 4-49 - 4-52
 - results, 4-54
- Design Column, 3-8, 4-50
- Design menu, 3-8
 - Condense/Expand, 3-3
 - Condense/Expand Output, 3-8, 4-47, 4-52
 - Design Column, 3-8, 4-50
 - Design System, 3-8, 4-6, 4-52, 4-56 - 4-58
 - Design System, 3-8, 4-6, 4-52, 4-56 - 4-58
- designing, 4-50
- desktop, 4-51
- development length, 2-11, 4-13 - 4-14, 5-9

DEVICE=, 1-2

diagrams

deflections, 3-3, 3-8, 3-10, 4-6,
4-63

printing, 4-57

shear and moment, 3-3, 3-7,
3-10, 4-6, 4-53, 4-56,

4-65

dialog box, 4-3

directory

ADOSS, 1-6

default data, 1-8

root, 1-3

subdirectory, 1-3

disk drives, 1-1

disks, 1-5

drops, 2-14 - 2-15, 2-40, 2-59, 4-29

defining, 4-29

depth, 2-52, 4-30

length, 4-30

standard, 4-29

valid, 2-2, 2-7

waffle depth, 2-9

width, 4-30

E

E_c , 2-60

E_{cb} , 2-32

eccentricity

transverse beam, 4-31

E_{cs} , 2-18, 2-32

edge

discontinuous, 2-7

edit box, 4-3

elevated view, 3-3, 3-10, 4-51, 4-62 -

4-63

engineer, 4-10

enhanced mode, 1-5

epoxy-coated, 2-57, 4-13, 5-9

equivalent frame method, 2-12, 2-21,
2-60 - 2-61

examples, C-1 - C-63

Exit, 3-3

expanding output, 4-47

Export, 3-2, 4-7

extension

capital, 4-26

exterior, 4-12, 4-22

F

factors, 4-40, 4-42

conversion, A-1

distribution, 2-31 - 2-33, 5-5 - 5-6,

f'_c , 2-11, 2-52, 4-14

f_{ct} , 2-10 - 2-11, 2-41, 4-13

File menu, 3-1

Exit, 3-3

Export, 3-2, 4-7

New, 3-1, 4-5

Open, 3-1, 4-8, 4-62

Print Results, 3-3, 4-55

Print Screen, 3-3, 4-57 - 4-58,

4-60

Revert, 3-3

Save, 3-2, 4-6, 4-50, 4-54 -

4-55

Save As, 3-2, 4-6

fixed, 4-33

fixity, 4-17

flange

- depth, 4-22
- overhang, 2-4
- width, 2-23

flat plate, 2-7, 2-34, 2-64 - 2-65

flat slab, 2-7, 2-34, 2-64 - 2-65

floor system, 4-10

- creating, 4-9
- modeling, 4-10
- specifying, 4-11
- forces, 4-11, 4-38

formwork

- quantity, 2-67, 5-10

f_r , 2-12, 4-14

f_y , 2-52, 4-14

f_{yv} , 2-46, 4-14

G

General Information, 3-4, 4-10, 4-14,
4-38, 4-61

printout, 5-2

using, 4-11

geometry, 4-18

defining, 4-16

Geometry menu, 3-5, 4-10, 4-12, 4-16

Columns/Capitals, 4-27

Drops, 4-30

Longitudinal Beams, 4-20,

4-22 - 4-24

Ribs, 4-21

Slabs, 4-18

Transverse Beams, 4-30

γ_f , 2-58

γ_v , 2-44

H

haunched beam, 4-23

height

column, 4-28

help, 4-66

Help menu, 3-11, 4-63

About ADOSS, 3-12, 4-65

Commands, 3-11, 4-63

Index, 3-11, 4-62

Procedures, 3-12, 4-63

Using Help, 3-12, 4-65

help system

Back, 4-64

Index, 4-64

Search, 4-64

I

I_e , 2-64 - 2-65

I_{frame} , 2-60

I_g , 2-65

Index, 3-11, 4-63 - 4-64

inflection points, 2-56, 5-4

input echo, 4-50, 5-2

Input menu, 3-4

Auto Input, 3-1, 3-4, 4-6, 4-10

Column Fixity, 3-5, 4-33

General Information, 3-4, 4-10-
4-11, 4-16, 4-39, 4-64

Labels, 3-4, 4-9

Material Properties, 3-4, 4-10,
4-12

