CSICOL™

For Analysis and Design of
Reinforced and Composite Columns

USER'S MANUAL AND TECHNICAL REFERENCE



Computers and Structures, Inc.
Berkeley, California, USA

Version 8.00
April 2005

COPYRIGHT

The computer program CSICOL and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Tel: (510) 845-2177
Fax: (510) 845-4096
E-mail: support@csiberkeley.com
Web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2003.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
CSICOL is a trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.
Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF **CSICOL**. THE PROGRAM HAS BEEN THOROUGHLY TESTED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.



Contents

Chapter 1: Introduction to CSICOL **1-1**

Key Features 1-3

Design and Analysis Capabilities 1-3
Slenderness Considerations 1-4
Cross-Section Generation 1-4
Material Properties 1-4
Results Generated 1-4
Miscellaneous 1-5

Terminology 1-6

Results and Output 1-7

Section Capacity 1-7
Magnified Moments 1-8
Stress Distribution and Plots 1-8
Geometric Properties 1-9

Auto Section Design 1-9

Other User Support Documents 1-10

Notation 1-10

Chapter 2: CSICOL's User Interface **2-1**

Drawing Area 2-1

Gridlines 2-1

Axis 2-2
Status Bar 2-3
Dimensions 2-3
Toolbar Buttons and Menu Commands 2-3
Customize the Work Space 2-16
General Options 2-16
Units 2-17
Design Code 2-17
View Options 2-17
Display Color 2-17
Drawing Scale 2-18
Refresh Graphic View 2-19
Section Capacity Options 2-19
3D View Options 2-20
Display Objects 2-20
Lights 2-20
Animation 2-20

Chapter 3: Designing Columns 3-1
Column Design Problem 3-1
Methods for Creating Columns 3-2
Quick Design Wizard 3-3
File Menu > New Rectangular Column 3-6
File Menu > New Circular Column 3-9
File Menu > New Column 3-11
Define the Base Material for the Section 3-14
Add Shapes from a Library 3-15
Shape Libraries 3-16
Add Shapes by Drawing 3-18
Add Shapes by Importing Shape Coordinates 3-19
Specify Column Framing Conditions 3-20
Effective Length Factor 3-23
“EI” Calculator 3-25

Specify Column Loads 3-26
Simple Loading Mode 3-26
Detailed Loading Mode 3-28
Column Auto Design 3-33
Auto Design Options 3-36

Chapter 4: Edit Column Cross-Sections 4-1
Types of Shapes 4-1
Shape Editor 4-2
Manage the Shape Editor Display Area 4-4
Add Rebar at Mouse Clicks 4-6
Rebar Calculator 4-7
Delete Rebar 4-9
Change Rebar Diameter 4-9
Align and Distribute Rebar 4-10
Edit Shape Point Coordinates 4-11
Shape Layout Editor 4-12
Align Shapes Graphically 4-13
Rotate, Flip and Stack Shapes 4-13
Merge Shapes 4-14
Create Holes in a Shape 4-15
Move Shapes 4-15
Add Fillets to Shapes 4-16

Chapter 5: Obtain and Interpret Results 5-1
Overview 5-1
Interaction Surface and Curves 5-2
Interaction Diagrams 5-3
P-M and M-M Curve Tabulated Output 5-5
Interaction Surface View Options 5-6
Capacity Calculations 5-7
Section Stresses 5-8

Moment Curvature Curves 5-11
Geometric Properties 5-12
Other Results 5-12

Chapter 6: Generate a Report 6-1
Overview 6-1
Create a Report 6-1
Preview a Report 6-2
Add to a Report 6-3
Print a Report 6-4
Save a Report 6-4
Save a Report as Text 6-5
Export a Report 6-5



Chapter 1

Introduction to CSICOL

CSICOL is a comprehensive software package for analyzing and designing concrete, reinforced concrete, and reinforced concrete composite columns. CSICOL's Quick Design Wizard provides access to all of the forms needed during the column design process, making design simple, organized and efficient. The design can be completed in accordance with user-specified codes. Analysis and design are interactive. Figure 1-1 illustrates the overall design and analysis process using CSICOL.

The program can design the column cross-sections for specified axial loads and moments directly or can compute the magnified moments caused by slenderness effects. An unlimited number of load combinations can be defined, both for sway and non-sway conditions. Sway and non-sway condition checks may also be performed as specified in the selected design code. In addition, CSICOL is capable of determining the Effective Length Factor on the basis of a column's framing and end conditions. An auto cross-section design tool helps in automatically selecting the column size and reinforcement for specified actions using user-defined rules.

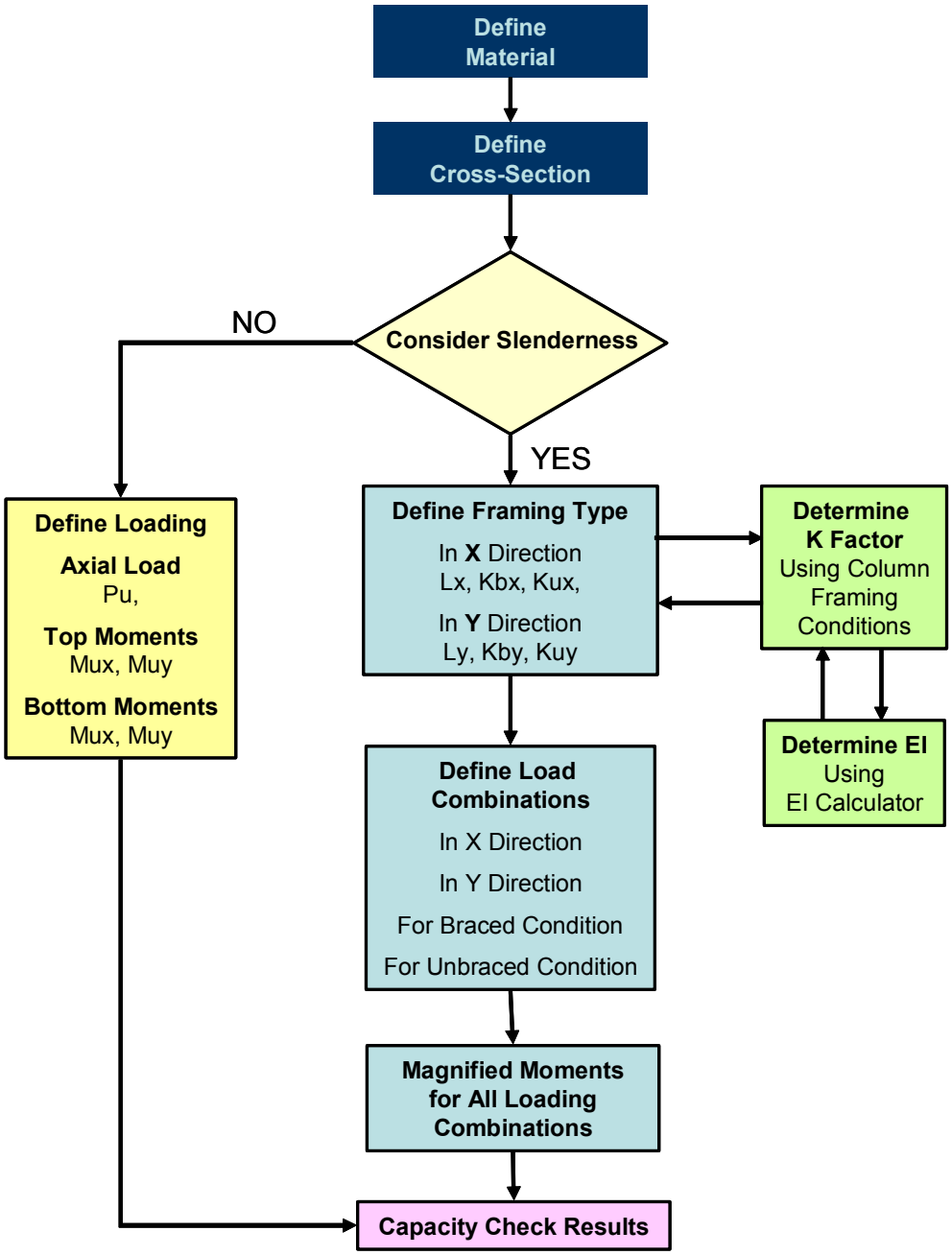


Figure 1-1: Overview of CSICOL Design and Analysis Process

A wide variety of results can be generated by the program. The outputs include the capacity interaction surface, load-moment curves, moment-moment curves, moment-curvature curves for various failure criteria, combined axial-flexural elastic stress contours, rebar stresses, cracked section stresses, load point location, capacity vector, and neutral axis depth and orientation, among others. Reports may be created as part of the output for the analysis and design process. The reports may be customized by adding information and graphics of your choice.

The program provides several predefined parametric shapes, including a variety of solid, hollow, and flanged shapes, in addition to a large collection of Standard Steel Database Shapes that can be used in composite columns. Merge, edit and draw Shapes to suit your geometry requirements and create complex cross-sections. The program provides effective, efficient and practical tools for aligning, stacking, and placing shapes. Rebar can be placed anywhere in the cross-section (corner, perimeter, sides, circle, irregular, and so forth) using several addition and placement tools. Several standard (ASTM, Metric, and Imperial) and user-defined rebar sets may be used. Context-sensitive HTML help makes use of the program easy and efficient.

Key Features

Design and Analysis Capabilities

- Use the Quick Design Wizard capabilities to guide you through the entire column design process.
- Design columns in accordance with user-specified codes.
- Define any number of load combinations for sway and non-sway conditions.
- Apply loads in detailed and simple mode for long and short columns.
- Perform analysis and design considering slenderness effects.
- Perform separate design for the top and the bottom ends of a column.

Slenderness Considerations

- Perform code-specific sway and non-sway condition checks.
- Use the program to determine the Effective Length Factor on the basis of the framing and end conditions.
- Perform analysis and design considering slenderness effects.
- Specify Stiffness Reduction Factors for column and bracing elements.

Cross-Section Generation

- Define and edit multiple column sections at the same time.
- Create rectangular and circular columns using simple tools.
- Use any of the program-provided predefined parametric shapes, including a variety of solid, hollow, and flanged shapes, in addition to a large collection of Standard Steel database shapes.
- Create and then edit complex column sections by combining basic concrete shapes, basic steel shapes, standard steel shapes and user-defined shapes.
- Merge shapes of different geometry to create complex shapes.
- Add rebar of any size anywhere in the cross-section.
- Apply ASTM, Metric, Imperial or user-defined rebar sets.

Material Properties

- Realistically model various materials using one of the many stress-strain curves available in the program for concrete and steel.

Results Generated

- Design the column and perform the check in accordance with the specifications of the selected design code.
- Generate capacity interaction surfaces and diagrams.

- Generate moment-curvature curves for any arbitrary column shape to determine performance and ductility.
- Plot the combined stress resulting from actions (P, M_x, M_y) on a column's cross-section.
- Display the location and orientation of the neutral axis corresponding to a specific loading.
- Display the load point location on the cross-section showing eccentricity.
- Display rebar stresses for selected load combinations.
- Report cracked section stresses.
- Compute and report basic geometric properties, such as A, I_{xx}, I_{yy}, and shear area.
- Calculate area, and I_{xy} as well as extended properties such as S_x, S_y, Z_x, Z_y, r_x, J, and r_y.

Miscellaneous

- Design composite cross-sections using several materials to handle strengthening and retrofitting problems.
- Set the working units to US, Metric or SI.
- Show the location of the centroid and the overall size of a section.
- Use the comprehensive context-sensitive HTML Help to apply the program efficiently.
- Create detailed, customized reports by adding information and graphics.
- Display each type of shape and material in a different color to make interpretation of output easier.

Terminology

Before working with column sections, it is essential that you understand the difference between a shape and a section as used in CSICOL. Other important terms that require clear understanding also are defined herein.

A **Shape** is the basic component that is used to create a cross-section. Rebar can be added to individual shapes. The properties of an individual shape can be determined and its dimensions modified. Predefined shapes from standard libraries can be used and modified to create cross-sections. In addition, CSICOL offers several tools and methods to create cross-sections from shapes.

A **Section** is a combination or collection of shapes placed together to act as a single cross-section. Properties are computed and the design is completed for a section and NOT for a shape (even when a section has only one shape). Figure 1-2 shows a section made using two shapes.

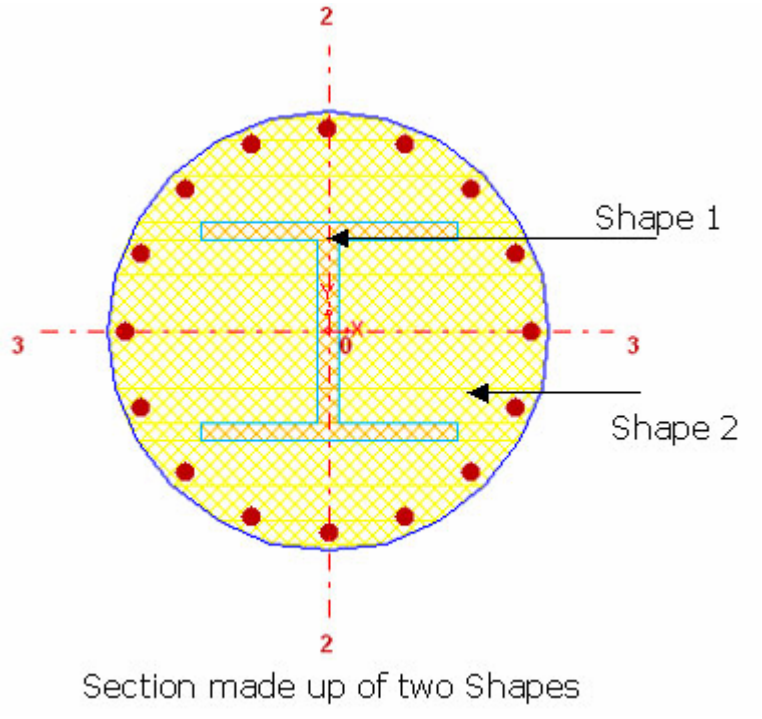


Figure 1-2 A Section Made Using Two Shapes

A **Shape Library** is a collection of standard predefined Shapes that can be edited parametrically (e.g., specifying dimensions) or imported from standard databases. The software includes a number of libraries, such as the Basic Concrete Shape Library and the AISC Steel Sections library.

A **Project File** stores the sections. More than one section can be defined and saved in a single project file.

The **Selection Mode** in CSICOL allows the user to select the various shapes in a section and then use the toolbar buttons and menu commands to align and edit the shapes.

The **Reshaper Mode** in CSICOL allows the user to move selected shapes and to resize them using the mouse.

Results and Output

CSICOL generates the following four types of results:

- Section Capacity
- Magnified Moments
- Stress Distributions and Plots
- Geometric Properties

Section Capacity

The section capacity can be obtained in one of the following three ways:

- Capacity interaction curves
- Moment-curvature plots
- Capacity ratio check

The interaction curves are displayed as follows:

- Load and biaxial moment interaction surfaces showing a 3D plot between the axial load P, moment M_x and moment M_y.

- Load-moment interaction curves showing the relationship of axial load capacity and resultant moment capacity for a specified angle of neutral axis.

- Moment-moment interaction curves showing a plot between moment capacity about the X-axis and moment capacity about the Y-axis at a specified axial load level.

The moment-curvature curves may be plotted for any orientation of the neutral axis and for a specified level of axial load. The user has the control to define the failure criterion of the Section. The user can also specify the limiting value of failure strain, and the strain increment for curve generation.

NOTE: Although the program can generate capacity curves for any section and combination of different shapes and materials, it is important that the user use this information with the proper understanding and check the validity and applicability of such calculations.

The program is capable of calculating and reporting the capacity ratios for all column sections. In addition, the program can display the M-M vector angle, P-M vector length, Capacity Vector, neutral axis depth and orientation; the program also provides a message that the section is or is not adequate. For capacity ratios greater than one, the program reports the section as inadequate.

Magnified Moments

The program performs detailed slenderness calculations to obtain magnified design moments, both for sway and for non-sway loading, in accordance with the procedures specified in the relevant design code. Those calculations can be viewed by selecting the Slenderness Calculation option while generating the report.

Stress Distribution and Plots

The program can generate the following stress plots on the section:

- Elastic combined stresses for axial load P, moment M_x and moment M_y. This stress calculation is based on elastic properties and linear

stress distribution assuming a fully composite and connected behavior of the various Shapes in the Section.

- Cracked section stresses
- Rebar stresses



Note:

Detailed properties, including the effect of modular ratios, can be computed using CSI SectionBuilder.

Geometric Properties

The geometric properties are computed assuming concrete as the base material. These properties are based on geometric dimensions and no modification is made for the modular ratios. This ensures consistent and uniform use of cross-section area and reinforcement ratios.

The program reports the following properties for the section:

- **Basic Properties.** Overall dimensions, centroid location with respect to the global origin, area, moment of inertia about 2-3 axis, elastic section modulus and radii of gyration.
- **Principal Properties.** Principal moments of inertia and the corresponding principal angle.
- **Additional Properties.** Torsional constant, shear areas and plastic section moduli.
- **Global Properties.** These properties are calculated about the global XY axis and are dependent on the location of the section with respect to the origin. These properties include moment of inertia and first moment of areas.

Auto Section Design

The Column Auto Design feature is a very effective, efficient and powerful tool for the design of sway and non-sway columns with or without considering slenderness effects. The Column Auto Design tool designs the section according to the various design parameters defined/specified by the user, including design codes. The user can modify the analysis and design interactively. The program optimizes the

design of the columns by limiting the rebar ratio or the rebar size to the minimum as specified by the user. Limits may be imposed on the maximum and minimum height and width of the section.

Other User Support Documents

In addition to this manual, support documentation includes context sensitive HTML Help and a verification manual. The support documents are available with the purchase of the program or can be ordered independently. For more details, visit www.csiberkeley.com, or email support@csiberkeley.com.

Notations

The following notations are used in the program and its reporting of results.

Note: The ACI-318 notations are used throughout this manual (both text and figures). The readers should use the equivalent notations when using other design codes.

ACI 318

E_c	Modulus of elasticity of concrete
E_s	Modulus of elasticity of reinforcement
EI	Flexural stiffness of compression member
f_c'	Specified compressive strength of concrete
f_s	Calculated stress in reinforcement at service loads
f_y	Specified yields strength of non-prestressed reinforcement

h	Overall thickness of member
k	Effective length factor for compression members
l_c	Length of compression member in a frame, measured from center to center of the joints in the frame
l_u	Unsupported length of compression member
M_c	Factored moment to be used for design of compression member
M_s	Moment due to loads causing appreciable sway
M_u	Factored moment at section
M_1	Smaller factored end moment on a compression member, positive if member is bent in single curvature, negative if bent in double curvature
M_{1ns}	Factored end moment on a compression member at the end at which M1 acts due to loads that cause no appreciable side sway, calculated using a first-order elastic frame analysis
M_{1s}	Factored end moment on a compression member at the end at which M1 acts due to loads that cause appreciable side sway, calculated using a first-order elastic frame analysis
M_2	Larger factored end moment on a compression member, always positive, in-lb
$M_{2\ min}$	Minimum value of M2
M_{2ns}	Factored end moment on a compression member at the end at which M2 acts due to loads that cause no appreciable side sway, calculated using a first-order elastic frame analysis
M_{2s}	Factored end moment on a compression member at the end at which M2 acts due to loads that cause appreciable

side sway, calculated using a first-order elastic frame analysis

P_b Nominal axial load strength at balanced strain conditions

P_c Critical load

P_n Nominal axial load strength at given eccentricity

P_o Nominal axial load strength at zero eccentricity

P_u Factored axial load at given eccentricity

r Radius of gyration of cross section of a compression member

δ_{ns} Moment magnification factor for frames braced against side sway, to reflect effects of member curvature between ends of compression member

δ_s Moment magnification factor for frames not braced against side sway, to reflect lateral drift resulting from lateral and gravity loads

Δ_o Relative lateral deflection between the top and bottom story due to Vu, computed using a first order elastic frame analysis and stiffness values

BS 8110-97

a_u Deflection at ultimate loading state for each column calculated from equation

a_{uav} Average deflection at ultimate loading state applied to all columns at a given level

h Depth of the cross section measured in the plane under consideration

l_e	Effective height of a column in the plane of bending considered
l_{ex}	Effective height in respect of the major axis
l_{ey}	Effective height in respect of the minor axis
l_o	Clear height between end restraints
l_c	Height of a column measured between centers of restraints
M_1	Smaller initial end moment due to design ultimate loads
M_2	Larger initial end moment due to design ultimate loads
M_i	Initial design ultimate moment in a column before allowance for additional design moments arising out of slenderness
M_x	Design ultimate moment about the x axis
M_y	Design ultimate moment about the y axis
M_{add}	Additional design ultimate moment induced by deflection of column
N	Design ultimate axial load on a column
N_{bal}	Design axial load capacity of a balanced section; for symmetrically-reinforced rectangular sections, it may be taken as $0.25 f_{cu} bd$
N_{uz}	Design ultimate capacity of a section when subjected to axial load only

EuroCode2

E_{ci}	Modulus of elasticity of concrete
f_{yd}	Yield strength of steel
f_{cd}	Strength of Concrete

Ns	Design axial force
e_{01}	Smaller value of the 1st order eccentricity of axial action effect
e_{02}	Greater value
M_{sd}	Design Moment
M_{sdx}	Design Moment about the x-axis
M_{sdy}	Design Moment about the y-axis
h	Depth of Section
λ	Slenderness ratio
l_0	Effective length
i	Radius of gyration
N_{ud}	Design Ultimate Capacity of section subjected to axial load only
N_{bal}	Design axial load applied to section to maximize its ultimate capacity
e_{tot}	Total eccentricity
e_a	Additional eccentricity
$M_{sd,1}$	Maximum design moment
λ_b	Equivalent slenderness ratio of the bracing elements
f_{ytd}	Design strength of steel in tension
f_{ytk}	Yield strength of steel

IS-456-2000

E_c	Modulus of elasticity of concrete

E_s	Modulus of elasticity of reinforcement
EI	Flexural stiffness of compression member
f_{ck}	Characteristic compressive strength of concrete
f_y	Characteristic strength of non-prestressed reinforcement
l_e	Effective length of compression member
l_u	Unsupported length of compression member
M_{ux}	Moment about x axes due to design load
M_{uy}	Moment about y axes due to design load
M_1	Smaller factored end moment on a compression member, positive if member is bent in single curvature, negative if bent in double curvature
M_{ux1}	Maximum uniaxial moment capacity for an axial load of P_u bending about x axis
M_{uy1}	Maximum uniaxial moment capacity for an axial load of P_u bending about y axis
M_{ax}	Additional Moment about x axis
M_{ay}	Additional Moment about y axis
P_b	Nominal axial load strength at balanced strain conditions
P_u	Axial load on member
r	Radius of gyration of cross section of a compression member



Chapter 2

CSICOL's User Interface

CSICOL's user interface is highly user friendly and intuitive. The interface consists of a drawing area with status bar and various menus and toolbars. Figure 2-1 shows CSICOL's main screen. The components of the interface are described in this chapter.