Insert button, 4-37

installing, 1-6 - 1-11

interior, 4-11, 4-21

ADOSS Graphics

I_s , 2-21, 2-32

I_{sb} , 2-21

isometric view, 3-3, 3-6, 3-10, 4-44
 changing angle, 4-44

J

J_c , 2-44

K

K_{cc} , 2-22

K_t , 2-18, 2-21, 2-48

K_{ta} , 2-21

L

l_1 , 2-28, 2-31, 2-46

l_2 , 2-21, 2-28, 2-33

Labels, 3-4, 4-9

lateral factors, 4-51

Lateral Forces, 4-38

lateral loads

 portal analysis, 4-38

 specifying, 4-12, 4-38

Lateral Moments, 4-39

layer, 4-16

l_c , 2-15

length

 drop, 4-31

 haunch, 4-25

 span, 4-19, 4-24 - 4-25

limit states, 4-11

Load Colors, 3-9, 4-59

loads

 additional, 4-50

 applied, 2-22

 colors, 3-9

 column service, 4-47

 combination, 2-27

 concentrated, 2-45

 dead to live, 2-13

 factors, 4-42

 gravity, 5-4

 hiding, 4-48

 lateral, 2-23, 2-25, 2-33 - 2-34,

 2-45, 4-11, 4-39, 5-4, 5-6

 moments, 4-35

 partial, 2-23, 2-41, 2-59, 4-20 -
 4-22, 4-34 - 4-39, 4-48

 patterns, 2-15, 2-25, 3-7, 4-11,
 4-56

 portal analysis, 4-38

 reduction factor, 4-40

 self-weight, 2-3 - 2-4, 2-22,
 2-41

 specifying, 4-35 - 4-41

 superimposed, 2-22

 surface, 2-41, 4-18, 4-35

 transverse beam, 2-47

 trapezoidal, 4-36

 uniform, 4-35

Loads menu, 3-5, 4-10, 4-11

 Factors, 4-40, 4-43

 Lateral Forces, 4-38

 Lateral Moments, 4-38

 Partial Loads, 4-36

 Surface Loads, 4-34

location, 4-12

longitudinal beam

 defining, 4-22

 depth, 4-22 - 4-24

 haunched, 4-24

shear, 2-45 - 2-46, 5-7
width, 4-22 - 4-24
Longitudinal Beams, 4-18, 4-22 - 4-24

M

margins, 3-10, 4-57, 4-59
Material Properties, 2-10 - 2-11, 3-4,
4-10
defining, 4-12
material quantities, 2-67, 4-49, 5-10
Maximize, 3-13
maximums
bar size, 4-14
bar spacing, 2-54
 M_{cr} , 2-65
Member Colors, 3-9, 4-57
members
colors, 3-9, 4-57
hiding, 4-44
menu line, 4-2
menus
accessing, 4-2
closing, 4-3
commands, 4-3
Control, 3-12, 4-1, 4-63
Design, 3-8
File, 3-1, 4-5
Geometry, 3-5, 4-10, 4-11
Help, 3-11
Input, 3-4
Loads, 3-5, 4-10, 4-11
Options, 3-9
pop-up, 4-2
Reinforcement, 3-5, 4-10, 4-13
View, 3-6

Window, 3-10, 4-53, 4-57 -
4-58, 4-62 - 4-63
message, 2-7 - 2-8, 2-52, 4-50
middle strip, 2-28, 2-31, 2-33 - 2-34,
2-51
Minimize, 3-13
minimums
bar size, 2-54, 4-15
bar spacing, 2-54, 4-15
modeling, 4-9
modulus of elasticity
concrete, 2-11
reinforcement, 2-12
modulus of rupture, 2-12, 4-13
moment
beam, 2-33 - 2-34
centerline, 4-50
column, 4-53
column strip, 2-31
cracking, 2-65
design, 2-23, 2-31 - 2-33, 2-51,
4-51, 5-5 - 5-6
diagram, 3-7, 3-10, 4-6, 4-54,
4-54, 4-63
lateral analysis, 4-12, 4-38
middle strip, 2-33 - 2-34
negative, 2-51
printing, 4-55, 5-5
proportioned, 2-41, 2-43, 2-45,
2-48
segmental, 4-50, 5-5
total static, 2-59
transfer, 2-59
unbalanced, 2-42
moment of inertia, 2-13
effective, 2-64 - 2-65
gross, 2-65