Drawing Area

All creation and modification of shapes and sections take place in the drawing area. CSICOL automatically updates the drawing area as shapes are added, moved, resized, rotated, and combined to create new sections of arbitrary geometry and size.

Gridlines

The drawing area is filled with a "graph paper" type grid that is useful for placing, aligning, and resizing shapes and sections. Press the **Grid** button  to open the Paper Grid Size form to edit the paper grid size or spacing.

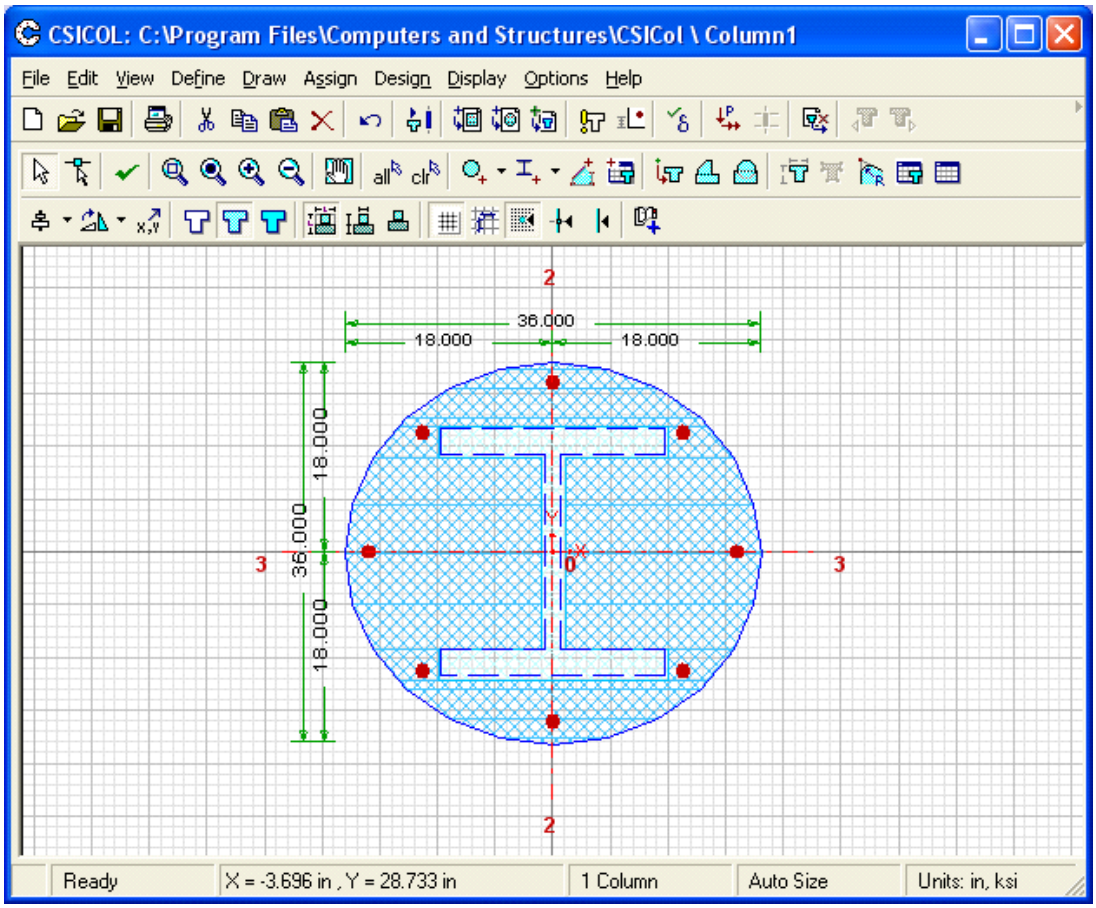


Figure 2-1: The main screen of CSICOL

Customize the grid using the **Hide/Show Gridlines** and **Snap to Grid** toggle buttons. When active, Snap to Grid will automatically restrict the movement and resizing of the shapes to the nearest grid lines.

Axis

Various editing operations, computed properties, and shape locations are referenced to the global X and Y axes. These axes help to locate the shapes properly by providing a common origin.

Two pairs of axes are used in the program. The first pair is X-Y, which is a global fixed reference on the screen. The second is the 2-3 coordinate system, which indicates the location of the section's centroid. The 2-3 axes origin keeps changing with changes in the shape's size and location. This system is provided to be compatible with the member local axis system used in the SAP2000 and ETABS programs. In those two programs, the 1-1 axis is used as the axis that passes through the members' longitudinal axis.

Status Bar

The status bar displays important information about the current section in the drawing area. The information includes the main material type, number of columns in the file, the current task and the working units.

Dimensions

When a shape is added to the drawing area, the overall dimensions of the section appear as default values. Use the **Show Section Dimensions** button  to display the overall dimensions (total width and total depth) of the section, and the **Show Shape Dimensions** button  to display the dimensions of each shape in the section. Use the **Hide Dimensions** button  to hide all dimensions.

Toolbar Buttons and Menu Commands

Almost all of the menu commands have an associated button. The buttons have been grouped into toolbars for ease of use based on type of function/task. The following table provides a graphic of the toolbar button, the menu command, and a description of the function of the button/command. The information is presented in order of the

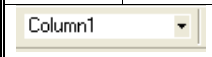
appearance of the buttons on the toolbar. Scan the second column of the table to locate menu commands. When a file menu command has no button, that command is provided following the other commands on the same menu.

Button	Command/Description
	**File menu > New Project** command. Creates a new file. The previous file will be closed and if it has not already been saved, the program will ask for the name of the file to save it before closing it.
	**File menu > Open Project** command. Opens an existing file. If the current file has not been saved, the user will be prompted to save it.
	**File menu > Save Project** command. Saves the current file. All the columns defined in the current file (work session) are saved as a single file.
	**File menu > Print Current Report** command. Prints the current report.
None	**File menu > Save Project As** command. Allows the current file to be saved using a different name.
None	**File menu > Import > Import Shape from AutoCAD DXF** command. Allows importation of section geometry from a DXF file format.
None	**File menu > Import > Import Shape from Text** command. Allows importation of section geometry from a comma separated, space separated, or tab separated text file.
None	**File menu > Export > Export Section as AudoCAD DXF** command. Exports the geometric data of the selected column section(s) to a DXF file, which can then be opened and edited using AutoCAD.

Button	Command/Description
None	**File menu > Project Information** command. Opens a window to edit general information about the current file or project. The information input using this command is included in any printed reports.
None	**File menu > Manage Columns in File** command. Accesses the Column List form, which can be used to modify the name/caption of columns and the type of column(s) in the Project file (short or long).
	**File menu > Print Current Column** command. Sends the current column section and its details for printing. The details consist of section properties, basic shape parameters, shape dimensions, column diagram, magnified moments, column framing conditions, column loads, capacity ratios, interaction curves, and the like.
	**Edit menu > Cut** command. Select a shape(s) and use the Cut command to remove the shape(s) from the working area. The cut shape(s) is temporarily stored on the Windows system Clipboard and thus can be pasted using the **Paste** command.
	**Edit menu > Copy** command. Selected shapes and sections can be copied to the Windows system Clipboard and then pasted into the working area using the **Paste** command.
	**Edit menu > Paste** command. Use the Paste command to paste the section(s) or shape(s) temporarily stored on the Windows system Clipboard into the working area.
	**Edit menu > Delete Selection** command. Removes/deletes the currently selected shape(s) or section(s) from the drawing area. The deleted object can be retrieved by clicking the **Undo** button.

Button	Command/Description
	**Edit menu > Undo command.** The program keeps track of every operation performed. Use the **Undo** command to reverse the previous action. Unlimited Undo back to the last time the file was saved is available.
	**File menu > New Column Using Quick Design** command. Activates the CSICOL Quick Design Wizard. Use the Wizard to easily define geometric properties, material properties, rebar layout, column shape, load combinations and framing conditions and to obtain results.
	**File menu > New Rectangular Column** command. Accesses the Rectangular Column form. Use the form to define a rectangular-shaped cross-section with various rebar layouts.
	**File menu > New Circular Column** command. Accesses the Circular Column form. Use the form to define the column cross-section dimensions and rebar layout for a circular cross-section.
	**File/ menu > New General Column** command. Adds a blank drawing area to the current file. Use the other commands on the main menu to add and edits shapes; define material properties, column framing, and loading; and specify rebar size and distribution to create the column and generate output.
	**Define menu > Material Properties** command. Opens the Column Material Parameters form. Use the form to define or modify the material properties for the base material (global or reference) for the column. Units on this form are in accordance with those selected using the **Options menu > General Options** command.

Button	Command/Description
	**Define menu > Confinement and Cover** command. Accesses the Confinement and Cover form. Use the form to specify the clear cover to the longitudinal rebar and specify the transverse reinforcement type as tied or spiral. The clear cover can be set for each shape in a section.
	**Define menu > Consider Slenderness Effects** command. Enable to specify that a column is slender. Disable to specify that a column is not slender.
	**Assign menu > Column Loads** command. Accesses the Column Loads or Column Loads: Simple Mode form. When slenderness effects are considered, use the Column Loads form to define detailed column load combinations consisting of a sway and a non-sway component. Different load combinations can be specified in the XZ and YZ directions. When slenderness effects are ***not*** considered, use the Column Loads: Simple Mode form to specify simple mode load combinations in terms of axial load, and top and bottom moments in the X and Y directions. Any number of combinations may be defined for the column section. CSICOL will automatically display the appropriate form based on the selection made using the **Define menu > Consider Slenderness Effects** command or associated toolbar button.
	**Assign menu > Column Framing Conditions** command. Accesses the Column Framing Conditions form. Use the form to specify framing conditions for the column in the XZ and YZ planes (different framing conditions may be specified in the two planes). The effective length factor 'k' is computed automatically using the framing conditions specified.
	**Remove Current Column from File** - button only. Removes the current section (shown in the current section drop-down list in the toolbar) from the file.

Button	Command/Description
	**Previous Column** – button only. Displays the previous section in the current file in the drawing area. This command is available only when more than one section is in a file.
	**Next Column** – button only. Displays the next section in the current file in the drawing area. This command is active only when more than one section is in a file.
	The current section's selection drop-down list. The caption/name of the section may be changed using the Column Material Parameter form.
	**Design menu > Column Capacity Ratios** command. Accesses the Capacity Calculation Result form where the results for the analysis of the current column section are displayed.
	**Design menu > Column Auto Design** command. Accesses the Auto Column Design form, where you can set the design options and complete the interactive design of columns.
	**Display menu > Show Geometric Properties** command. Calculates and displays the Geometric Properties of the column section.
	**Display menu > Show Interaction Diagrams** command. Activates the Interaction Diagrams form, which can be used to view the Interaction Capacity curves and surface for the column section.
	**Display menu > Show Moment-Curvature Curve** command. Accesses the Moment-Curvature Diagram generator and viewer window. This window also provides several options for generating the moment-curvature curves for a given failure criterion.

Button	Command/Description
	**Display menu > Show Cross-Section Stresses** command. Accesses the Stress Viewer where you can view in 2D and 3D the Elastic, Cracked and Rebar Stresses along with the Neutral Axis and Load Point location on the Section.
	**File menu > Report Creation Wizard** command. Starts the Report Creation Wizard, which guides you in a step-by-step manner to generate and preview reports.
	**File menu > Preview Current Report** command. Displays the print-preview of the current report.
	**File menu > Print Current Report** command. Sends the current column output for printing.
	**Help menu** command. Provides options for comprehensive HTML help that explains how to use the software effectively and get technical support. The options for sending email and connecting to the web site require Internet connection. The Content option shows the main HTML help topics organized in a systematic manner for searching the information on the topic of interest.
	**Draw menu > Select Pointer** command. Activates the selection mode. While in selection mode, shapes can be moved, aligned and edited using the various Draw buttons and menu commands or keyboard input.
	**Draw menu > Reshape Pointer** command. Activates the **Reshaper** tool. With this tool active, shapes can be moved and graphically resized using the mouse. Only one shape at a time can be reshaped.
	**View menu > Update Display** command. Refreshes the view in the working area.

Button	Command/Description
	**View menu > Rubber Band Zoom** command. Allows you to zoom in on the model by windowing. To use the command, depress and hold down the left mouse button. While keeping the left button depressed, drag the mouse to "rubber band" a window around that portion of the model of interest. The rubber band window that shows the extent the mouse has been dragged appears as a dashed line on the screen. When the left mouse button is released, the new view is displayed.
	**View menu > Restore Full View** command. Restores the default view of the model after the **Zoom In One Step** or **Zoom Out One Step** command has been used. The entire section is visible, showing all shapes in the drawing area.
	**View menu > Zoom In One Step** command. Allows the user to zoom in on the section, which displays more detail.
	**View menu > Zoom Out One Step** command. Allows the user to zoom out on the section to see more of it in the view.
	**View menu > Pan** command. The pan feature moves a view within the window such that you can see beyond the original edges of the view. Panning may be used in conjunction with the zoom in features.
	**Select All** – button only. Selects all the shapes currently displayed in the drawing area.
	**Clear Selection** – button only. Deselects or clears the selection of the currently selected shapes in the drawing area.
	**Draw menu > Add Basic Concrete Shape** command. Allows you to quickly select and add commonly used basic concrete shapes (rectangular, tee, circular, hollow, and so forth), and also provides access to other shape libraries.

Button	Command/Description
	**Draw menu > Add Basic Steel Shape** command. Allows quick selection and addition of commonly used basic steel shapes (I, C, T, L, and so forth); also provides access to other shape libraries.
	**Draw menu > Draw Shape** command. Changes the selection mode to shape drawing mode. Use this tool to draw the nodal point of any polygonal shape by left-clicking on the drawing area (double click to end). The program does not allow shapes with intersecting/crossing lines.
	**Draw menu > Add Shape by Coordinates** command. Opens the Define Shape by Points form where numeric input of shape coordinates can be used to define shapes. The coordinates can also be imported from or exported to other applications.
	**Draw menu > Add Shape From Library** command. Opens the View and Select Shapes from Library form where shapes can be selected from the available list of basic concrete/steel, standard steel database, and other shapes.
	**Draw menu > Merge Two Shapes > Merge Using Meshing Logic** command. Merges two selected shapes (overlapping or with a common boundary) using meshing. The overlapping mesh is removed when the shapes are merged.
	**Edit menu > Merge Two Shapes > Merge Using Intersection Logic** command. Merges two selected shapes (overlapping or with a common boundary) using outline. The lines within the shape boundary are neglected and removed after merging.
	**Edit menu > Edit Current Shape** command. Opens the Shape Editor for the currently selected shape. The Shape Editor is used to modify the dimensions and properties of a selected shape.

Button	Command/Description
	**Edit menu > Reshape Current Shape** command. Shows the node point of the current Shape for relocating with the mouse. If the current shape is a standard database shape, the program will prompt the user to convert the shape to a user-drawn shape.
	**Edit menu > Add Fillets to Shapes** command. Add fillets to the selected shape at specified nodal points. Fillets of user-specified radius can be inserted.
	**Edit menu > Edit Shape Point Coordinates** command. Opens the Define Shape by Points form, displaying the X and Y coordinates of the selected Shape. Use the form to modify, add, and delete the displayed coordinates. The coordinates can be exported to or imported from other applications.
	**Edit menu > Locate Shapes** command. Activates the Shape Layout Editor where the parameters defining the position (distance of the centroid of the shapes with respect to the origin of the Global Axis System and orientation) of a shape in the drawing area can be edited numerically by specifying coordinates and angles.
	**Edit menu > Align and Stack** commands. Allows changes to the Shape alignment (top, left, and so forth) and also provides tools for stacking one Shape above another. Several alignment and stacking options are available.
	**Edit menu > Align and Stack > Align Left** command. Aligns the selected shapes along one shape's left edge. The alignment is accomplished with respect to the least value of the x-coordinate of the first selected shape.
	**Edit menu > Align and Stack > Align Center** command. Aligns the selected shapes along the vertical centerline. The alignment is accomplished with respect to the x-centroid of the first selected Shape.

Button	Command/Description
	**Edit menu > Align and Stack > Align Right** command. Aligns the selected shapes along one shape's right edge. The alignment is accomplished with respect to the highest value of the x-coordinate of the first selected shape.
	**Edit menu > Align and Stack > Align Top** command. Aligns the selected shapes along one shape's top edge. The alignment is accomplished with respect to the maximum value of the y-coordinate of the first selected shape.
	**Edit menu > Align and Stack > Align Middle** command. Aligns the selected shapes along the horizontal centerline. The alignment is accomplished with respect to the y-centroid of the first selected shape.
	**Edit menu > Align and Stack > Align Bottom** command. Aligns the selected shapes along one shape's bottom edge. The alignment is accomplished with respect to the lowest value of y-coordinate of the first selected shape.
	**Edit menu > Align and Stack > Stack Vertically** command. Stacks the selected shapes vertically, one on top of the other. The x-coordinates of the shapes do not change.
	**Edit menu > Align and Stack > Stack Horizontally** command. Stacks the selected shapes horizontally, side-by-side. The y-coordinates of the shapes do not change.
	**Edit menu > Rotate and Flip Shape** commands. Use the options to rotate a shape by 90 degrees (clockwise or counter-clockwise) and to flip a Shape (about the vertical or horizontal axis). Several options are available.
	**Edit menu > Rotate and Flip Shapes > Rotate Right** command. Rotates the selected shape by 90 degrees in the clockwise direction.

Button	Command/Description
	**Edit menu > Rotate and Flip Shapes > Rotate Right** command. Rotates the selected shape by 90 degrees in the counter-clockwise direction.
	**Edit menu > Rotate and Flip Shapes > Flip Horizontal** command. Flips the selected shape on the horizontal axis (mirror).
	**Edit menu > Rotate and Flip Shapes > Flip Vertical** command. Flips the selected shape on the vertical axis (mirror).
	**Edit menu > Move Selection** command. Selected shapes can be displaced through a specified distance using this command. Multiple shapes can be moved simultaneously.
	**View menu > Show Shape Outline Only** command. Displays only the outline of the shapes in a section. The outline color can be changed using the **Options menu > View Options** command.
	**View menu > Show Filled Shape** command. Displays shapes in the drawing area using hatched filling. The fill color can be changed using the **Options menu > View Options** command.
	**View menu > Show Solid Filled Shape** command. Displays shapes in the drawing area using solid filling. The fill color can be changed using the **Options menu > View Options** command.
	**View menu > Show Overall Dimensions** command. Displays the over all dimensions of the section, along with the distance of the plastic centroid from the extremities of the section.

Button	Command/Description
	**View menu > Show Shape Dimensions** command. Displays the dimensions of the individual shapes in a section. This option is not available if the shapes are displayed as filled objects.
	**View menu > Turn Off Dimensions** command. Removes all dimensioning from the displayed shapes on the section.
	**Show/Hide Grid** – button only. Toggles the display of grids on and off.
	**View menu > Set Grid** command. Opens the Paper Grid Size form. The spacing of the major and minor grids can be edited here.
	**Draw menu > Snaps > Snap to Grid** command. If Snap to Grid is enabled, the mouse makes only discrete movements, which are defined by simple multiples of the grid. This affects the moving and resizing of shapes. For more precise control, modify the grid sizes, switch off the Snap to Grid option, or use the Shape Layout Editor. The left, right, up and down arrows on the keyboard can also be used to relocate the shapes.
	**Draw menu > Snaps > Snaps to Shape Points** command. Snaps to shape points to assist in aligning shapes with common nodes.
	**Draw menu > Snaps > Snaps to Shape Lines** command. Snaps to shape's lines to assist aligning of shapes with common edges.
	**File menu > Add Section View to Report** command. Adds the currently displayed graphic view to the current report, which can be customized, printed and previewed later.

Button	Command/Description
None	**Options menu > General Options** command. Sets options for working units and design codes. Also provides an option for turning off the default display of the Quick Design Wizard.
None	**Options menu > View Options** command. Sets options for display colors, drawing sizes, and automatic updating of drawing areas.
None	**Options menu > Section Capacity Options** command. Sets options that affect the Interaction Surface and Curve generation.
None	**Options menu > 3D View Options** command. Specify the settings for displaying three-dimensional views, including cross-section stresses and interaction surfaces.

In addition to the toolbar described in the table, CSCOL also provides context-sensitive toolbars that can be activated by right-clicking in the drawing area. This toolbar appears as a drop-down menu and its contents depend on the current status of the program. By default it contains adding and drawing shape options, along with copy and cut tools. If a shape is already copied to the Windows system Clipboard, the context toolbar will also show a paste tool. If a shape is selected and the right mouse is clicked, options related to editing and aligning shapes are shown.

Customize the Work Space

CSICOL provides a number of features and options to customize the workspace, color displays, interaction curves and surface generation, and default and initialization values. These options have been placed under the Options menu.

General Options

The parameters that affect all of the sections globally in the current file (i.e., working units and design code) can be set using the General options.

If the "Quick Design Wizard at Startup" option is unchecked, the Quick Design Wizard does not automatically appear when the program is started. Instead, the program adds a blank column to the drawing area. The blank column can then be modified using the various toolbar buttons and menu commands.

Units

Select any of the six working units. The units selected are saved with the section file.

Design Code

The standards specified in the codes are used for calculating the various parameters needed to generate the Capacity Interaction Surface and Moment-Curvature relationships. Those parameters include capacity reduction factors, cutoff values for axial compression, maximum allowable strain in concrete, and material reduction factors.

View Options

Use the **Options menu > View Options** command to specify options related to display color, drawing scales and auto refreshing on the View Tab of the Options and Preferences form.

Display Color

Different colors can be assigned to different shapes comprising a section. Color assignments apply to both on-screen display and printed output, assuming the output is generated using a color printer. The three options for defining color-coding are as follows:

- **Color Based on Shape Number.** Each shape in a section will display in a different color, regardless of the material properties assigned to the shape.
- **Color Based on Stress Strain Curve**. Each type of stress-strain curve will display in a different color. Use this option to display shapes assigned the same stress-strain curve in the same color.
- **Color Based on Material.** Each type of material (i.e., steel, concrete and hollow shapes) will display in a unique color. Use this option to display shapes assigned the same material type in the same color. Both border and fill colors can be specified for this option.

NOTE: If two or more overlapping shapes have the same Property Multiplier but different stress-strain relationship, it is possible that both shapes will display in the same color, signifying that one of the materials has superseded the other. In that case, increase or decrease the Modulus of Elasticity of either material slightly. The material with the higher Modulus of Elasticity will retain its mesh and the other material will not be shown in the area of overlap.

Drawing Scales

All shapes added from a shape library are drawn in true proportions on screen as well as when printed. Check the "Size the drawing area on screen when Reset is pressed" option to scale the drawing area to fit the Shapes.

Check the "Use fixed size of drawing area on screen" option and specify a fixed drawing area larger than necessary to create extra space around the Section or Shape drawing.

Refresh Graphic View

When the Auto Refresh option is active, the cross-section displayed in the working area will automatically be updated following any changes to the Shapes comprising the Section.

Section Capacity Options

Section capacity options affect the Interaction Surface and Curve generation, Auto Design and calculation of capacity ratios. The options include the number of points on the Interaction curve, and the number of curves for the generation of the Interaction Surface. Options for reporting Capacity Calculations are also available. The program offers four methods for reporting the capacity ratio Cr. A Cr value of less than one is safe. Figures 2-2, 2-3, and 2-4 illustrate the first three methods. Selecting the last option will report the largest value of capacity ratio calculated using the first three methods.

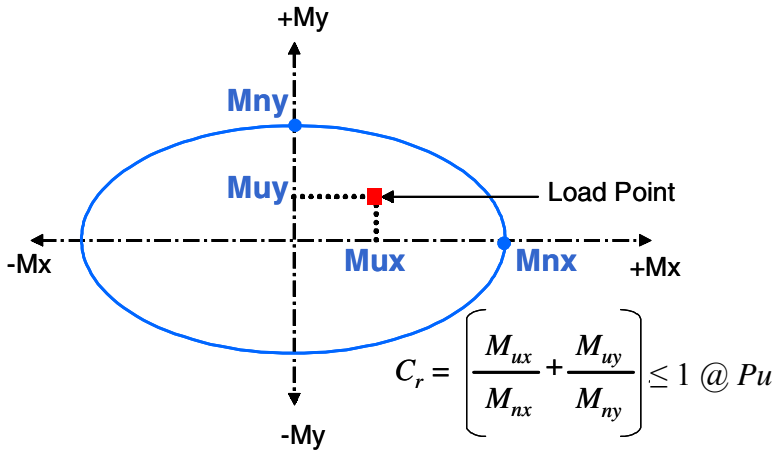


Figure 2-2:
Sum of M_x and M_y Capacities at P_u

Figure 2-3: Vector Moment Capacities at P_u

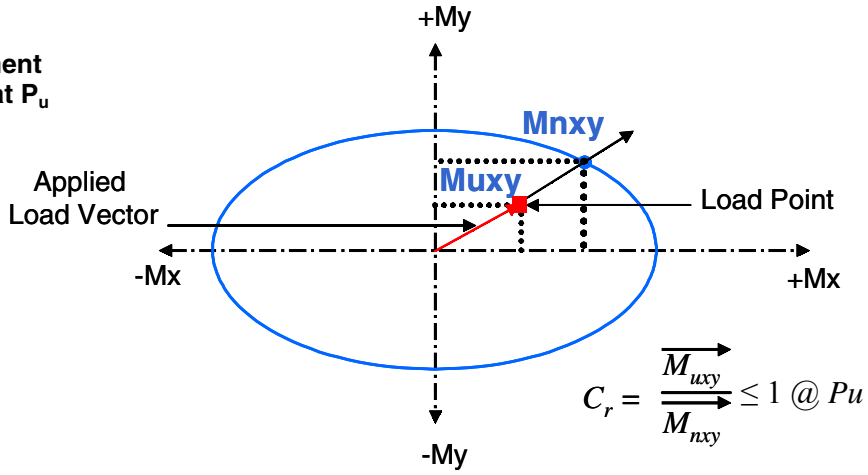
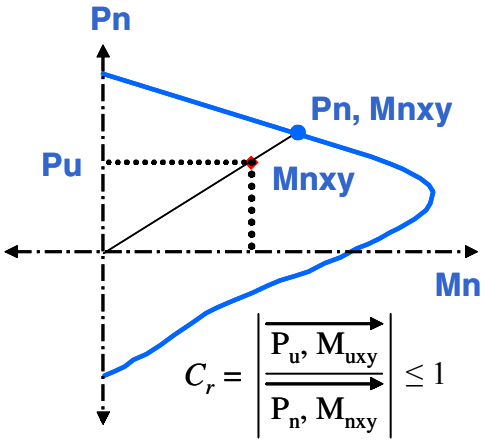


Figure 2-4: True P-M Vector Capacity



3D View Options

The 3D View Option settings specify the display for cross-section stresses and interaction surfaces. The settings appear on three tabs.

Display Objects

The Display Objects options set preferences for fill and border colors, line width, finish, and transparency for the various

elements of the display. An option is also available for specifying if the selected element is visible or not shown.

Lights

The Lights options set the preferences for lighting intensity, direction, type and color. An option is also available for turning a light source on or off.

Animation

The Animation options include rotating about the x, y and/or z axes and zooming in and out while rotating. Animation speed can be set to slow, medium or fast.



Chapter 3

Designing Columns

This chapter describes how to use CSICOL to quickly create column sections and compute their capacities. It also describes the design process. It is highly recommended that users read this chapter before using the program. It has been assumed that the user is familiar with the basic concepts of structural mechanics, structural analysis and the process of column design.

Column Design Problem

The design engineer must determine the appropriate dimensions, cross-section shape, material characteristics, and amount and distribution of reinforcement for a column on the basis of a set of applied actions, the column geometry and the framing conditions. CSICOL provides convenient tools to determine the most effective and efficient parameters for a given set of applied actions. Those tools include, among others, the Quick Design Wizard, Shape Libraries, and the Auto Section Design feature, which are described in the subsequent sections of this chapter. Tools for editing shapes in a section are described in Chapter 4.

Methods for Creating Columns

Columns can be created using the following methods:

- **File menu > New Column Using Quick Design** command, which accesses the Quick Design Wizard. The Quick Design Wizard form provides access to all of the forms needed to complete the entire column cross-section modeling and analysis process, including defining column framing and loading and generating results and reports. This is the "default process" for creating rectangular or circular columns.

- **File menu > New Rectangular Column** command, which can be used to create a rectangular column. Defining the column framing and loading and generating results must be completed independently using the various commands on the Define, Assign, Design, and Display menus. Reports for the column are generated using commands on the File menu.

- **File menu > New Circular Column** command, which can be used to create a circular column. Defining the column framing and loading and generating results must be completed independently using the various commands on the Define, Assign, Design, and Display menus. Reports for the column are generated using commands on the File menu.

- **File menu > New Column** command, which opens a blank working area that can be used to create a column of any shape. Shapes must be added to the column using the commands on the Edit and Draw menus. Defining the column framing and loading and generating results must be completed independently using the various commands on the Define, Assign, Design, and Display menus. Reports for the column are generated using commands on the File menu.

After a column section has been created using any of these methods, the shape(s) comprising the section can be edited using the tools described in Chapter 4, including adding, deleting, changing and distributing rebar.

Quick Design Wizard

The Quick Design Wizard, shown in Figure 3-1, appears when you start CSICOL.

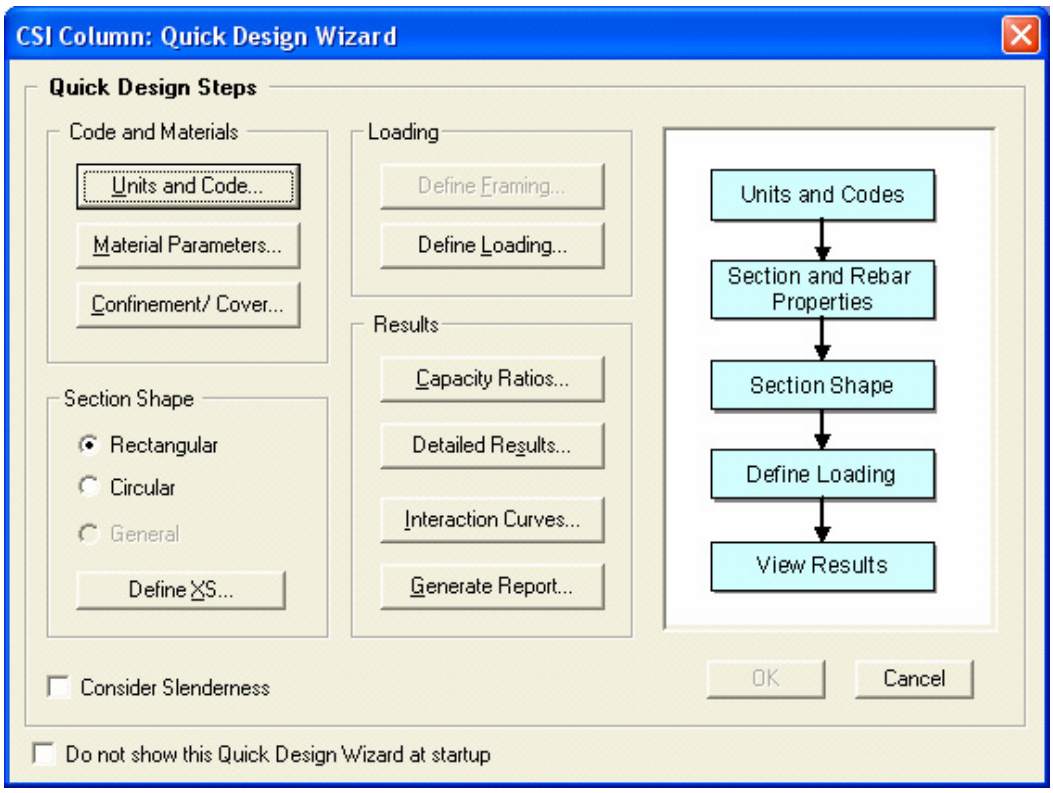


Figure 3-1: The CSICOL Quick Design Wizard

The Wizard provides access to all of the forms needed to complete the entire column cross-section modeling and analysis process. The general process involved in using the Quick Design Wizard is as follows:

1. If the Wizard is not already displayed, click the **File menu > New Column Using Quick Design** command or the **Quick Design Wizard** button to access the Quick Design Wizard form. If

the **Cancel** button on the Quick Design Wizard is clicked at any stage, all previous steps are cancelled.



Note:

If desired, turn off the Quick Design Wizard by checking the "Do not show this Quick Design Wizard at startup" checkbox. If you then decide you want to use this feature, access it using the ***File menu > New Column Using Quick Design*** *command.*

2. **Units and Code Button**. Click the **Units and Code** button to access the Options and Preferences form. Review and accept the defaults on the three tabs of the form (General, View, Section Capacity) or use the drop-down lists and edit boxes to specify the desired values. Additional information about the Options and Preferences form is provided in Chapter 2. The **Units and Code** button on the Quick Design Wizard will become inactive after it has been used to access the Options and Preferences form (and the **OK** button on that form has been clicked). If the button is inactive and changes are needed, continue using the Quick Design Wizard to create the cross-section (that is, until the section shape and loading have been defined) and then use the commands on the Options menu to access the appropriate forms to make changes to the working units or the design code.

3. **Material Parameters Button**. Click the **Material Parameters** button to access the Column Material Parameters form. Review and accept the defaults or use the drop-down lists and edit boxes to specify the desired values. See *Define the Base Material for Section* later in this chapter for more information. The **Material Parameters** button on the Quick Design Wizard will become inactive after it has been used to access the Column Material Parameters form (and the **OK** button on that form has been clicked). If the button in inactive and changes are necessary to the materials parameters definition, those changes can be made in Step 4.



Note:

After the parameters have been set and results have been generated, the Shape Editor can be used to modify the shapes in the section, including changing rebar, if necessary (see Chapter 4).

4. **Confinement and Cover Button**. Click the **Confinement and Cover** button to access the Confinement and Cover form, which has options for specifying the clear cover to longitudinal rebar in the current working units and choosing reinforcement as spiral or tied. Any rebar added to the shape will default to the parameters set on the Confinement and Cover form.

   Note: The clear cover can be set for each shape in a section. To do so, (a) complete design of the shape using the Quick Design

Wizard; (b) click the **Define menu > Confinement and Cover** command to access the Confinement and Cover form and specify new parameters; (c) double click on the shape to access the Shape Editor; (d) use the Shape Editor to delete rebar and then replace the rebar. The "replacement" rebar will default to the new clear cover setting. See Chapter 4 for information about the Shape Editor.

5. **Section Shape.** Select the shape type (Rectangular or Circular) and click the **Define XS** button to access the Rectangular Column or the Circular Column form and define the cross-section size, rebar distribution and, where applicable, the type of transverse reinforcement (Ties or Spiral). Note that the **Materials** button on those forms can be used to access the same Column Material Parameters form that was used in Step 3; use the form to modify the material parameters definition for the section if needed.

6. **Consider Slenderness Checkbox**. When this checkbox is checked, CSICOL will consider slenderness effects and column framing will need to be defined using the **Define Framing** button on the Quick Design Wizard form. When this checkbox is unchecked, CSICOL will ignore slenderness effects and you can skip Step 6 and the **Define Framing** button.

7. **Define Framing Button.** Use this button only if slenderness effects are being considered (see Step 5). Clicking this button accesses the Column Framing Conditions form. See *Specify Column Framing Conditions* later in this chapter for more information.

8. **Define Loading.** Detailed or Simple Loading may be defined. For a non-slender column, use the Simple Mode. For a slender column, use the Detailed Mode. If the Consider Slenderness checkbox at the bottom of the Quick Design Wizard form is checked, CSICOL will automatically display the form for defining the load case in the detailed mode when the **Define Loading** button is clicked. If the Consider Slenderness checkbox is unchecked, CSICOL will automatically display the form for defining the load case in the

simple mode when the **Define Loading** button is clicked. See *Specify Column Loads* later in this chapter for more information.

9. **Results.** After the column section and its loading and framing conditions have been defined, click the **Capacity Ratio** button to determine if the column section is adequate for the loading as defined. To view the detailed results, click the **Detailed Results** button. The Capacity Calculation Result form will display the detailed results for the top and bottom ends of the column, including the loading, the P-M Vector, Capacity Vector, orientation and depth of the Neutral Axis and the Capacity Ratio. If the Capacity Ratio indicates the column is inadequate (i.e., typically more than one), click the **Reset XS Step** button to reset all values to their defaults. Repeat Steps 4 through 7 to change the cross-section dimension or shape and material properties for the shape until the Capacity Ratio indicates the column is adequate (i.e., typically less than one). See Chapter 5 for more information about interpreting the results.

10. Click the **Generate Report** button to generate a printed report of the results. See Chapter 6 for more information about generating reports for sections.

File Menu > New Rectangular Column

Use the **File menu > New Rectangular Column** command to access the Rectangular Column form, which will add a new rectangular column section to the current Project file. The general process for using this method to model, analyze and report on a rectangular column is as follows:

1. If the Quick Design Wizard is active, close it by clicking the **Cancel** button or the X in the upper right-hand corner of the form.

2. Set the design codes and working units using the **Options menu > Options and Preferences** command (see Chapter 2 for more information).

3. Use the **Define menu > Confinement and Cover** command to access the Confinement and Cover form. Use the form to specify transverse lateral reinforcement as well as the clear cover for the longitudinal rebar.

4. Click the **File menu > New Rectangular Column** command to access the Rectangular Column form. Click the **Materials** button near the bottom of the form to access the Column Material Parameters form and define the column name, concrete properties, and rebar properties. See *Define the Base Material for the Section* later in this chapter for more information. The rebar properties defined are assigned to all rebar added to a particular shape. The units shown are the working units specified using the **Option menu > General Options** command.

5. **Cross-Section Size.** Accept the defaults or enter new values in the width and height edit boxes to define the size of the section.

6. **Rebar Layout.** Specify the arrangement, number and diameter of the main reinforcing bars. Click on the arrangement that best suits your requirements and type the number and diameter of bars in the Total edit box (e.g., 8#10), or click the ... button to access the Adding Rebar by List form to specify additional rebar to be distributed in accordance with the selected rebar layout. The Adding Rebar by List form is further described in the *Rebar Calculator* section in Chapter 4.

7. **Use this as Pattern for Auto Design.** When this checkbox is checked, CSICOL will consider the placement pattern of the reinforcing bars during the Column Auto Design process. The auto design process is described in the *Column Auto Design* section later in this chapter.

8. **Lateral.** Note that the transverse lateral reinforcement was set in Step 3. As appropriate, use this option to change the reinforcement for the rebar.

9. **OK button.** Click the **OK** button to close the Rectangular Column form.

10. **Consider Slenderness Effects**. If the column is being designed as slender, click the **Define menu > Consider Slenderness Effects** command or the **Consider Slenderness Effects** button  to enable this feature. If the column is not slender, skip this step and Step 12.

11. If necessary, double-click on a selected shape to edit its dimensions and other properties numerically or add, delete, or edit rebar using the Shape Editor. See the *Shape Editor* section in Chapter 4 for more information. Note that if changes are required to the clear cover, use the **Define menu > Confinement and Cover** command *before* double clicking on a shape to access the Shape Editor. The clear cover can be changed for each shape of a section.

12. If the column is slender (see Step 10), use the **Assign menu > Column Framing Conditions** command to define the framing scenario for the column cross-section. See the *Specify Column Framing Condition* section later in this chapter for more information.