More Windows, 4-65
Move, 3-12
M_u, 2-51, 2-58
M_{unbal}, 2-42, 2-44, 2-48

N

New, 3-1, 4-5
Next, 3-13
number of spans, 4-10

O

one-way slab, 2-8, 2-64 - 2-65
Open, 3-1, 4-8, 4-63
optimize, 4-9
Option menu
 Page Layout, 4-57
options
 code, 1-8
 default, 1-8
 units, 1-8
Options menu, 3-9
 Background Color, 3-9, 4-57
 Default Data Directory, 1-9,
 3-10, 4-61
 Default Units/Code, 1-8, 3-9,
 4-59
 Load Colors, 3-9, 4-59
 Member Colors, 3-9, 4-58
 Page Layout, 3-10, 4-59, 4-64
output
 condense, 4-47
 expand, 4-47

P

P_a, 4-40
Page Layout, 3-10, 4-55, 4-59, 4-62
partial loads, 2-41, 2-59, 4-18, 4-20
 changing, 4-35
 deleting, 4-35
 hiding, 4-46
 inserting, 4-34
 specifying, 4-34
 PATH, 1-3, 3-8, 4-50
P_b, 4-40
PCA Programs, 1-10
PCA SETUP, 1-6
PCA View, 3-7, 4-52
PCACOL, 3-8, 4-52, 4-52, 5-3
PCAPROGS.INI, 1-8, 3-9, 4-59 - 4-61,
 4-62
pinned, 4-28 - 4-29, 4-34
plan view, 3-3, 3-10, 4-64 - 4-62
portal analysis, 2-23
print, 4-56, 4-58
Print Results, 3-3, 4-57
Print Screen, 3-3, 4-55
printing
 deflections, 4-55
 results, 4-52
 shear and moment, 4-54
Procedures, 3-12, 4-63
project id, 4-9
proportioned moments and shears, 2-41,
2-43, 2-45, 2-48, 4-13
push buttons, 4-5