13. Use the **Assign menu > Column Loads** command to define loads for the column cross-section. Detailed or Simple Loading may be defined. For a non-slender column, use the Simple Mode. For a slender column, use the Detailed Mode. See the *Specify Column Loads* section later in this chapter for more information.

14. Use the **Design menu > Column Auto Design** command to apply an iterative process to identify the smallest cross-section with minimum reinforcement that satisfies all of the loading combinations (at the top and bottom ends of the column and along both axis). See the *Column Auto Design* section later in this chapter for more information.

11. Use the various options available on the Display menu to review outputs for the column cross-section. See Chapter 5 for more information about displaying and interpreting the results.

15. Select the **File menu > Report Creation Wizard** command to generate a printed report of the results. See Chapter 6 for more information about generating reports for sections.

File Menu > New Circular Column

Use the **File menu > New Circular Column** command to access the Circular Column form, which will add a new Circular column section to the current Project file. The general process for using this method to model, analyze and report on a circular column is as follows:

1. If the Quick Design Wizard is active, close it by clicking the **Cancel** button or the X in the upper right-hand corner of the form.

2. Set the design codes and working units using the **Options menu > Options and Preferences** command (see Chapter 2 for more information).

3. Use the **Define menu > Confinement and Cover** command to access the Confinement and Cover form. Use the form to specify transverse lateral reinforcement as well as the clear cover for the longitudinal rebar.

4. Click the **File menu > New Circular Column** command to access the Circular Column form. Click the **Materials** button near the bottom of the form to access the Column Material Parameters form and define the column name, concrete properties, and rebar properties. See *Define the Base Material for the Section* later in this chapter for more information. The rebar properties defined are assigned to all rebar added to a particular shape. The units shown are the working units specified using the **Option menu > General Options** command.

5. **Cross-Section Size.** Accept the default or specify the column diameter in the current working units.

6. **Rebar Layout.** Type the number and diameter of bars in the Total edit box (e.g., 8#10), or click the ... button to access the Adding Rebar by List form to specify additional rebar. The Adding Rebar

by List form is further described in the *Rebar Calculator* section in Chapter 4.

7. **Lateral.** Note that the transverse lateral reinforcement was set in Step 3. As appropriate, use this option to change the reinforcement for the rebar.

8. **OK button.** Click the **OK** button to close the Circular Column form.

9. **Consider Slenderness Effects**. If the column is being designed as slender, click the **Define menu > Consider Slenderness Effects** command or the **Consider Slenderness Effects** button  to enable this feature. If the column is not slender, skip this step and Step 11.

10. If necessary, double-click on a selected shape to edit its dimensions and other properties numerically using the Shape Editor. See the *Shape Editor* section in Chapter 4 for more information. Note that if changes are required to the clear cover, use the **Define menu > Confinement and Cover** command *before* double clicking on a shape to access the Shape Editor. The clear cover can be changed for each shape of a section.

11. If the column is slender (see Step 8), use the **Assign menu > Column Framing Conditions** command to define the framing scenario for the column cross-section. See the *Specify Column Framing Condition* section later in this chapter for more information.

12. Use the **Assign menu > Column Loads** command to define loads for the column cross-section. Detailed or Simple Loading may be defined. For a non-slender column, use the Simple Mode. For a slender column, use the Detailed Mode. See the *Specify Column Loads* section later in this chapter for more information.

13. Use the **Design menu > Column Auto Design** command to apply an iterative process to identify the smallest cross-section with minimum reinforcement that satisfies all of the loading combinations (at the top and bottom ends of the column and along

both axis). See the *Column Auto Design* section later in this chapter for more information.

14. Use the various options available on the Display menu to review outputs for the column cross-section. See Chapter 5 for more information about displaying and interpreting the results.

15. Select the **File menu > Report Creation Wizard** command to generate a printed report of the results. See Chapter 6 for more information about generating reports for sections.

File Menu > New Column

When the **File menu > New Column** command is used, CSICOL adds a blank working area to the current Project file. The general process for using this method to model, analyze and report on a general column is as follows:

1. If the Quick Design Wizard is active, close it by clicking the **Cancel** button or the X in the upper right-hand corner of the form.

2. By default, the program will display the working units of the last saved session. Use the **Options menu > General Options** command to change the working units, if necessary (see Chapter 2 for more information).

3. Use the **Define menu > Material Properties** command or the **Column Material Parameters** button  to access the Column Material Parameters form and define the base material properties for the column section. See *Define the Base Material for the Section* later in this chapter for more information. The rebar properties defined are assigned to all rebar added to a particular shape. The units shown are the working units specified using the **Option menu > General Options** command.

4. Use the **Define menu > Confinement and Cover** command to access the Confinement and Cover form. Use the form to specify transverse lateral reinforcement as well as the clear cover for the

longitudinal rebar. The parameters set on this form are assigned to all rebar added to a particular shape.

5. Use one of the following tools to add a shape to the current column section; these tools are described later in this chapter:

   
   Add Basic Concrete Shapes button will access a selection list for adding a Basic Concrete Shape to the current section.

   
   Add Basic Steel Shapes button will access a selection list for adding a Basic Steel Shape to the current section.

   
   Add Shape From Library button opens the View and Select Library Shapes form to add a shape to the current section.

   
   Draw Closed Shape button enables the draw mode, which can be used to add a shape to the current section using left mouse clicks.

   
   Add Shape by Coordinates button accesses the Define Shape by Points form, which can be used to specify coordinate points by direct input or by importing the coordinates from an external source.



Note:

*The dimensions of shapes created suing the **Draw menu > Draw Shape** command cannot be edited using the Shape Editor because those shapes are not defined parametrically.*

6. Repeat Step 5 to add more shapes into the current column.

7. Use the mouse and commands available on the Edit menu to place the shape(s) to obtain the required section. See Chapter 4 for more information about moving and merging shapes, adding fillets or holes and the like.

8. Double-click on a selected Shape to edit its dimensions and other properties numerically using the Shape Editor and to add, delete, change, and distribute rebar for the section. See the *Shape Editor* section in Chapter 4 for more information. Note that if changes are required to the clear cover, use the **Define menu > Confinement and Cover** command *before* double clicking on a shape to access

the Shape Editor. The clear cover can be changed for each shape of a section.

9. Use the **Define menu > Confinement and Cover** command to access the Confinement and Cover form. Use the form to specify transverse lateral reinforcement as well as the clear cover for the longitudinal rebar.

10. **Consider Slenderness Effects**. If the column is being designed as slender, click the **Define menu > Consider Slenderness Effects** command or the **Consider Slenderness Effects** button  to enable this feature. If the column is not slender, skip this step and Step 11.

11. If the column is slender, use the **Assign menu > Column Framing Conditions** command to define the framing scenario for the column cross-section. See the *Specify Column Framing Condition* section later in this chapter for more information.

12. Use the **Assign menu > Column Loads** command to define loads for the column cross-section. See the *Specify Column Loads* section later in this chapter for more information.

13. Use the **Design menu > Column Auto Design** command to apply an iterative process to identify the smallest cross-section with minimum reinforcement that satisfies all of the loading combinations (at the top and bottom ends of the column and along both axis). See the *Column Auto Design* section later in this chapter for more information.

14. Use the various options available on the Display menu to review outputs for the column cross-section. See Chapter 5 for more information about displaying and interpreting the results.

15. Select the **File menu > Report Creation Wizard** command to generate a printed report of the results. See Chapter 6 for more information about generating reports for sections.

Define the Base Material for the Section

Select the **Define menu > Material Properties** command or the **Column Material Parameter** button  to access the Column Material Parameters form to define the base material for the section. The base material definition is applied to the entire column section, rather than the individual shapes in the section. Modifications to the fc', cover and stress-strain curve for each shape can be accomplished using the Shape Editor, which is described in Chapter 4.

Make selections and provide input for the following areas on the Column Material Parameters form:

- **Column Caption:** Type in a column section name. This name will be displayed in the List of Sections drop-down list on the main screen. The caption also will be displayed on the report generated for the section.

- **Concrete Type:** Select the grade of concrete for the section. Several commonly used grades of concrete have been provided (fc'=2.5 ksi to 8 ksi). Concrete type date is provided for identification and convenience only. The Fc' value can be changed independently using the Shape Editor (see Chapter 4).

- **Concrete Fc':** Specifies the ultimate compressive strength of the concrete as defined in the ACI-318 Code for the concrete type selected. For other codes, it refers to the appropriate characteristic strength or design strength. It can be changed using the Shape Editor (see Chapter 4).

- **Concrete Ec:** Specifies the standard modulus of elasticity E of the material and is used as the basis for property calculation of composite sections and stiffness calculations. For Standard concrete types, this value is calculated by the program automatically. It can be changed using the Shape Editor (see Chapter 4).

  Important Note: If the Concrete Type is one of the default material library types and the user changes the units, the program does **not** convert the Concrete Fc' and Concrete Ec to the new system of

units. Instead, the program will change the Concrete Type to the first value on the available material library list for that system of units and assign the corresponding Concrete Fc' and Ec value to the section. However, if the Concrete Type is set to Custom and the units are changed, the program will automatically change the Concrete Properties to the current system of units. The same applies to Rebar properties.

- **Rebar Type:** Select Grade-40 or Grade-60 for the Steel Reinforcing bars. The rebar f_y is modified automatically after the rebar type has been selected.
- **Rebar Fy:** Specifies the default value of the yield strength of the steel to be used in design of the main reinforcement.
- **Modulus of Elasticity:** Specifies the modulus of elasticity E for the rebar material.
- **Stress-Strain Curve:** Use the drop-down list to select the Stress-Strain curve based on the Elasto-Plastic method or the Park's Strain Hardening Model.
- **Rebar Set:** Specify the rebar using the ASTM, Metric, or Inches system, or specify user-defined properties. If the User option is selected, also click the **Edit** button to access the Edit User Rebar Set form. Click in the cells of the spreadsheet on that form and type in the revised values for rebar diameter or area. Click the **OK** button to accept the changes and return to the Column Materials Parameters form.

Add Shapes from a Library

Regardless of the method used to begin the column, shapes from the standard shape libraries can be added to a column section, for example, to create a complex column section. The tools available for merging shapes are described in Chapter 4. Add a Shape from one of the standard libraries as follows:

1. Select the **Draw menu > Add Shape from Library** command or click the **Add Shape from Library** button . Alternatively, select the Add Shape from Library option from the drop-down list after clicking the **Add Basic Concrete Shape** or **Add Basic Steel Shape** buttons. Any of these actions will open the View and Select Library Shape form.

   If no libraries are shown, click the **Add Library from File** button on the View and Select Library Shapes form to open a file selection window to select and open a library file, such as the Internal Shapes.lsd file. Click on the Internal Shapes.lsd file to access the various shape libraries, such as the Rectangular Shape, Box Shape, T Shape libraries and so on.

2. Select an appropriate library and click the **Next** button , or double click on the selected (highlighted) library icon. A window showing all the Shapes under this library will appear on the screen. For Standard Steel Shapes, it may be necessary to click the button twice to display the list of standard steel sections.

3. Select the desired shape from the list. Double click on the selected shape or click the **Add Shape and Close** button . The selected shape will be added in the middle of the drawing area.

TIP: If several shapes from the library are to be added, check the Keep Library Open checkbox to keep the form visible until you close it.

Shape Libraries

Shape libraries consist of a collection of basic shapes that can be used to create new shapes and sections. The shapes in the collection are grouped into the following categories:

- Basic Concrete Shapes
- Basic Steel Shapes
- Bridge Pier Shapes
- Standard Steel Shapes

- AISC Steel Shapes- Metric units
- AISC Steel Shapes- US units
- CISC Steel Shapes

Figure 3-2 shows the View and Select Library Shapes form. The toolbar buttons on the form are enabled or disabled depending on which level of the library is in use. For example, if the user is at the top level (only main library names are visible), the **Add Shape and Close** button  is disabled. The functions of the toolbar buttons are as follows:


Saves the current library with default or current name.


Saves the current library with a user specified name.


Saves all library files.


Moves back one level in the library


Moves down one level in the library.


Jumps to the top level (the main library list).


Adds the currently highlighted shape to the main work space and closes the View and Edit Library Shapes form.


Opens a file selection window to select and open a new library file.


Removes the currently highlighted library from the current file.


Cuts the currently selected shape and places it on the Windows system Clipboard.


Copies the currently selected shape and places it on the Windows system Clipboard.


Pastes the temporarily stored section from the Windows system Clipboard to the current library.


Prints the shapes list in the current library.

Note:

If no shapes are shown in the View and Select Library Shapes form, click the ***Add Library from File*** *button*  *and click on the files located in the "/Shape Libraries" subfolder or other library file.*

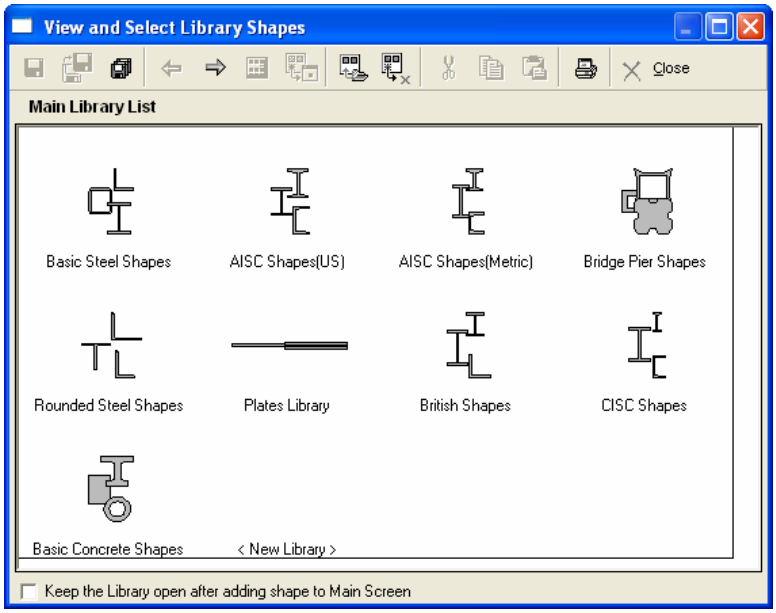


Figure 3-2: View and Select Library Shapes Form

Add Shapes by Drawing

If the shape of your preference is not available in any one of the CSICOL built-in libraries (basic concrete, basic steel, standard steel.), create the shape by drawing it on the screen. Shapes of any arbitrary geometry can be defined graphically.

1. Set the grid spacing, snapping, and other properties as appropriate to draw the new shape (refer to the *Toolbar Buttons and Menu Commands* section of Chapter 2 for more information about these options).

2. Click the **Draw Closed Shape** button  or select the **Draw menu > Draw Shape** command. Note that the cursor will change to a "+" sign.

3. Click the left mouse button on the screen at the location where you want to draw a node. Repeat the process to draw all of the nodes of the shape.

4. Double click to end the drawing.

A drawn shape can be a "hole" as follows:

a. After the shape that is to be a hole has been drawn using Steps 1 through 4, exit the draw mode and enter the select mode by clicking on the **Select** button .

b. Double click on the outline of the shape to access the Shape Editor.

c. Set the Type of Shape to Hole.

Add Shape by Importing Shape Coordinates

A Shape can be defined by specifying its nodal point coordinates using keyboard input or by importing the coordinates from an external source. Those sources include data from the Windows system clipboard, comma-separated, tab-separated or spaced text file, or a .dxf file.

1. Use the **Draw menu > Add Shape by Coordinates** menu or click on the **Add Shape by Coordinates** button  to access the Define Shape by Points form.

2. Provide the coordinate points of the shape by typing directly into spreadsheet on the Define Shape by Points form. Or use the following steps to import the coordinates from an external source:

   a. Click the **Import** button on the Define Shape by Points form to access the Import Shape Coordinates form.

   b. **File Type.** From the drop-down list, select the type of the source file to be imported.

   c. **File Name.** Click on the **Select File** button and locate the input file. The Import Shape Coordinates form will display the data in the file and a preview of the shape in the Data In File and Shape Preview areas of the form, respectively.



Note:

If the imported coordinates draw lines that intersect, the program will prompt the user to change the coordinates appropriately. Intersecting lines are not allowed in CSICOL.

d. Click the **OK** button to import the data and close the Import Shape Coordinates form and return to the Define Shapes by Points form.

- Use the **Insert** button to insert additional coordinate points.
- Use the **Remove** button to remove unwanted coordinates.
- Use the **Clear** button to delete all of the coordinate points.

The display area on the Define Shape by Points form is updated showing the sketch of the shape formed by the coordinates provided.

After data has been input on the Define Shape by Points form, the **Export** button can be used to access the Export Shape Coordinate form and export the coordinates in various formats to other programs for further processing. After processing, the data can then be imported back into the Define Shape by Points form.

a. **File Type.** From the drop-down list, select the type of format to be used in exporting the coordinate points.

b. **File Name.** Click on the **Select File** button to access the Export to Text File form and specify the path and filename for exporting the coordinates. Click the **Save** button.

c. Click the **Done** button to close the Export Shape Coordinates form and return to the Define Shapes by Points form.

3. Click the **OK** button to close the Define Shapes by Points form and return to the CSICOL drawing area.

Specify Column Framing Conditions

Use the **Assign menu > Column Framing Conditions** command when the Consider Slenderness Effects feature is enabled to access the Column Framing Conditions form and specify the information needed to compute

the magnified moments while considering slenderness effects. Figure 3-3 illustrates the Column Framing Conditions relative to the XZ and YZ planes.

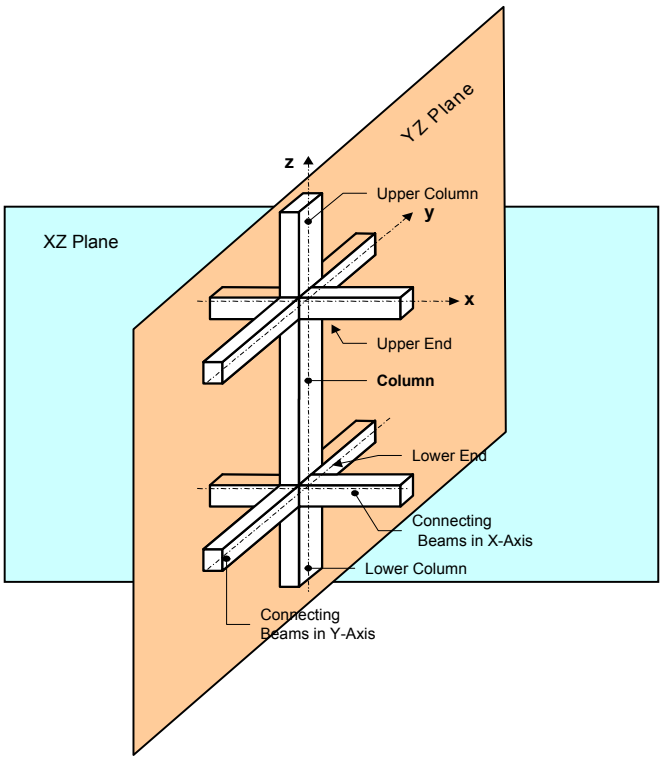


Figure 3-3: Column Framing Conditions

Clicking the **Assign menu > Column Framing Conditions** command displays the Column Framing Conditions form. A 3D view of the framing of the column under consideration is shown in the display area of that form. Use the form as follows to specify the column framing conditions.

1. **XZ Plane and YZ Plane Tabs.** Select the tab corresponding to the plane of bending for which the framing condition is being defined.

2. **Total C/C Length and Unsupported Length.** Type values for the clear height of the column and the story height in the respective edit boxes.

3. **Framing.** The framing options depend on the code selected for the project. The information required includes the clear column length and the effective length factor for both braced and unbraced conditions. CSICOL can calculate those factors based on the selected end/support conditions of the column. CSICOL also can accommodate different effective length factors for the two directions of bending (XZ plane and YZ plane). Alternatively, the effective length values can be specified through direct input.

   For BS 8110-97

   - Choose the Simplified or Detailed Approach:

     - The Simplified Approach is based on Tables 3.19 and 3.20 of the BS-8110 Part 1 (1997) code. Select the options that best represent the top and bottom end conditions of the column. The Braced β Factor and the Unbraced β Factor text boxes will be updated automatically.

     - If the Detailed Approach is selected, additional information is needed regarding the framing condition of the column. Click the graphic that best represents the column end/support condition (i.e., Lowest Story Column, fixed on one end and connecting to frame elements on the other; or Intermediate Story Column, connected to frame elements on both ends). Type the parameters in the edit boxes directly or click the ... button to access the Effective Length Factor {Framing or With Foundation} form to calculate the β Factor. The computed β Factor values are based on equations given in CP 65 Part 2 1996 (1999).

   Repeat steps 1 through 3 to calculate 'β' in the other direction.

   For All Other Codes

   - Click the graphic that best represents the column end/support condition (e.g., Pin Ends, pin supports on both ends; Propped

Cantilever, pin-fixed support; Cantilever, free-fixed support; Lowest Story Column, fixed on one end and connecting to frame elements on the other; and Intermediate Story Column, connected to frame elements on both ends).

- CSICOL calculates and reports 'k' (braced and unbraced) if the Pin Ends, Propped Cantilever, or Cantilever option was selected; accept the calculated values or type alternative values in the edit boxes. If the Lowest Story Column or Intermediate Story Column was selected, type the parameters in the edit boxes or click the ... button to access the Effective Length Factor {Framing or With Foundation} form to calculate 'k.' See the next section for more information about the effective length factor.

Repeat steps 1 through 3 to calculate 'k' in the other direction.



Note:

The Consider Slenderness Effects feature must be enabled or the ***Assign menu > Column Framing Conditions*** *command will not be available.*

Effective Length Factor

The effective length factor 'k' (for braced and unbraced conditions) can be calculated for a column based on the end/support conditions of the column. Different 'k' factors for the two directions of bending (XZ plane and YZ plane) can be provided. Figure 3-4 shows the Effective Length Factor form.

Use the **Assign menu > Column Framing Conditions** form to access the Column Framing Conditions form, select the Lowest Story Column or Intermediate Story Column framing scenario and click the ... button to access the Effective Length Factor form to calculate the 'k' factor. When the framing type has been specified as Lowest Story Column, CSICOL displays the Effective Length Factor - With Foundation form. When the framing type has been specified as Intermediate Story Column, CSICOL displays the Effective Length Factor - Frame form. Make selections on the Effective Length Factor (Frame or With Foundation) form for the following:

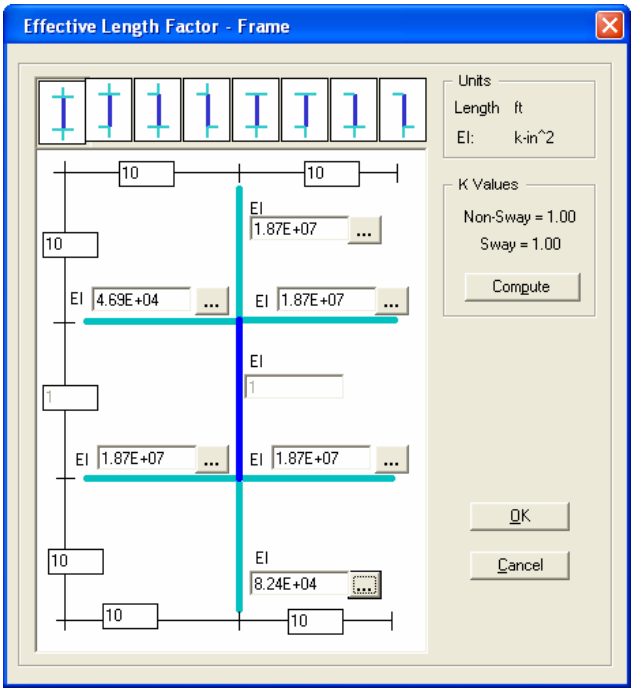


Figure 3-4: Effective Length Factor Form

1. Click the graphic along the top of the display area that reflects the framing condition for which the 'k' factor is being calculated. The blue line represents the column under consideration and the other lines the framing surrounding the column (for the Effective Length Factor - Frame form) or above the column under consideration (for the Effective Length Factor - With Foundation form). CSICOL will show the selected framing scenario in the display area of the form. The numbers displayed in outlined boxes (e.g., 10) along the edges of the display are column (vertical) and beam (horizontal) lengths. The column lengths are story-to-story heights and the beam lengths are the length to the adjacent column.

2. **EI edit boxes**. CSICOL will display default values for the stiffness of the columns above and below the column under consideration and of the beams connecting to the column. Use the defaults or click the ... button to access the EI Calculator form, which can be used to modify the column shape and rebar distribution and then recompute the effective EI for the members connected to the

column under consideration. See the next section for more information about the EI Calculator form.

3. **Compute Button.** Click the **Compute** button and CSICOL will calculate and report the 'k' values for Non-Sway and Sway conditions in the K Values area of the form.

4. Click the **OK** button and CSICOL will transfer the non-sway (braced) and sway (unbraced) 'k' values to the appropriate edit boxes on the Column Framing Conditions form.

"EI" Calculator

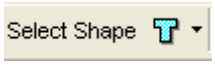
Use the EI Calculator form to specify the shape, dimensions, and rebar distribution for the member connected to the column under consideration and then calculate the associated EI factor. The EI factor is then used by the program to determine the Effective Length Factor for sway and non-sway conditions for the column under consideration.



Note:

The coordinate points of the shape shown in the display area cannot be changed. The shape dimensions can be changed.

The EI Calculator form has a graphical display area, a Properties window (the data display area on the right side of the form), shape caption and material properties area, coordinate and dimensions tabs and several toolbar buttons.

1. **Display Area.** CSICOL will display a default shape in the display area. If necessary, use the **Concrete Shape** button 

to access a drop-down list and select a new shape, which will be added to the display area. The display area is also where rebar can be added, deleted, copied, and distributed using the various toolbar buttons, which are identified later in this chapter.

2. **Properties Window.** The Properties window displays the shape and rebar data, such as height, width, area, inertia, computed and modified I22 and I33 for the shape and the area and ratio for the rebar. After making changes to the shape or rebar in the display area, click the **Accept Changes** button  and CSICOL will update data display accordingly.

3. **Shape Caption, Material Type, Sub-Material Type, Concrete Fc, Modulus E.** Use the drop-down list and edit boxes in the lower right-hand side of the form to specify a caption for the Shape and other factors such as the concrete type and fc'. This information is used to calculate the stiffness of the cross-section of the connecting member.

4. **EI Reduction Factor.** Various codes require that the calculated EI value be reduced by a certain factor to cater the effects of cracking and so forth. Use the EI Reduction Factor edit box to specify an EI reduction factor other than that specified by the code.

5. **Coordinate and Dimensions Tabs.** The coordinate points of the shape cannot be changed. Thus the Coordinates tab is for display purposes only. Click on the cells of the Dimensions tab and enter new values to change the shape's dimension.

The toolbar buttons on the EI Calculator form function in a manner similar to the function of the toolbar buttons on the Shape Editor. See the *Shape Editor* section in Chapter 4 for an explanation of the toolbar buttons.

Specify Column Loads

The two modes to define loads on the column are the simple mode and the detailed mode. Column loads can be defined in the simple mode if the column being considered is a short column (no slenderness considered); otherwise, the detailed mode should be used. Figure 3-5 illustrates loading on the column and cross-section.

Simple Loading Mode

Use the Simple Load option when the following conditions are met:

- The column is to be designed to be a short column or when slenderness effects can be ignored.

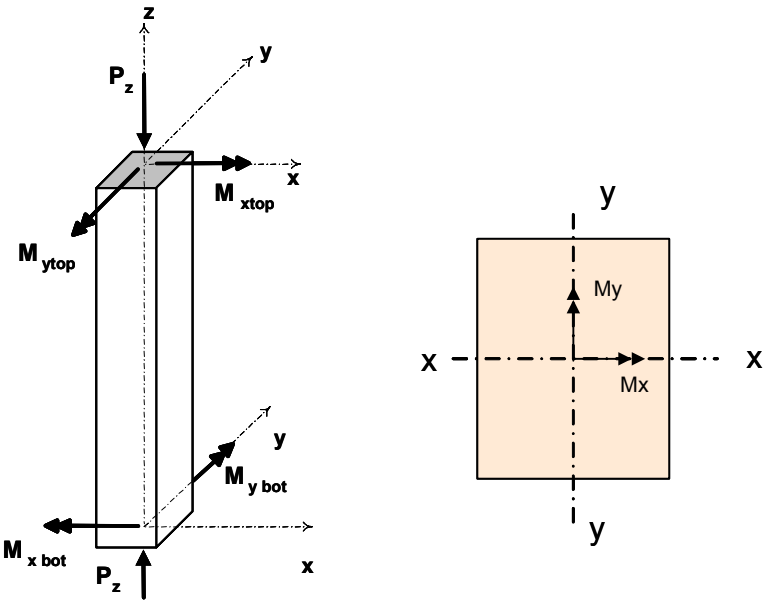


Figure 3-5: Column Loading on Column and Cross-Section

- The loads and moments have already been magnified by separate analysis, such as the P-Delta analysis option in ETABS or by other procedures.
- Detailed loading or column framing information is not available.

For the simple load combination case, specify the combination name, axial load value, $M_{x,top}$ (moment about the x-axis at the top end of the column), $M_{x,bot}$ (moment about the x-axis at the bottom end of the column), $M_{y,top}$ (moment about the y-axis at the top end of the column) and $M_{y,bot}$ (moment about the y-axis at the bottom end of the column). Figure 3-5 shows the loading on a column.

Define simple loading on the column as follows:

1. Define the column section (geometry, rebar and material properties) as described in the previous sections in this chapter. Use the **Assign menu > Consider Slenderness Effects** command to ensure that slenderness effects are ***not*** considered.

2. Use the **Assign menu > Column Loads** command or click the **Column Loads** button  to access the Column Load: Simple Mode form. Use that form to specify the load combination name, axial load and the four moments ($M_{x,top}$, $M_{x,bot}$, $M_{y,top}$, $M_{y,bot}$). Specify as many load combinations as required for analysis and design.

   - Use the **Import** button to import load combinations from previously saved text files or from data on the Windows system clipboard. This will activate the Import Design Loads from File form.

     a. **File Type.** Select the file type from which you want to import data.

        If the Data in System Clipboard option is selected, CSICOL will copy the data from the system clipboard onto the Import Design Loads from File form, in which case, skip b and c and proceed to Step 3.

     b. **File Name.** Use the **Select File** button to locate the file.

     c. Click the **OK** button on this form to import the data to the Column Load: Simple Mode form.

3. Click the **OK** button on the Column Load: Simple Mode form to return to the drawing area.



Note:

Right clicking on the cells of the Column Load: Simple Mode form will display context-sensitive toolbars with options for selecting, copying, cutting, pasting and clearing the selection.

Detailed Loading Mode

Unlimited combinations can be defined for detailed loads for a column. In addition, load combinations can be defined in the XZ and YZ planes, separately. Detailed Loads are needed only if slenderness effects need to be considered. To specify the final design loads directly, use the Simple Loading Mode option defined in the previous section.

Detailed Loads-ACI 318

Define a detailed load combination in accordance with the ACI-318 design code as follows:

1. Define the column section (geometry, rebar and material properties) as described in the previous sections in this chapter. Use the **Assign menu > Consider Slenderness Effects** command or the **Consider Slenderness** button to ensure that slenderness effects are considered.

2. Select the **Assign menu > Column Load** command or click the **Column Load** button to access the Column Load form.



Note:

Right clicking on the cells of the Column Load form will display context-sensitive toolbars with options for selecting, copying, cutting, pasting and clearing the selection.

3. On the Column Load form, select the loading direction (Along X or Along Y) by clicking on the respective tab and then specify the combination name.

4. Specify the axial load, top moment, and bottom moment for the sway and non-sway conditions. Input Sway condition data only if the Consider As Sway option is checked at the bottom of the form.

5. Specify story shear, story load and critical load for sway condition. This data is used to calculate the magnification factor.

6. Click the **Check Sway Condition** button to check for sway conditions. This will activate the Sway Load Combination Check form (see the next section for more information). The results of the sway check form will automatically set the appropriate options on the main form.

- Use the **New** button to add another load combination.

- Use the **Modify** and **Delete** buttons to modify and delete existing load combinations respectively.

Sway Load Combination Check

Click the **Check Sway Condition** button on the Column Load form to access the Sway Load Combination Check form. The sway check is performed based on the following three criteria:

- **Stability Index.** This test or check is performed for the entire story. If the Stability Index, **Q,** of a story is less than or equal to 0.05, the story may be assumed to be non-sway. The data required to perform

this test includes the sum of the factored axial load of the story, relative lateral deflection between the top and bottom of that story, the shear causing the lateral deflection, and the clear length of the column member. The program also reports the stability index as computed based on the input data. If data has already been entered in the sway portion of the Detailed Column Load form, those data are displayed here. Otherwise, the data entered for this option is transferred to the Detailed Load form in the sway part of the load combination.

- **Second Order Analysis Results.** If the increase in the end moments of a column caused by second-order effects does not exceed 5% of the first-order end moments, the column in the structure will be considered to be non-sway. The input required is the end moments for the first and second-order analysis results. The program reports the percentage difference for both end moments obtained by the first and second-order analyses.

- **Relative Stiffness of Bracing to Column.** A column may be assumed to be non-sway if it is located in a story in which the bracing elements (shear walls, shear trusses, and other types of lateral bracing) have such substantial lateral stiffness to resist the lateral deflection of the story, so that any resulting lateral deflection is too small to affect the column strength substantially. The input required is the sum of lateral stiffness of all elements bracing the column and the lateral stiffness of the column itself.

After providing the required input data, click the **Check** button. The program will report if the column or story under consideration is sway or non-sway. Closing the form will update the data on the Detailed Column Load form. The Sway Load Combination Check form is shown in Figure 3-6.

Detailed Loads-BS 8110-97

Define detailed loading in accordance with the BS-8110 design code as follows:

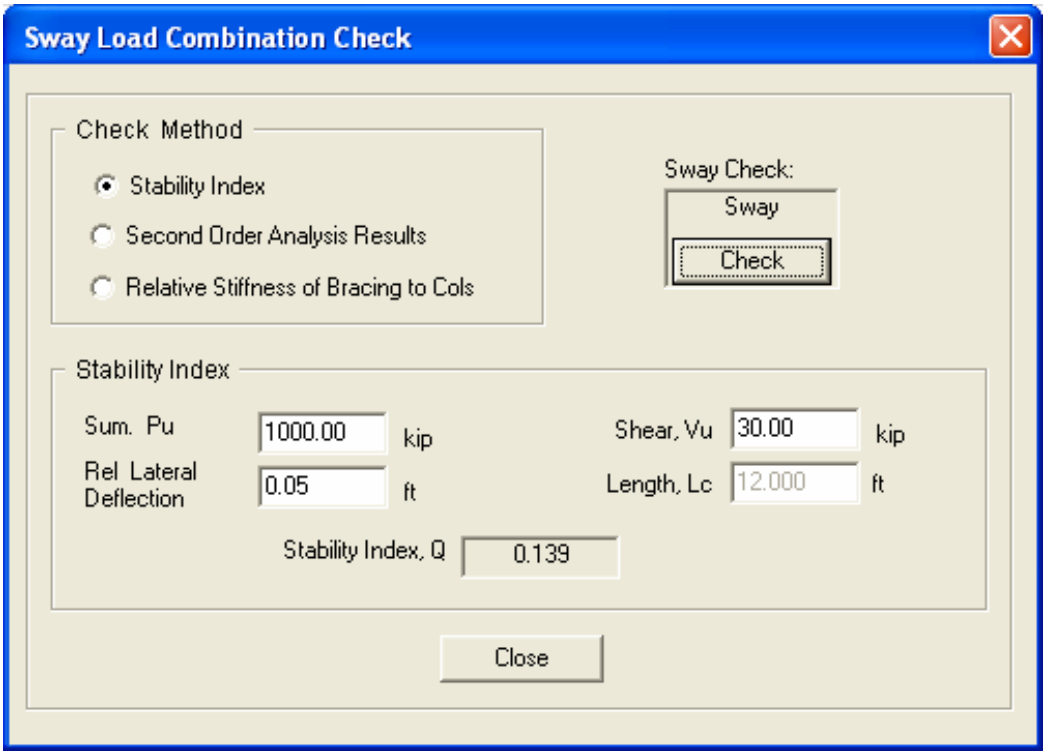


Figure 3-6: Effective Length Factor form

1. Define the column section (geometry, rebar, and material properties) as described earlier in this Chapter. Use the **Assign menu > Consider Slenderness** command or the **Consider Slenderness** button to include slenderness effects.

2. Select the **Assign menu > Column Load** command or click the **Column Load** button to access the Column Load form.

3. Select the loading direction (Along X or Along Y) by clicking the respective tab and then specify the combination name.

4. Specify the axial load, bottom moment, and top moment values. By default the column is considered to be braced in the selected direction. Check the Consider as Un-braced check box to change this default assumption.

5. Specify the average deflection at ultimate loading state applied to columns (a_u) after selecting the Use Average au check box.

- Use the **New** button to add another load combination.
- Use the **Modify** and **Delete** buttons to modify and delete the existing combinations respectively.
- Use the **Copy from X** or Copy **from Y** buttons to copy the load combinations from one loading direction to the other.

Detailed Loads-EuroCode2

Define detailed loading in accordance with the EuroCode2 design code as follows:

1. Define the column section (geometry, rebar, and material properties) as described in earlier in this Chapter. Use the **Assign menu > Consider Slenderness** command or the **Consider Slenderness** button  to include slenderness effects.

2. Select the **Assign menu > Column Load** command or click on the **Column Load** button  to access the Column Load form.

3. Select the loading direction (Along X or Along Y) by clicking the respective tab and specifying the combination name.

4. Specify the axial load, bottom moment, top moment and sustained load values. By default the column is considered to be braced in the selected direction. Check the Consider as Un-braced check box to change this default assumption.

- Use the **New** button to add another load combination.
- Use the **Modify** and **Delete** buttons to modify and delete the existing combinations respectively.
- Use the **Copy from X** or **Copy from Y** buttons to copy the load combinations from one loading direction to the other.

Detailed Loads-IS 456-2000

Define the detailed loading in accordance with the IS-456-2000 design code as follows:

1. Define the column section (geometry, rebar, and material properties) as described in earlier in this Chapter. Use the **Assign menu > Consider Slenderness** command (or **Consider Slenderness** button) to include slenderness effects.

2. Select the **Assign menu > Column Load** command or click the **Column Load** button to access the Column Load form.

3. Select the loading direction (Along X or Along Y) by clicking the respective tab, and specify the combination name.

4. Specify the axial load, bottom moment, and top moment values. By default the column is considered to be braced in the selected direction. Check the Consider as Un-braced check box to change this default assumption.

- Use the **New** button to add another load combination.
- Use the **Modify** and **Delete** buttons to modify and delete the existing combinations respectively.
- Use the **Copy from X** or **Copy from Y** buttons to copy the load combinations from one loading direction to the other.

Column Auto Design

The Column Auto Design feature is a very effective, efficient and powerful tool for the design of sway and non-sway columns with or without considering slenderness effects. Use the Column Auto Design feature to design the column cross-section in accordance with the various design parameters defined/specified by the user.

Use the Column Auto Design feature to find the smallest cross-section with minimum reinforcement that satisfies all of the loading combinations (both at the top and bottom ends of the column and along

both axes). The Column Auto Design process can include the evaluation of slenderness effects during each iteration. Column Auto Design is performed based on rules specified by the user. Figure 3-7 shows the Auto Design Cross-Section form as iterations are completed during the design process.

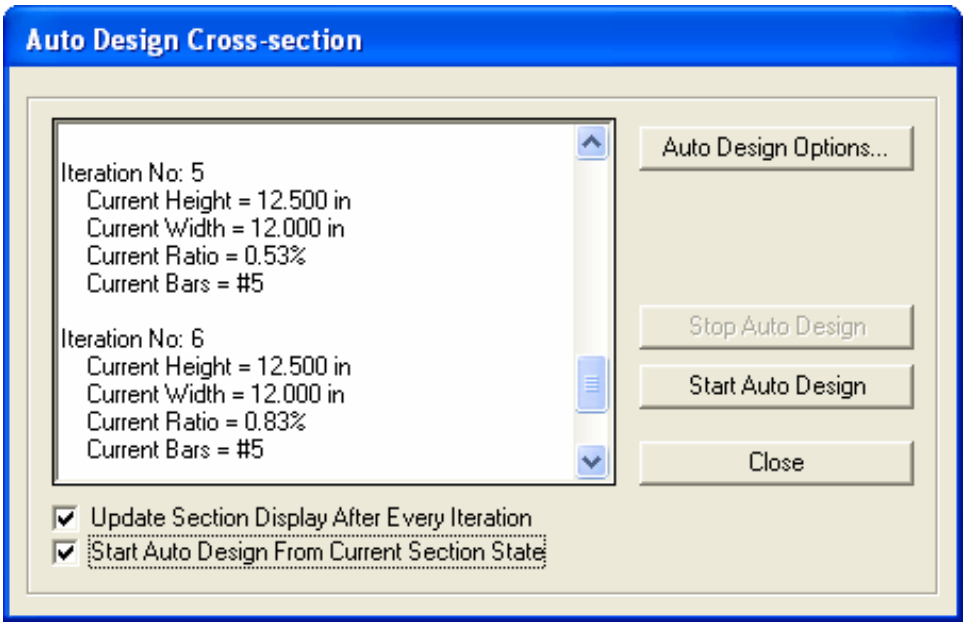


Figure 3-7: Iterations during the Design of Column Sections

The Column Auto Design tool designs the column section in accordance with the various design parameters defined/specified by the user. Use the Column Auto Design feature as follows:

1. After the section has been defined (both geometry and material properties) and the rebar location has been specified (when using the **File menu > New Column** command, see the description of the Shape Editor in Chapter 4 for an explanation of adding rebar to the section), access the Auto Design Cross-Section form using the **Design menu > Column Auto Design** command of the **Column Auto Design** button .