Q

quantities, 2-67, 5-10

R

radio buttons, 4-4, 4-51

README, 1-10

Redraw, 3-6, 4-43

reduction factor, 4-39

references, B-1

reinforcement

 additional, 2-58, 4-49, 5-9

 bend, 2-54

 cover, 2-54

 defining, 4-15

 development length, 2-11, 4-13
 - 4-15, 5-9

 distance to centroid, 4-15

 epoxy-coated, 2-57, 4-13, 5-9

 layer, 2-54, 4-16

 maximum bar size, 4-13

 maximum spacing, 2-54

 metric, 4-11

 minimum, 2-53

 minimum bar size, 2-54, 4-13

 minimum spacing, 2-53 - 2-54,
 4-15

 negative, 5-8

 positive, 5-8

 quantity, 2-67, 5-10

 required, 4-51

 schedule, 2-59, 4-49, 5-8

 yield strength, 2-46, 4-13

Reinforcement menu, 3-5, 4-10, 4-13

 Beams, 4-10

 Slabs, 4-10

Replace button, 4-16

requirements

 hardware, 1-5

 software, 1-5

Restore, 3-12, 4-65

results

 printing, 4-52

 viewing, 4-52

Revert, 3-3

ribs

 clear spacing, 4-22

 defining, 4-21

 depth, 4-22

 valid, 2-4, 2-40

 width, 4-22

Ribs Geometry, 4-22

run, 1-11

running ADOSS, 1-11

ρ_b , 2-52

ρ_{min} , 2-53

S

Save, 3-2, 4-6, 4-50, 4-54

Save As, 3-2, 4-6

Scale, 3-6, 4-42

Search, 4-64

segmental

 envelopes, 5-5

 moments, 4-50, 5-5

shears, 5-5

self-weight, 2-3 - 2-4, 2-22, 2-41, 4-34

shear

 allowable, 2-39, 2-45

 area, 2-35

- centerline, 4-48
 - column, 4-51
 - diagram, 3-3, 3-7, 3-10, 4-6, 4-52, 4-65
 - direct, 4-51
 - longitudinal beam, 2-45 - 2-46, 5-7
 - moment transfer, 2-35, 2-49
 - one-way, 2-34, 2-36
 - printing, 4-57, 5-7
 - proportioned, 2-41, 2-43, 2-45, 2-48
 - punching, 2-34 - 2-35, 2-45, 5-7
 - segmental, 4-50, 5-5
 - transverse beam, 2-47
 - wide beam, 2-34
 - shear/moment transfer, 2-34
 - shrinkage, 2-66
 - SI, 4-11
 - sign convention, 2-24
 - Size, 3-13
 - Slab Geometry, 4-18
 - Slab Reinforcement, 4-10, 4-14
 - slabs
 - defining, 4-18
 - thickness, 4-22
 - soffit, 2-1, 2-4
 - span
 - length, 4-18, 4-23
 - width, 4-22
 - span id, 4-10
 - split tensile strength, 2-11, 2-41, 4-13
 - standard drop, 4-29
 - stiffness, 2-13
 - beam, 2-4
 - capital, 2-4
 - column, 2-15
 - drop, 2-3
 - torsional, 2-15
 - waffle , 2-9
 - stirrup, 2-46 - 2-47
 - STMFR II, 2-23
 - strength design, 4-11
 - strips
 - beam, 2-51
 - column, 2-28, 2-31, 2-33 - 2-34, 2-51
 - middle, 2-28, 2-31, 2-33 - 2-34, 2-51
 - surface loads, 2-41, 4-18, 4-33
 - specifying, 4-32
- ## **T**
- T-beam, 2-2, 2-4, 2-23, 2-52 - 2-53, 4-21
 - Tc, 2-49 - 2-50
 - thickness
 - minimum, 2-6 - 2-7
 - slab, 2-52
 - waffle slab, 2-9
 - Tile All Windows, 3-11, 4-63
 - Tile View Windows, 3-11, 4-63
 - time, 4-10
 - title bar, 4-1
 - torque, 2-47 - 2-48
 - torsion
 - transverse beam, 2-47
 - transverse beam
 - defining, 4-31
 - depth, 4-31
 - eccentricity, 4-32

shear, 2-47 - 2-50, 5-7
torsion, 2-47 - 2-50, 5-7
width, 4-30
Transverse Beams, 4-30
 T_u , 2-48, 2-50
type, 4-14

U

units, 4-11
Untitled, 4-5
UNTITLED.ADS, 4-6
UNTITLED.OUT, 4-54
US, 4-11
Using Help, 3-12, 4-68

V

valid
 drops, 2-2, 2-7
 ribs, 2-4, 2-41
 V_c , 2-11, 2-40 - 2-41, 2-46, 2-50
version, 4-68, 5-2
view
 elevated, 3-3, 4-62
 isometric, 3-3, 4-44, 4-64
 plan, 3-3, 4-62
View Deflections, 3-8, 4-58
View Loads, 3-7, 4-48
View Members, 3-7, 4-47
View menu, 3-6
 Change Isometric Angle, 3-6,
 4-44, 4-63
 Redraw, 3-6, 4-46
 Restore, 4-65
 Scale, 3-6, 4-43

View Deflections, 3-8, 4-56
View Loads, 3-7, 4-46
View Members, 3-7, 4-46
View Results, 3-7, 4-54
View Shears/Moments, 3-7,
 4-54
Zoom, 3-6, 4-43

View Results, 3-7, 4-52
View Shears/Moments, 3-7, 4-54
viewing
 deflections, 4-57
 floor system, 4-42
 results, 4-52
 scale, 4-42
 shears and moments, 4-54
views
 elevated, 3-10, 4-53
 isometric, 3-6, 3-10
 plan, 3-10
 V_u , 2-41, 2-43, 2-45 - 2-50

W

waffle slab, 2-2, 2-7, 2-34, 2-64 - 2-65,
4-22, 4-30, 4-32
 ribs, 2-4
 special considerations, 2-8 -
2-9
 w_{bend} , 2-54
width
 drop, 4-32
 effective slab, 2-58
 longitudinal beam, 4-23
 ribs, 4-22
 span, 4-21
 transverse beams, 4-31

ADOSS Graphics

wind, 2-27, 4-12, 4-40

Window menu, 3-10, 4-52

 Arrange Icons, 3-11

 Cascade, 3-10, 4-63

 More Windows, 4-65

 Tile All Windows, 3-11, 4-62

 Tile View Windows, 3-11, 4-62

windows

 deflections, 3-3, 3-8, 3-10,
 4-55

 elevated view, 3-3, 3-10, 4-51

 isometric view, 3-3, 3-6, 3-10,

4-61

 opening, 4-62

 partial loads, 4-62

 plan view, 3-3, 3-10, 4-61

 shear and moment, 3-3, 3-7,
 3-10, 4-55

 surface loads, 4-62

 working with, 4-64

X

x, 2-21, 2-48

x_1 , 2-50

Y

y, 2-21, 2-48

y_1 , 2-50

yield strength, 4-14

Z

Zoom, 3-6, 4-45

zooming, 4-45