2. The Auto Design Cross-Section form has the following areas:

- **Result Display Area:** All of the iterations completed in accordance with the specifications are shown in this area. The information displayed includes the Iteration Number, current column cross-section height and width, current rebar ratio and current bar size. Those parameters may or may not change depending on the Auto Design Options specified. If the design is satisfied by those parameters, the design process is terminated and a Design Completed message is displayed on the screen.

- **Auto Design Options:** Click the **Auto Design Options** button to activate the Auto Design Options form, which is described in the next section.

- **Stop/Start Auto Design:** These buttons start or stop the auto design process. The design process can be terminated at any time during the iteration by clicking the **Stop Auto Design** button. This process can be continued from the current section state by checking the "Start Auto Design from the Current Section State" checkbox and clicking the **Start Auto Design** button. If the "Start Auto Design from the Current Section State" checkbox is unchecked, the design process will start from the initial conditions specified on the Auto Design Options form when the **Start Auto Design** button is clicked.

- **Update Section Display After Every Iteration:** If this checkbox is checked, the program will update the section displayed in the main CSICOL work area after each iteration. If this checkbox is unchecked, the program will not update the display even after the design process has been completed. In that case, click the **Done** button and CSICOL will then update the display of the section in the main CSICOL work area.

Auto Design Options

Clicking the **Auto Section Design Options** button on the Auto Design Cross-Section form will access the Auto Design Options form. Use the form to specify the various criteria for the auto column design process. The criteria determine how the iterations will be formulated. The

iterations may change depending on rebar size, rebar ratio, section width, or section height. The Auto Design Options Form is shown in Figure 3-8.

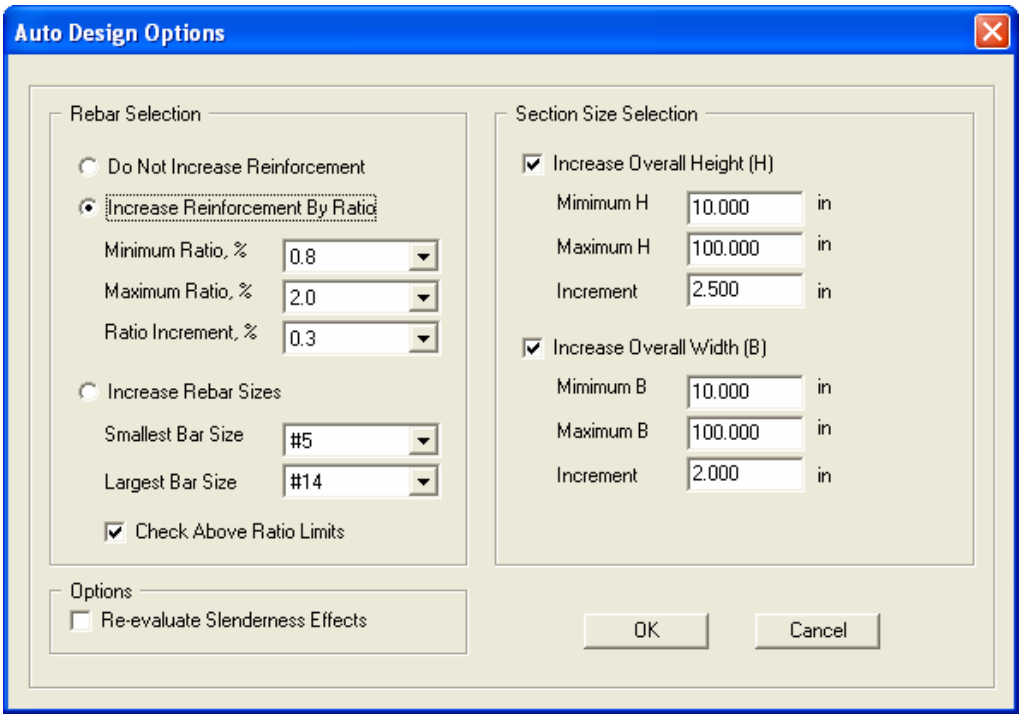


Figure 3-8: Auto Design Options

- **Rebar Selection:** Choose one of the three options provided. The program will design the column, keeping the rebar ratio and area to a minimum. The options include the following:

  - **Do Not Increase Reinforcement:** Select this option if you do not want to change the area of reinforcement provided in the column section. The program will only change the column section size within the maximum and minimum limits specified to achieve the required capacity.

  - **Increase Reinforcement Ratio:** Select this option to obtain the required section design by increasing the rebar ratio within the minimum and maximum limits specified. The program will start the design using the minimum rebar ratio and then increase the



Note:

The program does not check for minimum or maximum rebar ratio as specified in the selected code. To ensure that the design is within the code-specified limits, select the code-specified reinforcement ratio limit using the Increase Reinforcement Ratio and check the Check Above Ratio Limits option on the Auto Design Options form.

ratio by an amount specified in the Ratio Increment drop-down box for each iteration. The rebar ratios may be specified from 0.4% to 10% of the cross-section area. The increment step may range from 0.1% to 0.9%. The program does not check for minimum or maximum rebar ratios as specified in the selected code. To ensure that the design is within the code's specified limits, select the code specified reinforcement ratio limit using the Increase Reinforcement by Ratio option and check the Check Above Ratio Limits option.

- **Increase Rebar Sizes:** Use this option to complete design within a range of rebar sizes. Specify the smallest and largest rebar sizes. Select the Check Above Ratio Limits option to ensure that the ratio is within required limits. The program will then also check the rebar ratio after each iteration. If this option is not selected, the program will only design the column section based on the smallest and largest bar size, regardless of the rebar ratio.

- **Section Size Selection:** Specify the range for the column section size. The program will design the column, keeping the cross-section size to a minimum. The following options are available:

  - **Increase Overall Height:** Specify the minimum and maximum height, and the increment for the cross-section height. The program starts the design with the minimum height and increases it by an amount equal to the increment provided in each iteration until the maximum height is reached or the desired capacity is obtained. The height and width of the section are not changed unless the rebar ratios or rebar sizes within the limits have been checked and found insufficient.

  - **Increase Overall Width:** Specify the minimum and maximum width, and the increment for the cross-section. The program starts the design with the minimum width and increases it by an amount equal to the increment provided in each iteration until the maximum width is reached or the desired capacity is obtained.

- **Re-Evaluate Slenderness Effects:** Select this option to recalculate the slenderness parameters based on the new column section size and reinforcement after each iteration.

CISCOL®

CSI®

Chapter 4

Edit Columns Cross-Sections

This chapter describes how to use the various tools available in CSICOL to edit shapes and add, edit and distribute rebar in a section.

Types of Shapes

After a Shape has been added to a section, its dimensions and properties can be modified. However, before attempting to modify a Shape, refer to Table 4-1 to determine the appropriate editing operation because the original source of the shape determines which editing operations can be used to edit the shape.

- **DB Shapes and Parametric Shapes**: These classes include the basic steel/concrete shapes defined parametrically and the standard steel shapes obtained from the database. In this class of shapes, the user adds the shape using the Quick Design Wizard, the **File menu > New Rectangular Column** or **File menu > New Circular Column** command, or selects the shapes from one of the available shape libraries, adding it to the current section without any modification.

Table 4-1 Shape Editing Methods

Editing Operation Type	DB Shapes	Parametric Shapes	User Drawn Shapes	Shapes Created by Merging	Holes
Use Shape Layout Editor to change parametric dimensions	N/A	OK	N/A	N/A	Depends on the source shape
Use the mouse and resizing handles to resize the overall shape	N/A	OK	OK	OK	
Use the mouse to move nodes/change node coordinates	N/A	N/A	OK	OK	
Use Shape Editor to change coordinate values	N/A	N/A	OK	OK	

Note: OK= allowed, N/A = Not allowed

- **User Drawn Shapes, Shapes Created by Merging, and Holes:** These classes of shapes are created in several ways: by drawing on the screen, by modifying the library based shapes, merging more than one shape and performing other operations to achieve the desired shape.

Shape Editor

Complete information about a particular shape can be viewed and edited, including modifying rebar, using the Shape Editor. The Shape Editor can be accessed using one of the following methods:

- Click the **Edit menu > Edit Current Shape** command.
- Click the **Edit Current Shape** button .
- Double click the left mouse button on the shape to be edited.

The following describes some important features and functions of the Shape Editor:

- The **Shape dimensions** are visible on the graphic display/drawing area of the editor. The shape is shown with the correct orientation as shown/defined in the drawing area. The dimensions shown in the table just below the drawing area are editable; the coordinates are not editable. The graphic display will be updated to reflect changes in the table.



Note:

Coordinate points shown on the Shape Editor are for display only and CANNOT be changed for parametric and database shapes. They can be changed for user-defined shapes.

- The **Properties** area of the screen displays the properties computed for the shape based on the specified dimensions and material properties. When changes are made, click the **Accept Changes** button  on the toolbar to update or recompute the properties.
- **Shape Caption** is useful to identify a shape in this editor and also in the report. It can be changed by the user.
- **Main Material Type:** Click the drop-down list to select the main material type (Concrete or Hot Rolled Steel).



Note:

Shapes can have individual Shape Captions separate from the Section Caption.

- **Sub Material Type:** These are the subtypes of the main material type. Concrete and steel may have different grades. If Hot Rolled Steel is the main material type, the subtype can be ASTM-36, ASTM-50 or others.
- **Modulus of Elasticity (E):** This is the modulus of elasticity of the material. These values should be specified for each component of a composite Shape. The value specified applies only to the corresponding shape. This means that it is possible to input different E for different shapes forming a single section.
- Use the **Stress-Strain Curve** drop-down list to assign a stress-strain relationship to the current shape. Assigning a stress-strain curve to each shape in the section is important with respect to computing Section capacities, as explained in Chapter 5.
- Use the **Shape Type** checkboxes to specify that the current Shape be a solid or hollow part of the section. Use this option in conjunction with the **Edit menu > Align Shapes > Stacking** commands to create a section that has a hole. Note that the program does not check the validity of the hole's location or size.

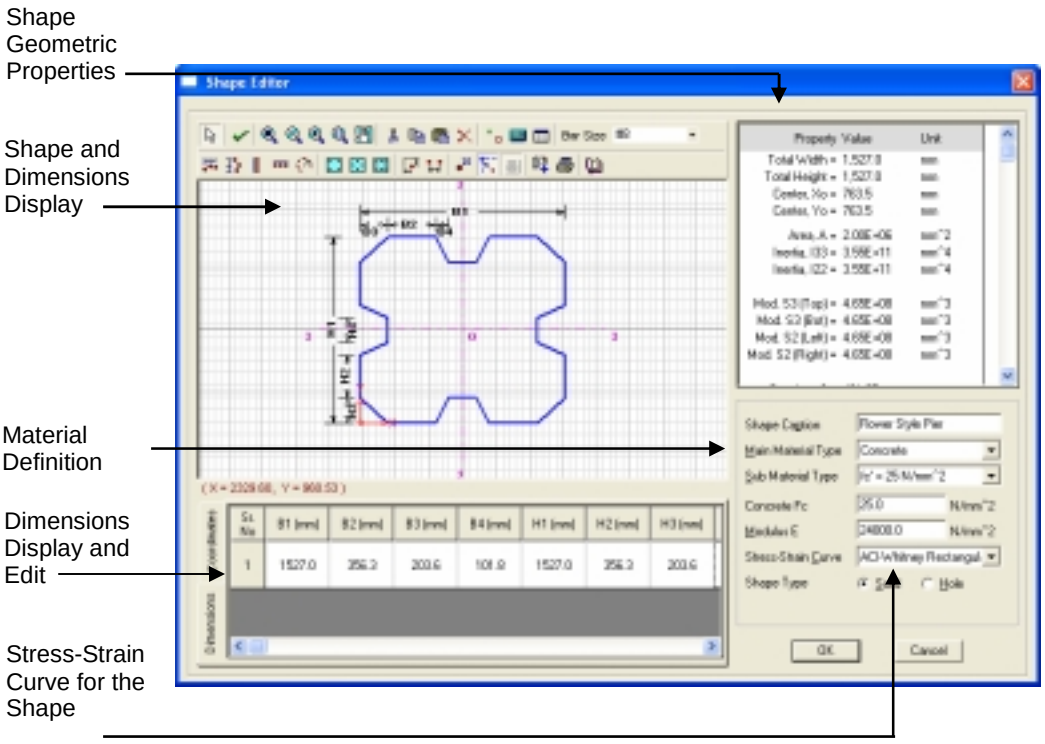


Figure 4-1 The Shape Editor

- **Dimensions/Coordinate Tab:** It is possible to switch between the overall dimensions of the shape and the actual X and Y coordinates of its nodal points. You can specify/modify the numeric values of each node on the Coordinate tab if the Shape is a user defined or merged Shape. You cannot change the coordinates of parametric and database Shapes.

Manage the Shape Editor Display Area

The Shape Editor has the following toolbar buttons that can be used to manage the display area on the Shape Editor, including selecting objects; updating the display; controlling zoom features; cutting, copying, and pasting shapes and rebar; displaying rebar dimensions; turning the grid and Snap to Grid option on and off; and adding the shape to the current report.



Note:

*The Shape Editor does **not** have an **Undo** button.*

Button	Function
	Sets the cursor to select mode so you can select items in the display area.
	Accepts the latest changes and updates the graphics and properties window. The Shape Editor window remains active.
	Restores the default view of the graphic display after you have zoomed in or out of the shape. This will reset the screen view in such a way that the entire shape is visible.
	This command allows you to zoom out on the column to see more of the column in the view.
	This command allows you to zoom in on the column to view details.
	This allows you to zoom in on the model by windowing. To use the command, depress and hold down the left button on your mouse. While keeping the left button depressed, drag the mouse to "rubber band" a window around the area of interest. The rubber band window that shows the extent you have dragged the mouse appears as a dashed line on your screen. When you release the mouse left button, the new view is displayed.
	The Pan feature moves a view within the window such that you can see beyond the original edges of the view. Panning may be used in conjunction with the Zoom In feature.
	Select rebar in the display area and click this button to remove it and place it on the Clipboard. The cut shape can then be pasted onto the Shape Editor display area. Hold down the Shift key while clicking on the rebar or use windowing to select multiple rebar.
	Select rebar in the display area and click this button to copy it to the Clipboard. The copied rebar can then be pasted within the Shape Editor display area. Hold down the Shift key while clicking on the rebar or use windowing to select multiple rebar.

Button	Function
	Click this button after using the Cut or Copy commands to Paste the rebar from the Clipboard into the Shape Editor display area. Note that CSICOL pastes the rebar into the center of the display area. Click the **Selection** button , click on the rebar, and drag the mouse to move the rebar to the desired location.
	Select rebar and click this button to delete it. Hold down the Shift key while clicking on the rebar or use windowing to select multiple rebar.
	Click to display/hide the rebar diameter/size captions.
	Click to turn Snap to Grid on and off.
	Click to display or hide gridlines in the display area.
	Click to add the current output (for the shape shown) to the report.
	Click to print this shape only.

Add Rebar at Mouse Clicks

Rebar can be added to a shape using the Add Rebar on Mouse Clicks feature as follows:

1. With a Shape displayed in the drawing area, access the Shape Editor using one of the following methods:
   - Select the **Edit menu > Edit Current Shape** command.
   - Click the **Edit Current Shape** button .
   - Double click on the Shape to be edited in the drawing area.
2. If necessary, select a new diameter from the drop-down list  on the Shape Editor toolbar.
3. Click the **Add Rebar on Mouse Clicks** button  on the Shape Editor toolbar.

4. Move the mouse pointer to the location on the shape where rebar is to be added.

5. Click the left mouse button. Continue adding rebar using additional left mouse clicks.

6. To exit the Add Rebar at Mouse Clicks mode, click the **Select Objects** button  on the Shape Editor toolbar.

Rebar Calculator

The Rebar Calculator can be used to easily add multiple rebar in a single operation, as follows.

1. With a Shape displayed in the drawing area, access the Shape Editor using one of the following methods:

   - Select the **Edit menu > Edit Current Shape** command.
   - Click the **Edit Current Shape** button .
   - Double click on the Shape to be edited in the drawing area.

2. Specify spacing of the rebar using the **Minimum Rebar Spacing** button  and the **Rebar Offset from Edge** button . Both buttons access input forms where you type in the desired value. The **Minimum Rebar Spacing** button specifies the minimum distance between the rebar. The **Rebar Offset from Edge** button specifies corner clearance.

3. Click the **Add Rebar Calculator** button  to access the Adding Rebars by List form, shown in Figure 4-2.

4. Click the numbers in the keypad area of the form to specify the number of rebar to be added. If you make a mistake, click the C (Clear) key on the form keypad.

5. Click the appropriate bar size number (e.g., #3, #4, #5) on the form keypad.

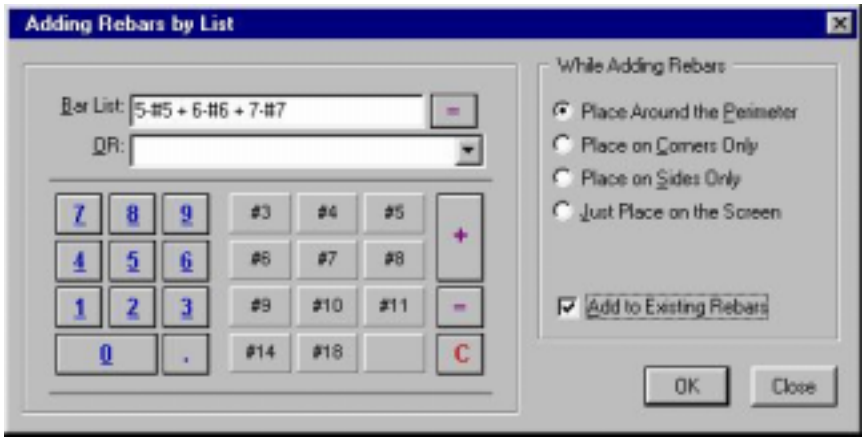


Figure 4-2: Add Rebar Using the Rebar Calculator

To add another group of rebar of a different size, click the + key on the keypad and repeat Steps 4 and 5. Notice that the specified numbers and sizes of rebar appear in the Bar List edit box.

6. When all rebar have been specified, click the = key on the keypad.

7. Check the desired checkbox in the While Adding Rebar area of the form to specify rebar alignment. The names of the options (Place Around the Perimeter, Place on Corners Only, Place on Sides Only, and Just Place on the Screen) are self explanatory.

   In the event that the current corner clearance and minimum bar spacing specified in Step 2 are incompatible with the alignment choice, CSICOL will place as many bars as possible in the specified alignment and then place the remaining bars adjacent to the Shape for manual placement. Depending on the number of rebar not aligned, it may be advantageous to delete the rebar that have been added and return to Step 2 to specify new values for the clearance and minimum spacing.

8. Indicate if the rebar are to be added to existing rebar (check the Add to Existing Rebar checkbox) or replace existing rebar (uncheck the Add to Existing Rebar checkbox).

Delete Rebar

The rebar already added on the Shape can be deleted at any stage as follows:

1. With a Shape displayed in the drawing area, access the Shape Editor using one of the following methods:
   - Select the **Edit menu > Edit Current Shape** command.
   - Click the **Edit Current Shape** button .
   - Double click on the Shape to be edited in the drawing area.
2. Select the rebar to be deleted by clicking on them or windowing over them in the display area of the Shape Editor.
3. Click the **Delete Selection** button  or the Delete key on the keyboard.
4. Click the **Accept Change** button  to update the properties.
5. Click the **OK** button to close the Shape Editor and return to the CSICOL drawing area.

Change Rebar Diameter

Change the diameter of the rebar in a shape as follows:

1. With a Shape displayed in the drawing area, access the Shape Editor using one of the following methods:
   - Select the **Edit menu > Edit Current Shape** command.
   - Click the **Edit Current Shape** button .
   - Double click on the Shape to be edited in the drawing area.
2. Select the rebar to be changed by clicking on them or windowing over them in the display area of the Shape Editor.
3. Select the new diameter from the drop-down list 

.
4. Click the **Accept Changes** button  to update the properties.

5. Click the **OK** button to close the Shape Editor and return to the CSICOL drawing area.

Align and Distribute Rebar

Tools on the Shape Editor can be used to align rebar for a section as follows:

1. With a Shape displayed in the drawing area, access the Shape Editor using one of the following methods:
   - Select the **Edit menu > Edit Current Shape** command.
   - Click the **Edit Current Shape** button .
   - Double click on the Shape to be edited in the drawing area.
2. Select the rebar to be aligned by clicking on them or windowing over them in the display area of the Shape Editor.
3. Click the appropriate Shape Editor toolbar button to align or distribute the selected rebar as required.



Note:

In the event that the current corner clearance and minimum bar spacing are incompatible with the alignment choice, CSICOL will place as many bars as possible in the specified alignment and then place the remaining bars adjacent to the shape for manual placement.


Distributes the selected rebar horizontally at equal spacing.


Distributes the selected rebar vertically at equal spacing.


Aligns the selected rebar vertically so that a vertical line could pass through their midpoints.


Aligns the selected rebar horizontally so that a horizontal line could pass through their midpoints.


Distributes the selected rebar to the shape's corners.


Distributes the selected rebar at the shape's sides.


Distributes the selected rebar at along the shape's perimeter.


Accesses the Distribute Bars in Arc form. Use the form to provide the center, radius and the start and end angle of the arc. The coordinates of the center are with respect to

the local coordinate system of the Shape for which the bars are being distributed. The origin of this local coordinate system coincides with the bottom left corner of the shape. CSICOL will distribute the selected bars considering the minimum bar spacing specified. If the number of bars cannot fit in the specified arc, CSICOL will not distribute the additional bars.

Edit Shape Point Coordinates

The nodal point coordinates of a selected shape can be edited and nodes can be added to or removed from a selected shape using the **Edit menu >Edit Shape Coordinates** command or the **Edit Shape Point Coordinate** button .

1. Select the shape whose nodal coordinates are to be modified. If the shape is a parametric shape, CSICOL will prompt you to change it to an editable polygon.

2. Use the **Edit menu > Edit Shape Coordinates** command or the **Edit Shape Point Coordinate** button  to access the Define Shape by Points form. The coordinates and the number of points of the selected shape along with the sketch of the shape will be displayed on the form.

   - Use the **Remove** button to delete a selected shape point.
   - Use the **Insert** button to insert a shape point at a selected location.
   - Use the **Export** button to export the coordinates of the shape to various file types (spaced text file, comma-separated text file, tab-separated text file or .DXF file).
   - Use the **Import** button to import coordinates of the shape from input files of various types (spaced text file, comma-separated text file, tab-separated text file, .DXF file).

3. Click the **OK** button to apply the changes to the shape displayed in the drawing area.



Note:

The coordinates to be imported must be provided with respect to the Global Coordinate System.

Shape Layout Editor

In several situations, it may be convenient to align shapes graphically first and then refine their placement numerically using the Shape Layout Editor.

Access the Shape Layout Editor using the **Edit menu > Locate Shape** command or by clicking the **Shape Layout Editor** button . The Editor, which is shown in Figure 4-3, displays the number of Shapes and their coordinate locations in the current section based on global coordinate values.

Note:

Right-clicking on the cells of the Shape Layout Editor will display context-sensitive toolbars with options for selecting, copying, cutting, pasting and clearing a selection.

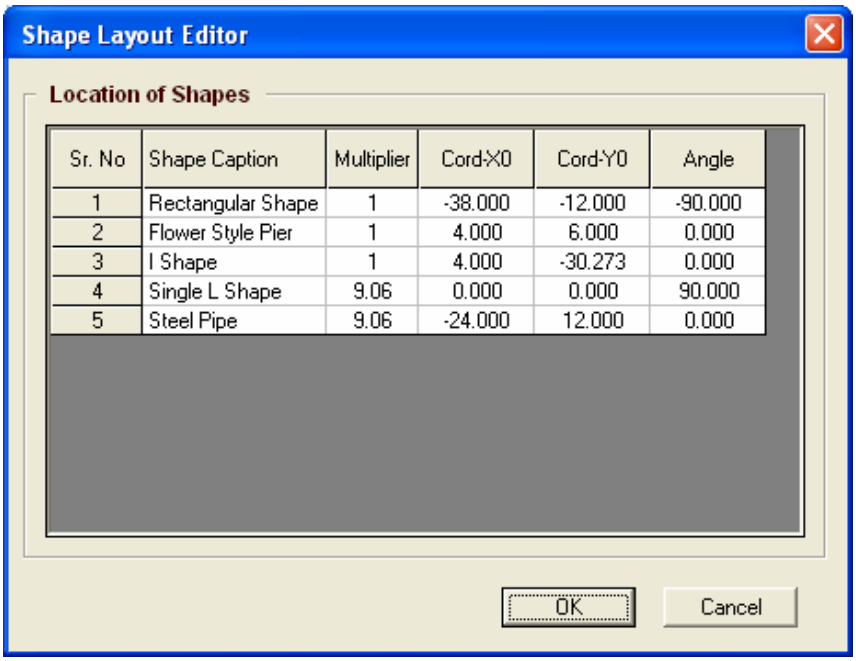


Figure 4-3: Shape Layout Editor

The various areas on this Editor are described as follows:

- **Sr/No:** The serial number of the shape; it cannot be edited.
- **Shape Caption:** The name of the shape; it can be edited.



Note:

After a Shape has been rotated or flipped, some of the editing features normally available in the Shape Layout Editor may no longer be available.

- **Multiplier:** The property multiplier, or the modular ratio of the shape and the base material.
- **Cord-X0:** The X-coordinate of the center of the shape; it can be edited to move the shape to a specific position along the X-axis. The X coordinate is in respect to the coordinate origin shown as the X –Y coordinates in the drawing area (not the 2-3 system).
- **Cord-Y0:** The Y-coordinate of the center of the shape; it can be edited to move the shape to a specific position along the Y-axis. The Y coordinate is in respect to the coordinate origin shown as the X –Y coordinates in the drawing area (not the 2-3 system).
- **Angle:** The angle of the shape with respect to the X-axis measured in the counter-clockwise direction. The angle can be specified to align a shape at any orientation between 0 and 360 degrees. Vertical or horizontal orientation of a shape can be set or changed using the Rotate and Flip toolbar buttons or Edit menu commands.

Align Shapes Graphically

In graphical alignment, no text or numeric input is required from the user. Align shapes graphically as follows:

1. Select the shapes to be aligned
2. Click one of the alignment toolbar buttons or select the **Edit menu > Align Shapes** command and the appropriate option. See the *Toolbar Buttons and Menu Commands* section in Chapter 2 for a listing of the available buttons and commands.

Graphic alignment in CSICOL is performed on the basis of the first shape selected, in the order of selection.

Rotate, Flip, and Stack Shapes

Rotate, flip or stack shapes as follows:

1. Select the shapes to be rotated, flipped, or stacked.
2. Click one of the rotate, flip or stack toolbar buttons or select the appropriate option after using the **Edit menu > Rotate and Flip Shapes** command. See the *Toolbar Buttons and Menu Commands* section in Chapter 2 for a listing of the available button and commands.



Note:

Stacking is performed on the basis of the relative location of the shapes in the order of selection.

Two important notes about rotating and flipping:

- After a shape has been rotated or flipped, some of the other editing features may not work effectively.
- If you want to rotate and flip the shape, rotate it first and then flip it. The opposite order may create some unexpected results.

Merge Shapes

Two editable shapes (see Table 4-1) with the same material properties that have a common edge or that overlap can be merged, with the following exceptions.

- Holes cannot be merged with solid shapes.
- Holes cannot be merged with holes.
- No more than two shapes can be merged at a time.

Merge two shapes as follows:

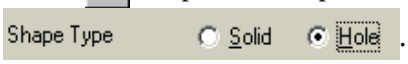
1. Select the two shapes to be merged.
2. Click the **Edit menu > Merge Two Shapes > Merge Using Meshing Logic** command or the **Merge Shapes Using Meshing Logic** button or the **Edit menu > Merge Shapes Using Intersection Logic** command or the **Merge Shapes Using Outline Logic** button .
3. Verify that the two shapes have been successfully merged by displaying the shape outline ( ) or as a filled Shape ( ).

Tip:

If one merge command does not provide the desired result, Undo the merge and try the other command.

Create Holes in a Shape

Use the following procedure to create a hole in a shape or section:

1. Add a shape (first shape). By default, the Shape Type is Solid until you change it.
2. Add a shape whose size is equal to the size of the hole (second shape).
3. Move the second shape to the place where the hole is to be created (generally inside the first shape). To move the shape, click the **Select to Reshape** button , click on the shape and hold down the left mouse button as you drag the shape to the desired location.
4. Double click on the second shape or use the **Edit Current Shape** button  to open the Shape Editor. Set the Shape Type to Hole .
5. Click the **OK** button to accept the change and return to the drawing area.

Verify that the hole has been created properly by displaying the shape as a filled shape ().

Move Shapes

Selected shapes in the drawing area can be moved using the mouse, the arrow keys on the keyboard, or the **Edit menu** > **Move Selection** command and the **Move Selection** button .

Move an individual shape using the mouse as follows:

1. Click the **Select to Reshape** button  to activate Reshaper mode.
2. Select a single shape to be moved.

3. Hold down the left mouse button and drag the mouse to move the shape to the desired location.

Move a shape using the arrow keys on the keyboard as follows:

1. Click the **Select Object** button  to activate Selection mode.
2. Select the shape(s) to be moved.
3. With the shape(s) selected, use the appropriate arrow key on the keyboard to move the shape to the desired location.

Move a shape(s) using the **Edit menu** > **Move Selection** command or the **Move Selection** button  as follows:

1. Click the **Select Object** button  to activate Selection mode.
2. Select the shape(s) to be moved.
3. Use the **Edit menu** > **Move Selection** command or click on the **Move Selection** button  to access the Move Selection form.
4. On the Move Selection form provide the displacement values in X and Y directions.
5. Click the **OK** button.

Add Fillets to Shapes

Fillets of user-specified radii can be added to selected shapes. The data required to add a fillet to a shape is the point where you want to add the fillet, the radius of the fillet and the number of points on the curve of the fillet. Add a fillet to a shape as follows:

1. Select the shape to which the fillet is to be added.
2. Click on the **Add Fillet at Selected Point** button  to activate the Add Fillet at Shape Point form. The coordinate points (nodal points) of the selected shape will be displayed on the drawing area.

If the selected shape is a parametric or a database shape, the program will prompt you to change it to an editable polygon.

3. From the drop-down list, select the node point to which the fillet is to be added.

4. Specify the radius of the fillet to be added and the number of points on the curve of the fillet. The number of points will determine the smoothness of the curve. More points will result in a smoother curve. The program will automatically calculate the included angle at the selected point for the given radius.

5. Click the **OK** button. The specified fillet will be added to the selected shape in the drawing area.



Chapter 5

Obtain and Interpret Results

Overview

In this chapter, it has been assumed that the user is familiar with the basic concepts of column design and analysis, structural concrete mechanics (especially the structural interpretation of the design parameters) and structural analysis results. The main topics presented in this chapter address the following:

- Generate interaction curves and surfaces.
- Check the adequacy or capacity of a column section for different load combinations
- Plot the combined stresses resulting from various load combinations
- Plot moment-curvature curves
- Display geometric properties and other results

Interaction Surface and Curves

Three stress resultants (axial load, moment about x and moment about y) can be determined for a particular strain profile on a cross-section. As the strain profile is varied or changed, the values of the stress resultants vary. The three stress resultants can be plotted in a 3D space to generate a continuous surface for all possible variations of the strain profile on a particular cross-section. This is generally known as the stress resultant Interaction Surface.

In addition, if the strain profiles used to generate the surface are derived from material failure conditions, this surface becomes the "Capacity Surface" or the "Failure Surface." Any combination of applied actions P, M_x and M_y that is inside the volume enclosed by this surface is safe, whereas any combination that results in a point that is outside this surface is considered unsafe. As the interaction surface exists in three-dimensional space, it cannot be plotted on a two-dimensional paper space directly. The interaction surface can however be converted to two-dimensional curves by appropriate "slicing" of the surface. The two most common types of curves derived from the interaction surface are:

- Load Moment Curve
- Moment-Moment Curve

If the capacity surface is sliced vertically, along any angle about the origin, we obtain a plot between the resultant moment and the axial stress-resultant, often termed the P-M interaction curve. This is a very common and useful tool for the design and investigation of columns. Special P-M curves can be obtained for a slice of the capacity surface along the x and y axes.

The capacity surface can be sliced on the xy plane to obtain the plot between moments M_x and M_y. This plot between the moment capacity about the x and y axes provides several useful insights into the behavior of the cross-sections. It shows how the moment capacity varies around the cross-section at a particular load level.

Interaction Diagrams

The variation of all three parameters governing the section capacity of a column section can be plotted in a single 3D graph. This gives an overall picture of the variation of all three parameters for the given column section. P is plotted along the vertical axis and M_x and M_y along two orthogonal horizontal axes.

Generate the Interaction Surface for any given column cross-section by clicking the **Show Interaction Diagrams** button  or using the **Display menu > Show Interaction Diagrams** command. Display the interaction surface and curves of a column section as follows:

1. Define the column (geometry and material properties) for which you want to generate the surface (see Chapter 3 for more information).

2. Use the **Display menu > Show Interaction Diagrams** command or click the **Show Interaction Diagrams** button  to access the Interaction Diagrams form. The form consists of the following tabs:

   - **P-M Curve:** P-M curves are outlines obtained when a vertical plane cuts the interaction surface at different specified section rotations showing the relation between the axial load and the moment at the specified neutral axis rotation. The P-M curve is the default view on the Interaction Diagrams form. Select the neutral axis angle for which you want to view the P-M Curve. The curve display will be updated automatically. The following toolbar is available on this tab:

      Accepts changes and refreshes display.

     Drop-down list that displays the neutral axis angle for the currently displayed P-M curve. It also displays the user-specified load combinations for capacity checks.

      Copies the currently displayed view to the Clipboard.

      Displays the Tabulated Output or the curve generation points of the curve on display. See the *P-M and M-M Curve Tabulated Output* section later in

this chapter for more information.



Adds the currently displayed Interaction Curve to a Report.

- **M-M Curve:** The M-M curves are the horizontal cut surface outlines at certain heights along the vertical P axis (specified values of axial load) in P-M_x-M_y surface plots. To view the M-M curves, click on the M-M Curve tab on the Interaction Diagrams form. Select the axial load level for which you want to view the M-M Curve. The curve display will be updated automatically. The following toolbar is available on this tab:



Accepts changes and refreshes display.



Drop-down list that displays the load level for the currently displayed M-M curve. It also displays the user-specified load combinations for capacity checks.



Copies the currently displayed view to the Clipboard.



Displays the Tabulated Output or the curve generation points of the curve on display. See the *P-M and M-M Curve Tabulated Output* section later in this chapter for more information.



Adds the currently displayed Interaction Curve to Report.

- **Interaction Surface:** The interaction surface plots the variation of axial load and moment in both directions for a cross-section. To view the interaction surface, click on the Interaction Surface tab on the Interaction Diagrams form. The following toolbars are available on this tab:



Sets the interaction surface display color for P, M_x or M_y. Can also set the surface display as opaque or transparent.



Refreshes the Display Area view.



Changes the view angle according to mouse movements when the left mouse button is clicked and the mouse is dragged across the display area.



Pans the view when the left mouse button is clicked and the mouse is dragged across the display area..



Zooms in and out dynamically with left and right mouse clicks respectively.



Sets the Interaction Surface View Options such as surface color, lighting, animation and background. See the *Interaction Surface View Options* section later in this chapter for more information.



Copies the currently displayed view to the Clipboard.



Adds the current view to a Report.

P-M and M-M Curve Tabulated Output

The tabulated output can be viewed for interaction curves and moment-curvature diagrams. These can then be saved in various file formats (comma-separated text files, tab-separated text files, or spaced text files). Those tabulated outputs can also be copied to the Windows system clipboard or added to the report.

Access the tabulated output form as follows:

- On the Interaction Diagrams form, click on the **Tabulated Output** button .
- On the Moment-Curvature Diagram form, click on the **Curve Points** button.

Save the output as a text file as follows:

1. Activate the Tabulated Output form.
2. Click the **File menu > Save As Text File** command on the form.
3. Select the type of Text File in which to save the output.

4. Specify the Text File name and click the **Save** button.

Add the output to the report as follows:

1. Access the Tabulated Output form.
2. Click on the **File menu > Add to Report** command on the form.

Copy the output to the system clipboard as follows:

1. Access the Tabulated Output form.
2. Click on the **Edit menu > Copy to Clipboard** command or the **Edit menu > Copy Selection** command on the form.
3. Use the Windows-based Paste command (Ctrl V) to paste the copied output into another program.

Interaction Surface View Options

Click the **Surface View Option** button  on the Interaction Surface tab to access the Interaction Surface View Options form and set the view settings for the interaction surface. The form is shown in Figure 5-1.

The Interaction Surface View Option form consists of the following three tabs:

- **Display Objects:** Use this tab to set the various options related to visibility, finish, coloring, background and transparency for the Interaction Surface Display area. The components that can be customized include the main surface, the grid planes, axis vectors, axis planes, axial load plane, cross-section, main column, attached columns and beams in the x and y planes.
- **Lights:** Use this tab to set options for lighting intensity, direction, type and color. A total of seven light types are available and can be customized for position, direction, color, and attenuation. Each light type can be switched on or off using the "Light is On"'option on the form.

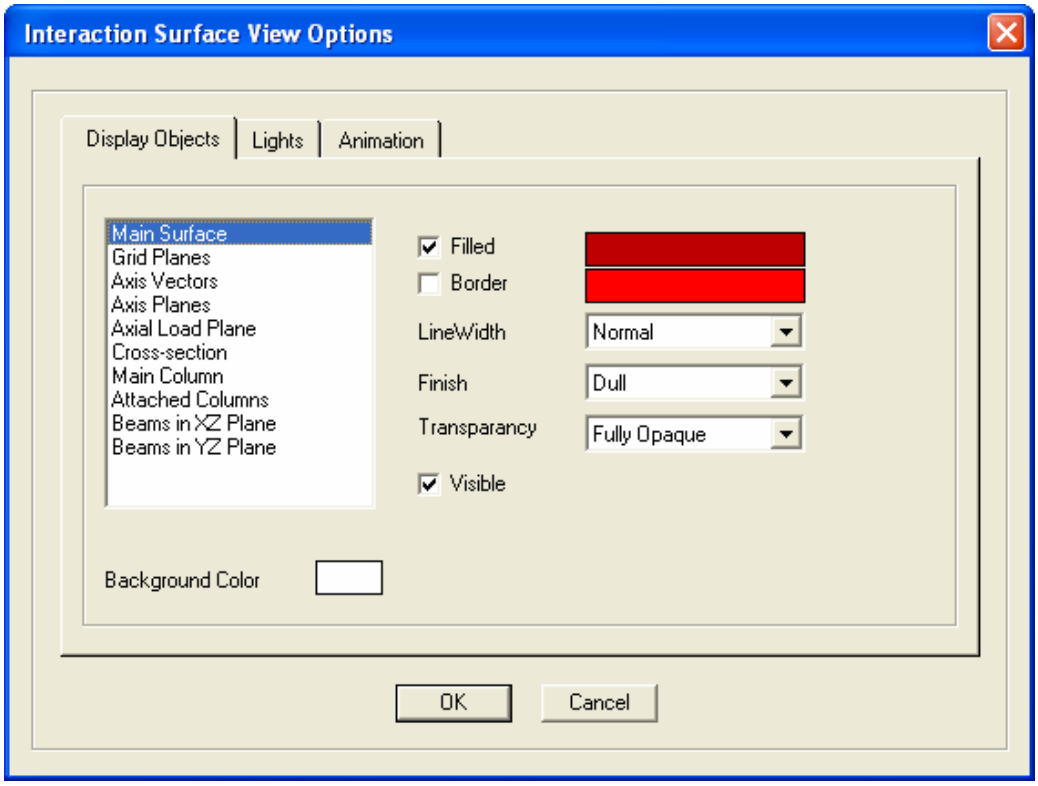


Figure 5-1: Interaction Surface View Options

- **Animation:** Use this tab to animate the object currently displayed on the Interaction Surface Display area. Checking the "Animated Display" option on the form will activate various options, including rotate the display about the x, y or z axis and zoom in and out while rotating. The animation speed can be set to slow, medium or fast using the drop-down menu available on the form.

Capacity Calculations

After a section has been defined (geometry and material properties), it is checked against all specified load combinations for adequacy. The results are displayed in terms of capacity ratio on the Capacity Calculation Result form. Results are displayed separately for the top and bottom ends and for combined effects in both directions of the column. The form is displayed in Figure 5-2.

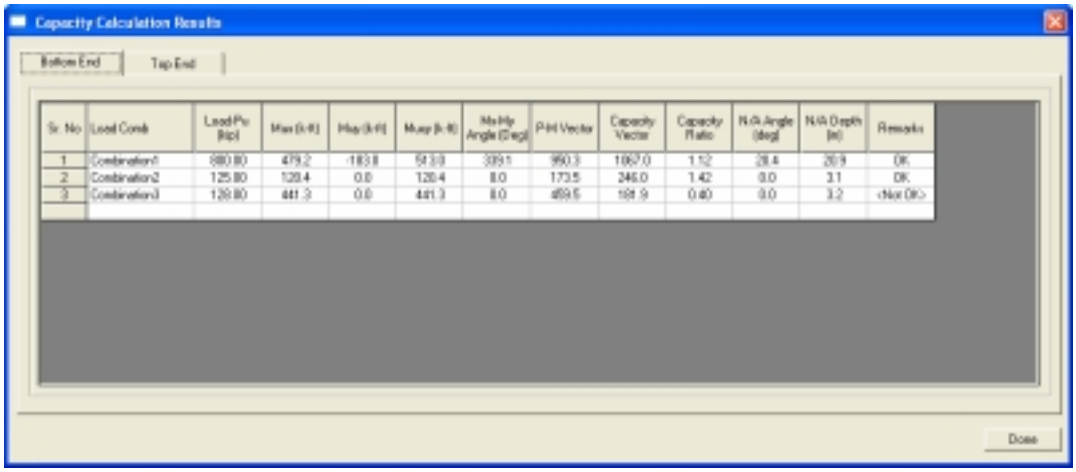


Figure 5-2: Capacity Calculation Results

Display the capacity calculation ratio as follows:

1. Define the column cross-section for which you want to check the capacity.

2. Click the **View Capacity Calculation Results** button  to activate the Capacity Calculation Result form. The form may also be activated using the **Result menu > Column Capacity Ratio** command.

The results are displayed for the top and bottom ends of the column. Click the appropriate tab to view the results.

The form displays all of the load combinations defined for the column. In addition, the M-M vector angle, P-M vector length, Capacity Vector, Capacity ratio, neutral axis depth and orientation, and a statement of cross-section adequacy are displayed. For capacity ratios greater than one, the program reports the section as inadequate.

Section Stresses

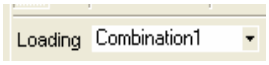
CSICOL displays three types of stress on the section: Combined Elastic Stresses, Cracked Section Stresses on the concrete area, and the Rebar stresses for each load combination. Each type of stress is displayed

independently over the column section on the Stress Viewer form. The stresses are displayed as 2D and 3D color-coded contours.

Display stresses on a column section for various load combinations as follows:

1. Create the column section for which you want to view the Stresses (see Chapter 3 for more information).
2. Define the load combinations for the section (see Chapter 3 for more information).
3. Click on the **Show Cross-Section Stresses** button  or use the **Display menu > Show Cross-Section Stresses** command to access the Stress Viewer.
4. On the Stress Viewer, select the 2D View or the 3D View tab.
5. Select the Load Combination and the column end (upper or lower) for which you want to view the stresses.
6. Select the stress type (Elastic, Cracked or Rebar) that you want to view on the Section by clicking on the appropriate button.

Each type of stress is displayed independently over the column section. A color-coded key is displayed at the bottom of the form to give an idea of the stresses generated on the section. In addition to these stress displays, you can view the Load Point and Neutral Axis Location in 3D on this form. The following toolbar is available on the form:



Drop-down list of the load combination for which different displays will be generated.



Displays output for loading at the upper end of the column.



Displays output for loading at the lower end of the column.



Displays/hides the load point.


Displays/hides the neutral axis location and orientation on the column section for the selected load combination case.


Displays/hides the combined elastic stresses in 3D on the column section for the selected load combination case.


Displays/hides cracked stresses in 3D on the column section for the selected load combination case.


Displays/hides rebar stresses in 3D on the column section for the selected load combination case.


Refreshes the view of the display area.


Changes the view angle with movement of the mouse while the left mouse button is held down over the display area.


Pans the view.


Zooms in and out dynamically with left and right mouse clicks, respectively.


Adds the currently displayed view to the report.


Copies the currently displayed view to the clipboard.

The 3D Stress View form is shown in Figure 5-3.

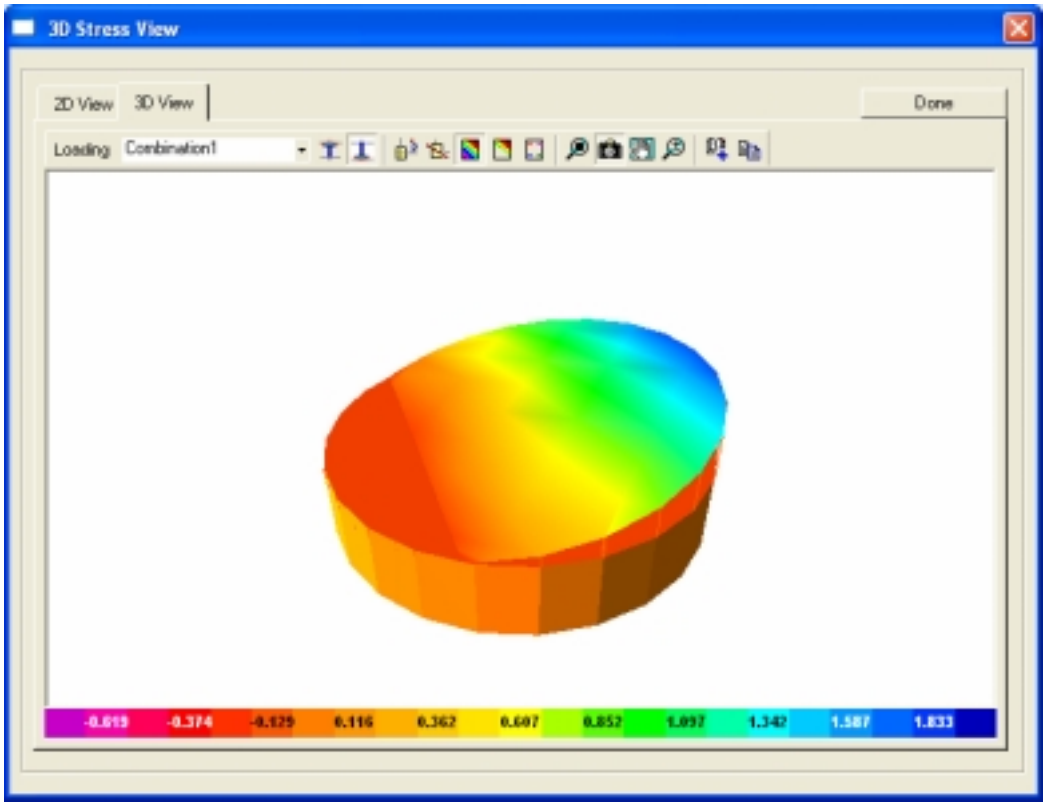


Figure 5-3: 3D Stress View Form Displaying 3D Elastic Stress on a Circular Section

Moment Curvature Curves

The program is capable of plotting moment-curvature curves for a section. Click the **Moment-Curvature Curves** button to activate the Moment Curvature Diagram window, which is shown in Figure 5-4. Moment-curvature curves can be plotted for any user specified axis. The moment-curvature for a given axial load can be plotted by specifying the axial load value and clicking the **Recompute** button. The curves are primarily intended for Reinforced Concrete Sections. The value of axial load, including zero, can be specified. Similar to the interaction curves, the moment-curvature curves for known (user specified) neutral axis angles can be generated. Generation of the moment curvature curve is controlled by several options available on the form. Those options

include edit boxes that allow entry of values for moment angle, axial load, and maximum strain as well as selecting options to specify when computing will stop: when the maximum strain has been reached; when any part of the section fails; when all parts of the section fails; when the first rebar fails; or when a selected part of the section fails—select the specific part of the section from the available drop-down list.

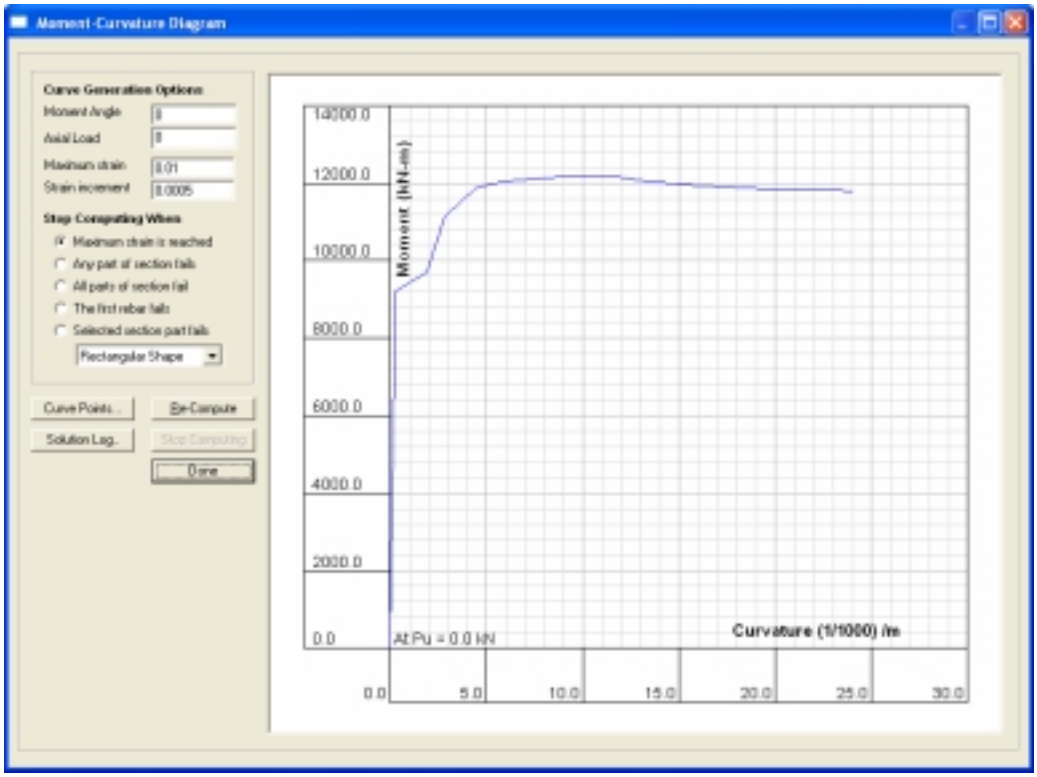


Figure 5-4: Generating Moment-Curvature Curves

Geometric Properties

To view a summary of the overall dimensions and geometric properties of the current column Section in the drawing area, select the **Display menu > Show Geometric Properties** command or click the **Show Geometric Properties** button to access the Geometric Properties form. This form displays the following for all sections:

- **Overall Dimensions**: The overall dimensions of the section and the centroid location with respect to the global origin.
- **Basic Properties.** The area, moment of inertia about 2-3 axis, elastic section modulus and radii of gyration.
- **Principal Properties.** Principal moments of inertia and corresponding Principal angles.
- **Additional Properties.** Torsional constant, shear areas and plastic section moduli.
- **Global Properties.** These properties are calculated about the Global XY axis and are dependent on the location of the section with respect to the origin. They include the moment of inertia and the first moment of areas.

Other Results

In addition to the results described previously, CSICOL reports the Load Point location and the neutral axis location and orientation. Those results are reported for the upper and lower ends of the column. The load point location depends on the direction of the eccentricities in the x and y directions. Figure 5-5 shows the direction of the eccentricity vector and the load point on the Section. The direction of eccentricities is also shown.

The corresponding load point as shown in Figure 5-5 is represented on the M-M curve and shown in Figure 5-6, and its direction and location is represented by the applied load vector. The representation of the positive and negative directions of moments in both the x and y directions are also shown. The angles are measured from the positive M_x axis and clockwise directions are considered as positive.

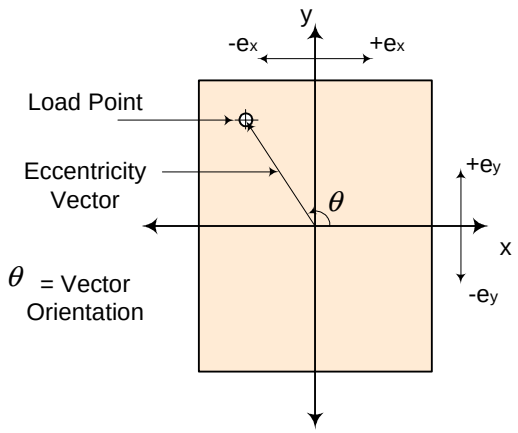
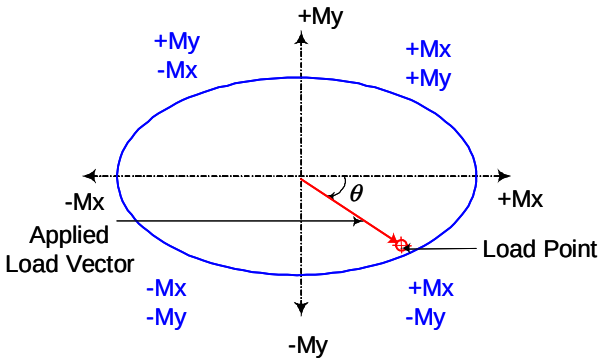


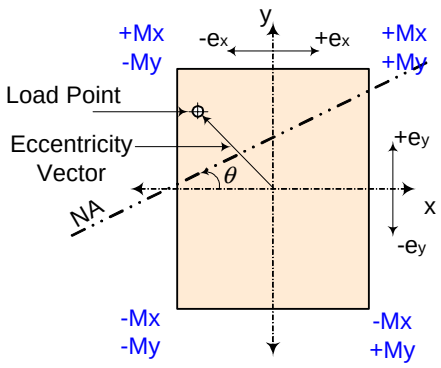
Figure 5-5: Representation of Load Point and Eccentricity Vector on the Section



Moment Directions on the M-M Curve

Figure 5-6: Representation of Load Point and Applied Load Vector on the M-M Curve

The orientation of the neutral axis is shown in Figure 5-7 for the same load point location and eccentricity vector. The angle of the neutral axis is measured from the positive x axis and is considered positive for counter-clockwise rotations.



Moment Directions on the Cross-Section

Figure 5-7: Orientation of the Neutral Axis on the Section

Generate a Report

Overview

This chapter describes creating, previewing, and printing a report. The report automatically takes into consideration the currently selected code and updates the reporting accordingly. Therefore, the content of the report for various codes may differ, but the procedure for creating the report remains unchanged.

Create a Report

Click the **File menu > Report Creation Wizard** command or the **Report Creation Wizard** button , to access the Report Creation Wizard, which is a step-by-step guide through the reporting process.

Note: If you are using the Quick Design Wizard, simply click the **Generate Report** button on the Quick Design Wizard form to access the Report Creation Wizard. Select the items to be included in the report using the Report Creation Wizard form and click the **Generate Report** button. CSICOL will automatically display a preview of the report.

The first screen of the Report Creation Wizard shows different options for selecting items to be included in the report. The left list shows the number of Sections defined in the current file. The right list displays the output items available for printing. It is possible to select different items for different sections. For example, items such as project information, calculation procedures, and so forth can be selected for the first section, and then for the remaining sections, you can choose to print only the results, assuming that the project information, calculation procedures and so on are consistent across all sections in the project file.

After the required items have been added/selected, click the **Generate Report** button to finish the report generation process.

Preview a Report



Note:

If a report has not been generated previously, the Report Creation Wizard will appear. Choose the items to be included in the report, generate the report, and then choose Yes when prompted to preview the report.

Click the **File menu > Preview Current Report** command or the **Preview Current Report** button  to preview the report before sending it for final printing.

Figure 6-1 shows a preview of a sample report. Various navigational buttons (first page, next page, page up / down and so forth) are provided to view different report pages. Clicking the left and right mouse button on the preview window will zoom in and out of the view respectively.

To print the report, click the **Print** button in the upper left-hand corner of the report previewer.

To save the report, click the **Save As** button in the upper left-hand corner of the report previewer.

To close the previewer without printing, click the **Close** button near the middle of the toolbar at the top of the screen.

A secondary toolbar is provided at the top left of the screen. This toolbar can be used to select a page to view and also to change the display mode. Various options are available for setting the display scale, including zooming to 25%, 50%…150%, and showing a single page or multiple pages.

Add to the Report

After previewing the report, additional graphics and data can be added to the report before printing as follows:

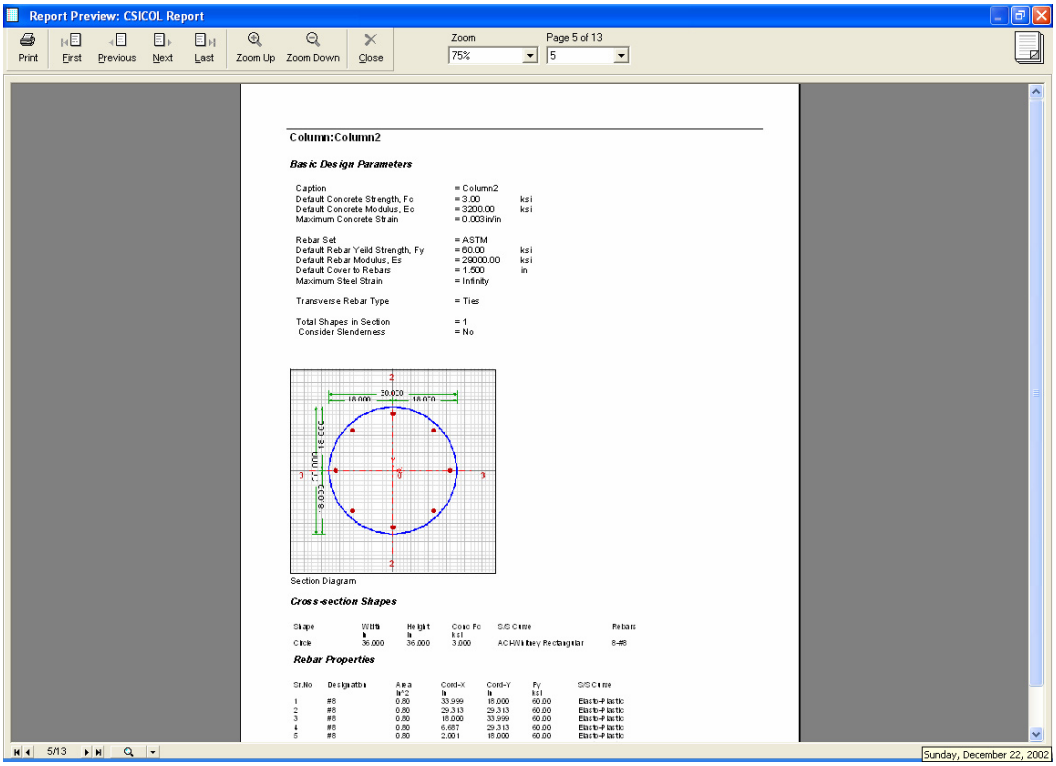


Figure 6-1: Previewing the Report

1. Use the Section List drop-down list to display, in the current working area, the section to be added to the report.

2. Click the **File menu > Add Section View to Report** command or the **Add Current View to the Report** button .

Also note that the Shape Editor and the various forms used to display results have **Add Current View to the Report** buttons . Thus, to add a view of a specific shape or rebar distribution to the current report, access the Shape Editor (see Chapter 3 for more information) and click

the **Add Current View to the Report** button  on the Shape Editor. Similarly, to add results, use the commands available on the Display menu to access the desired results and click the **Add Current View to the Report** button  on the associated form (see Chapter 5 for more information).

Print a Report

Before a report can be printed, it must be generated using the Report Creation Wizard, which is described in the *Create a Report* section of this chapter.

A generated report can be printed by clicking the **Print** button on the Report Preview window, which is described in the *Preview a Report* section of this chapter.

A generated report can also be printed using the **File menu > Print Current Report** command.

In addition, any graphic view displayed on the drawing area can be sent for printing by selecting the **File menu > Print Current Column** command or the **Print Current Column** button . This is useful for quick printing of the current section without using the Report Creation Wizard.

Save a Report

A previously generated report can be saved using the **Save As** button on the Report Preview form. Access the Report Preview form using the **File menu > Preview Report** command or in conjunction with the use of the Report Creation Wizard, which can be used to generate to report (see the *Create a Report* section for more information).

The **Save As** button will access the Save and Export Report form. The form can be used to save the report as a text file or to export the file to another file format.

Save Report as Text

Use the ... button and the File Name edit box on the Save and Export Report form to specify the filename and path for saving the file.

Choose the text file format for saving the report by selecting the appropriate option: Save as Simple Text or Save As Text Separate. As appropriate, use the drop-down list to specify that the text file be comma, space, tab or slash separated.

Click the **Save** button to save the file as specified.

Click the **Done** button to close the form and return to the CSI Preview Report: CSI Report form.

Export Report

Use the ... button and the File Name edit box to specify the filename and path for saving the file.

Use the drop-down list to choose the file format for exporting the report (e.g., MS Excel File, MS Word File, RTF File, and HTML or DHTML file for the web).

Click the **Save** button to save the file as specified.

Click the **Done** button to close the form and return to the CSI Preview Report: CSI Report form.